NDT SetupBuilder

User's Manual

Software Version 1.1

DMTA-20090-01EN — Rev. B
September 2022

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

Keep this instruction manual in a safe, accessible location.

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

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 1.1
Part number: DMTA-20090-01EN
Rev. B
September 2022

Printed in Canada

Table of Contents

List of Abbreviations

AUT	automated ultrasonic testing
CAD	computer-aided design
ED	electronic delay
EMAT	electro-magnetic acoustic transducer
GB	gigabyte
GD	global delay
ID	inside diameter
LCP	lack of cross-penetration
LD	law delay
LHS	left-hand side
LW	longitudinal wave
MB	megabyte
NDT	non destructive testing
OD	outside diameter
PA	phased array
PCS	probe center separation
RAM	random-access memory
RHS	right-hand side
SW	shear wave
TOFD	time-of-flight diffraction
USB	universal serial bus
UT	ultrasonic testing
WD	wedge delay

Important Information — Please Read Before Use

Carefully read the following important information before installing the NDT SetupBuilder software and configuring the inspection setups.

Intended Use

The NDT SetupBuilder software is designed to create inspection setups for nondestructive ultrasonic inspections of industrial and commercial materials. NDT SetupBuilder is used to calculate phased array, ultrasound, and time-of-flight diffraction probe-element delays for Evident OmniScan instruments.

Software Compatibility

NDT SetupBuilder is compatible with OmniScan MX2 and OmniScan SX flaw detectors that run on MXU software version 3.2 or higher.

Instruction Manual

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

Safety Symbols

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

 General warning symbol

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

 Shock hazard caution symbol

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

Safety Signal Words

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

 **DANGER**

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

 **WARNING**

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

Note Signal Words

The following 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.

Introduction

NDT SetupBuilder software is complementary software for Evident automated ultrasonic testing (AUT) products. This software is designed to assist nondestructive testing (NDT) personnel by reducing the time required to define the appropriate inspection configuration for OmniScan instruments, ensuring that the configuration is compliant with the inspection code, and eliminating setup configuration errors.

NDT SetupBuilder software helps NDT supervisors define inspection techniques on a computer, enabling them to visualize how ultrasonic beams behave in the part to be inspected so that they can determine the appropriate number of beams and angles. This allows supervisors to define the inspection technique at the office or in the field, and more importantly, without the need for an actual inspection unit.

NDT SetupBuilder features a flexible graphical user interface that allows easy visualization of the inspected area, which helps make sure that it is 100 % covered by beams. Generic graphical features such as view-type selection, probe-set parameter adjustment (a few of the parameters are interactively adjustable), zoom tools, and measurement cursors are available.

NDT SetupBuilder runs on the Microsoft Windows 7 (32-bit and 64-bit), and Windows 8 operating systems.

1. Software Overview

NDT SetupBuilder software provides you with the ability to configure your inspection setups for OmniScan acquisition units without the need for any connected instrument.

1.1 NDT SetupBuilder Features

NDT SetupBuilder allows you to select the type of OmniScan acquisition unit for an inspection and define the part to be inspected. You can create weld overlays, configure the ultrasonic testing (UT) parameters, and define linear, sectorial, compound, or pitch-catch groups. (Pitch-catch groups are only available for time-of-flight diffraction [TOFD] and dual matrix groups.) These groups are defined according to the chosen OmniScan instrument's capabilities. The software can compute the attenuation, export to the acquisition software, and generate setup reports. Additionally, NDT SetupBuilder is able to calculate delays that cannot be calculated on the OmniScan instrument. NDT SetupBuilder can import .law files to enable inspections with configurations that exceed the OmniScan instrument's built-in capabilities.

NDT SetupBuilder software makes it easy to build inspection setups from scratch. The application helps you precisely program angle beams and focal points. Parameter entry is easy, because a large array of probe, wedge, and material data is preprogrammed into the database. Predefined part, probe, wedge, and group parameters can be modified to create custom parts, probes, and wedges. These options provide practically infinite possibilities when creating inspection setups.

In addition, the resulting groups and angle beams are displayed as a graphical representation, which is an important validation tool for inspection setups on OmniScan instruments.

You can verify whether the theoretical configuration of probes and groups (respectively named *probe sets* and *groups* in NDT SetupBuilder) can be supported by your OmniScan acquisition units and models. To summarize, the software performs the following functions:

- Creates weld overlays
- Sets UT position parameters
- Defines linear, sector, compound, TOFD, and specific dual matrix groups
- For exported .law files, automatically computes focal laws, which are called *beams* in NDT SetupBuilder
- Generates setup reports
- Exports the inspection setups to the OmniScan acquisition software

---

NOTE

For exported connectivity files, the entire setup configuration must be verified and computed after it has been imported into an OmniScan instrument.

The NDT SetupBuilder software's graphical user interface provides a visual representation for the following adjustments:

- Part and weld overlay definition
- Probe sets definition: Build series of probe sets for ultrasonic testing (UT), phased array (PA), and time-of-flight diffraction (TOFD) inspection, and only display the ones that are needed.
- Probe and wedge definitions
- Group definitions: Create sector scan, linear scan, compound scan, or TOFD, and dual matrix pitch-catch inspection.
- UT parameter adjustments
- File export management
- Report generation

1.2 File Transfer between NDT SetupBuilder and the OmniScan

Setup files can easily be transferred between NDT SetupBuilder and the OmniScan. The application can generate .ondtsetup connectivity files or .law delay files for export to OmniScan instruments. For more information, see “Supported File Formats” on page 27.

Connectivity files are a flexible option that can be used to transfer entire inspection setup configurations to the OmniScan. These configurations are then computed by the OmniScan prior to inspection and optimized according to the OmniScan model and the module being used. For more information, see “Exporting a Workspace As a Connectivity File” on page 162.

When delays cannot be computed natively by the OmniScan, .law files are an available option. Group setups computed in NDT SetupBuilder can be transferred to OmniScan instruments, where they are simply executed. To generate .law files, specific parameters must be configured. For more information, see “Exporting a Workspace as a .law File” on page 164.

2. Installing the NDT SetupBuilder Software

Before you install the NDT SetupBuilder software on your computer, make sure the minimum requirements for installation are met. Installing NDT SetupBuilder is easy and straightforward.

2.1 Minimum Computer Requirements

The minimum requirements for the computer on which you want to install NDT SetupBuilder are:

- 2 GB RAM or higher
- 100 GB free hard disk space
- Graphics display card supporting 3-D computer-aided design (CAD) and DirectX 9.0
- 1280 × 1024 pixels or higher display adaptor and monitor resolution
- One USB port for the HASP security hardware key
- A keyboard
- A pointing device. A mouse with three buttons is highly recommended to manipulate the RayTracing view.
- One of the following operating systems:
  - Microsoft Windows 7, 32-bit and 64-bit versions
  - Microsoft Windows 8

IMPORTANT

NDT SetupBuilder can be installed on a drive other than that on which Windows resides. In such case, the NDT SetupBuilder installer will still require 150 MB on the drive where Windows resides in order to install the hardware security key driver, DirectX updates, and Windows temporary installation.

NOTE

In this document, NDT SetupBuilder screenshots were taken on a computer running Windows 7. The appearance of the screenshots may differ if you are using NDT SetupBuilder on a computer running Windows 8.

For most examples in this document, NDT SetupBuilder was arbitrarily configured to use the metric measurement units (see "Changing the Measurement Unit Parameters" on page 25 for information about how to change measurement units).

2.2 Installing NDT SetupBuilder

Installing NDT SetupBuilder is simple and is done with a standard software installer.

To install NDT SetupBuilder

1. Log on to the computer on which you want to install NDT SetupBuilder using a user account with administrator rights.
2. Access the installer for the NDT SetupBuilder software on the Evident website at EvidentScientific.com. The component can also be found on the USB flash drive provided with Evident instruments.

NOTE

The Evident website contains the most recent version of the NDT SetupBuilder installer.

3. Double-click **InstallSetupBuilder-[*VersionNumber*].exe**.

The installation wizard appears.

4. Follow the steps in the NDT SetupBuilder installer wizard that appears on the screen.

3. Getting Started with NDT SetupBuilder

After NDT SetupBuilder has been installed, you can access the software by inserting the provided HASP security hardware key into one of your computer's USB ports.

NDT SetupBuilder has been designed as a simple and easy-to-use tool used to configure inspection setups.

Three menus (**File**, **Tools**, and **Help)** contain all of the commands that provide access to the main application, file, part, weld, acquisition unit, probe, wedge, element deactivation or activation, and report management features.

Two tabs (**Probe Sets** and **Groups)** contain toolbars with controls that can be used to manage the probe sets and groups, in addition to parameters that can be used to configure inspection setups.

The RayTracing view displays a graphical representation of the part, weld, probe sets, and groups used to define your inspection setups. A toolbar provides interactive controls to change the view and modify certain probe-set parameters.

3.1 Starting NDT SetupBuilder

With NDT SetupBuilder, you configure inspection setups without a connected OmniScan instrument. After your inspection setups have been completed, the saved inspection setup files can be transferred to an OmniScan using a storage device such as a USB key or an SD card.

To start NDT SetupBuilder**,** use the usual commands provided under the Windows operating system.

To start NDT SetupBuilder

1. Connect the HASP security hardware key to the appropriate USB port of the computer. NDT SetupBuilder must be able to detect the security hardware key in order to operate.
2. Turn on the computer and wait for Windows to complete its start-up process. Do not start NDT SetupBuilder until this process has completed.
3. Start NDT SetupBuilder:

   - On the Windows desktop, double-click the NDT SetupBuilder icon .
     OR
     On the Windows taskbar, select **Start > All Programs > Evident NDT >** NDT SetupBuilder ***n.n*** > NDT SetupBuilder ***n.n***.

3.2 Exiting NDT SetupBuilder

To exit NDT SetupBuilder, use the standard commands available under the Windows operating system.

To exit NDT SetupBuilder

- On the **File** menu, select **Exit**.
  OR
  Click the **Close** button located on the right-hand side of the main window’s title bar (see Figure 3-1 on page 24).

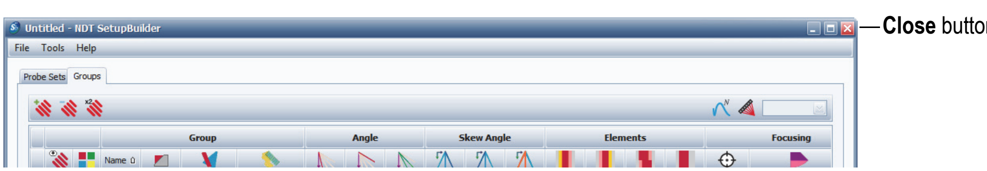


Figure 3-1 The Close button on the title bar

3.3 Workspace

Starting the NDT SetupBuilder software opens a workspace. A workspace is actually an inspection setup file. It contains all the parameters of an inspection setup: acquisition unit, part, weld (if appropriate), probe set, and group.

By default on start-up, NDT SetupBuilder opens a new untitled workspace with default parameters. An existing workspace with configured parameters can also be opened and edited using menu commands, management dialog boxes, and toolbars and parameters on tabs.

3.4 Changing the Measurement Unit Parameters

The NDT SetupBuilder preferences can be used to change the measurement unit for your inspection setup.

The following two measurement units are available: metric (millimeters) and imperial (inches).

The metric measurement units, values, precision, and increments are displayed in Table 1 on page 25.

Table 1 Metric measurement units

Measurement type	Measurement unit	Symbol	Precision[a]	Increment
Velocity	Meters per second	m/s	No digit	1
Distance	Millimeters	mm	One digit	1
Frequency	Megahertz	MHz	Two digits	0.25
Angles	Degrees	°	One digit	1.0
Others	—	—	One digit	1

a. In the **Manage Probes** and **Manage Wedges** dialog boxes, several of the parameter values are displayed with a three-digit precision.

The imperial measurement units, values, precision, and increments are displayed in Table 2 on page 26.

Table 2 Imperial measurement units

Measurement type	Measurement unit	Symbol	Precision	Increment
Velocity	Inches per microsecond	in./µs	Four digits	0.04
Distance	Inches	in.	Four digits	0.04
Frequency	Megahertz	MHz	Two digits	0.25
Angles	Degrees	°	One digit	1.0
Others	—	—	Four digits	1

To change the measurement unit

1. On the **Tools** menu, select **Preferences**.
2. In the **Preferences** dialog box, click the **Units** list, and then select the appropriate measurement unit (see Figure 3-2 on page 26).

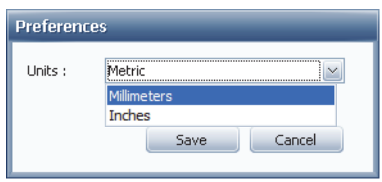


Figure 3-2 Selecting the measurement unit in the Preferences dialog box

There are two measurement units to choose from: **Millimeters** and **Inches**. The default measurement unit is **Millimeters**.

3. Click **Save**.

The selected measurement unit is immediately updated in all the NDT SetupBuilder interfaces, and in the newly generated reports.

NOTE

In the **Manage Wedges** dialog box, point the mouse to each parameter to display a tooltip with a short description.

3.5 Supported File Formats

NDT SetupBuilder, allows you to configure entire inspection setup parameters and then generate .ondtsetup or .law files, which can then be exported to OmniScan instruments.

Setup files can be easily transferred from NDT SetupBuilder to an OmniScan.

NDT SetupBuilder can export beam (focal law) configurations to the OmniScan using the .law file format described in Table 3 on page 27.

Table 3 File format supported by NDT SetupBuilder

File type	Extension	File content
Evident connectivity	.ondtsetup	All the parameters needed to recreate the setup. The parameters are computed on the OmniScan instrument, with the results optimized in accordance with the model and module used for inspection (see “Exporting a Workspace As a Connectivity File” on page 162).
OmniScan setup	.law	Calculated ultrasonic group parameters readable by the OmniScan and other equipment. The parameters are computed by NDT SetupBuilder, and the inspection is executed on the OmniScan instrument without additional calculations. This makes it possible to calculate delays that could not be obtained natively by the OmniScan instrument and then perform inspections that exceed the unit’s built-in capabilities (see “Exporting a Workspace as a .law File” on page 164). See “Description of the .law File Format” on page 179 for more information.

All the groups created using NDT SetupBuilder can be exported as .law files. To generate .law files, the following parameters need to be configured:

- Focal laws for long seam inspection
- Dual matrix array inspection

- Linear lateral inspection
- Compound scan

For more information on exporting inspection setups, see “Exporting Setup Files to an OmniScan Instrument” on page 161.

4. Interface Overview

The NDT SetupBuilder user interface includes a menu bar, two tabs along the top portion, and a RayTracing view in the lower portion (see Figure 4-1 on page 29).

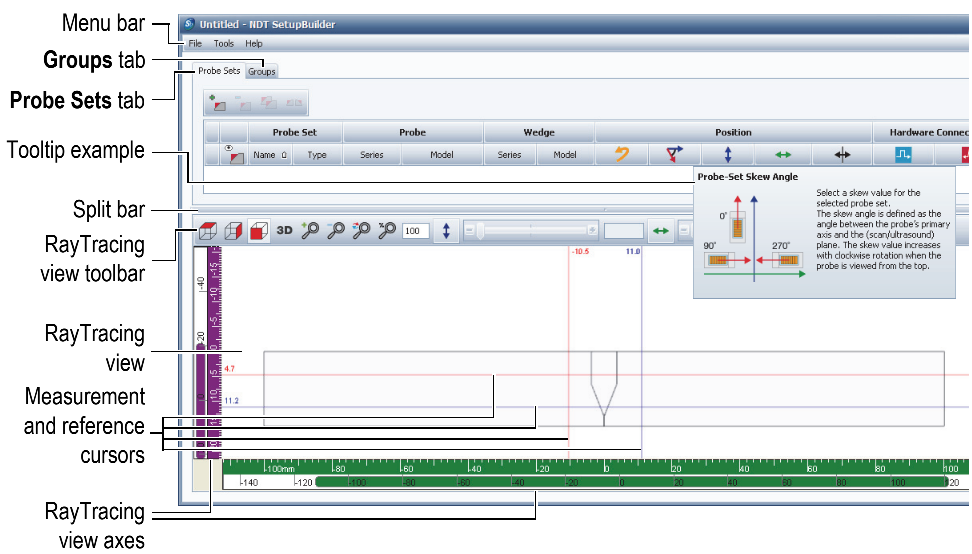


Figure 4-1 The NDT SetupBuilder user interface

The menu bar is simple. It provides file-related, tool-related commands (acquisition unit, part, probe manager, wedge manager, preferences, and probe-element deactivation) in addition to help-related commands.

The two tabs provide toolbars and parameters to define probes and groups for the inspection setups. In each tab, the probe sets (see “Probe Sets Tab” on page 32) or the groups (see “Groups Tab” on page 32) of the inspection setup are displayed as lists from which parameters can be configured.

The RayTracing view located in the lower portion of the main interface (see “RayTracing View” on page 40) displays a graphically rendered 2-D or 3-D illustration of the part, probes, wedges, in addition to display options, interactive parameter configuration controls, and probe-element information.

Between the tabs area and the RayTracing view, a split bar allows you to change the relative surface occupied by the tabs and the RayTracing view (see “Modifying the Pane Size in the Main Window” on page 54).

Explanatory tooltips are displayed by pointing the mouse over almost all of the interface elements, including the tabs, toolbars, and dialog boxes (see “RayTracing View” on page 40). The only exceptions are the **File**, **Tools**, and **Help** menus and commands and the contents displayed in RayTracing view.

From the NDT SetupBuilder interface, you can refer to an online help version of this user’s manual. For more information, see “Accessing the Online Help” on page 55.

4.1 Menus

The NDT SetupBuilder menu bar provides three menus: **File**, **Tools**, and **Help**.

The **File** menu contains commands for operations related to managing workspaces, exporting setups to various file formats, and exiting NDT SetupBuilder.

The **Tools** menu contains commands for operations related to selecting the acquisition instrument, defining the part and weld for the inspection setup, managing probes and wedges, configuring NDT SetupBuilder preferences, and deactivating or reactivating phased array probe elements.

The **Help** menu contains commands for operations related to managing the NDT SetupBuilder software.

4.2 Accessing Commands Using Keyboard Shortcuts

Several of the menu commands available in the **File**, **Tools**, and **Help** menus can be activated using keyboard shortcuts. Keyboard command shortcuts are found in the menus on the right-hand side of the corresponding command (see Figure 4-2 on page 31 and Table 4 on page 31).

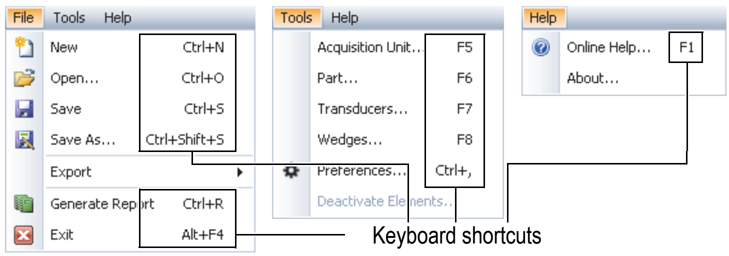


Figure 4-2 Menu commands and keyboard shortcuts

Table 4 Menu commands with keyboard shortcuts

File menu		Tools menu		Help menu	
Command	**Shortcut**	**Command**	**Shortcut**	**Command**	**Shortcut**
New	CTRL+N	**Acquisition Unit**	F5	**Help**	F1
Open	CTRL+O	**Part**	F6		
Save	CTRL+S	**Probes**	F7		
Save As	SHIFT+CTRL +S	**Wedges**	F8		
Generate Report	CTRL+R	**Preferences**	CTRL + Comma		
Exit	ALT+F4				

On the **Probe Sets** tab, **Groups** tab, and in the RayTracing view, several commands from the toolbars or parameters can also be activated using keyboard shortcuts (see Table 4 on page 31).

Table 5 Tab and RayTracing view commands with keyboard shortcuts

Probe Sets tab		Groups tab		RayTracing view	
Command	**Shortcut**	**Command**	**Shortcut**	**Command**	**Shortcut**
Add Probe	CTRL+Insert	**Add Group**	CTRL+Insert	**Show Top (C) View**	CTRL+1
Delete Probe	CTRL+Delete	**Delete Group**	CTRL+Delete	**Show Side (B) View**	CTRL+2
Duplicate Probe	CTRL+D	**Duplicate Group**	CTRL+D	**Show End (D) View**	CTRL+3
Visible Probe	CTRL+H	**Visible Group**	CTRL+H	**Show 3-D View**	CTRL+4

4.3 Probe Sets Tab

The **Probe Sets** tab contains the functions and parameters needed to configure the probes, probes, wedges, and hardware connections for the inspection setup. For more details, see "Configuring One or More Probe Sets" on page 91.

4.4 Groups Tab

The **Groups** tab contains functions and parameters needed to configure the groups for the inspection setup. For more details, see "Configuring One or More Groups" on page 137.

4.5 Identifying Selected Probe Sets in RayTracing View

When you select one or more probe sets in the **Probe Sets** tab, or when you select one or more of their associated groups in the **Groups** tab, the probe sets become highlighted in green in RayTracing view (see Figure 4-3 on page 33).

A probe set selected in the tab is highlighted in green.

Selecting one or more groups associated with the probe set also highlights the probe set.

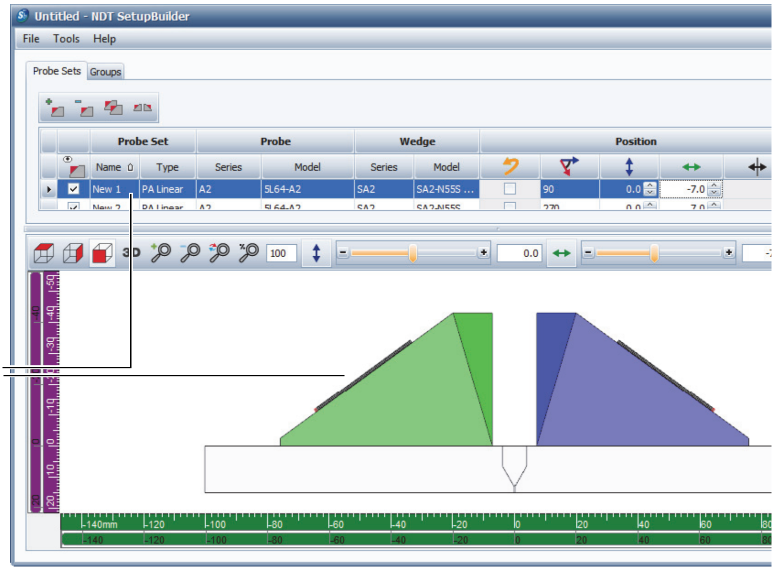


Figure 4-3 The selected probe set highlighted in green in RayTracing view

4.6 Customizing the Display of the Probe Sets Tab or Groups Tab Data

The default column configuration and the data display from the **Probe Sets** and **Groups** tabs can be customized according to your needs. This allows you to relocate, hide, or display columns as well as sort, group, or filter alphanumeric data to reorder rows.

NOTE

The section headers on the **Probe Sets** and **Groups** tabs do not contain the shortcut menu with the column-configuration and data display commands.

4.6.1 Relocating Columns or Sections on a Tab

On the **Probe Sets** or **Groups** tab, section columns or column headers can be easily relocated using a drag-and-drop operation (see Figure 4-4 on page 34).

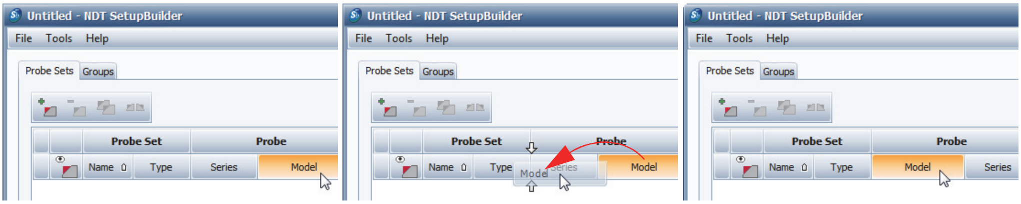


Figure 4-4 Dragging a column header on a tab

To relocate a column or a section header on a tab

◆ Drag the column header or section header to the desired location on the same tab.

On the **Probe Sets** or **Groups** tab, when a section header (the column in the top row) is dragged, the columns below it are also relocated (see Figure 4-5 on page 34).

For example, when you drag the **Probe** section header on the **Probe Sets** tab, the **Series** and **Model** columns of that section are also simultaneously relocated.

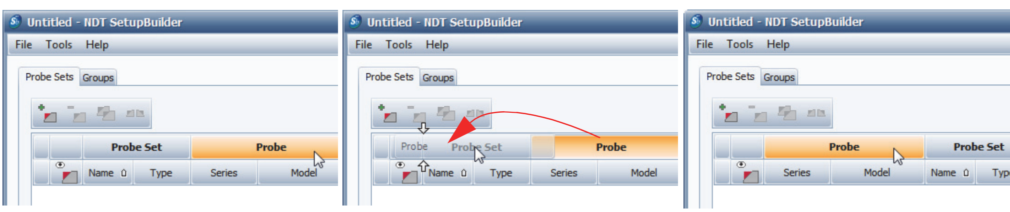


Figure 4-5 Dragging a section header on a tab

4.6.2 Sorting Data in a Column

The data in the **Name** column of the **Probe Sets** tab and in the **Probe** column of the **Groups** tab can be assorted in ascending or descending order. The availability of the sort function is indicated by an arrow on the column header (see Figure 4-6 on page 35).

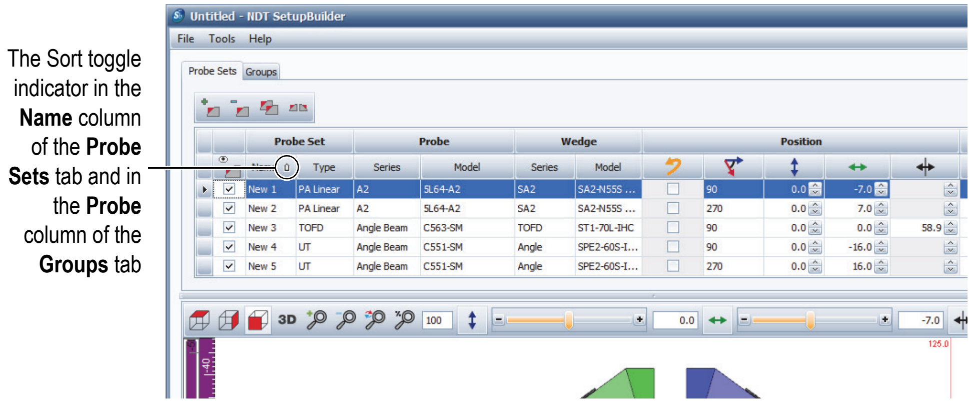


Figure 4-6 The Sort toggle indicator

The items in the tab are sorted in ascending order when the arrow on the column header is pointing upward and in descending order when the arrow is pointing downward.

To sort data within a column

- Click the **Name** or the **Probe** column header to sort the data in ascending or descending order, as indicated by the arrow on the column header.

  The sort function is a toggle command.

4.6.3 Removing and Inserting Columns or Sections on a Tab

On the **Probe Sets** or **Groups** tabs, columns, column headers, and sections can be removed or reinstated.

To remove or insert a column or a section header on a tab

- Right-click the column header on the **Probe Sets** or **Groups** tab, and then, on the shortcut menu, select one of the following commands (see Figure 4-7 on page 36):

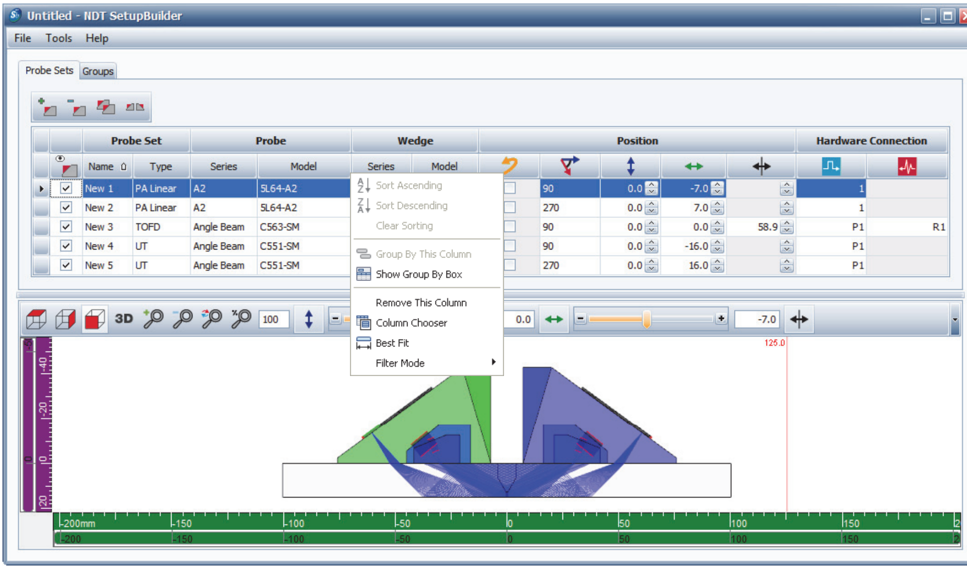


Figure 4-7 Removing a column from a tab

— **Remove This Column**

The column disappears from the tab.

— **Column Chooser**

The **Customization** dialog box appears (see Figure 4-8 on page 37), in which one of the following operations can be performed:

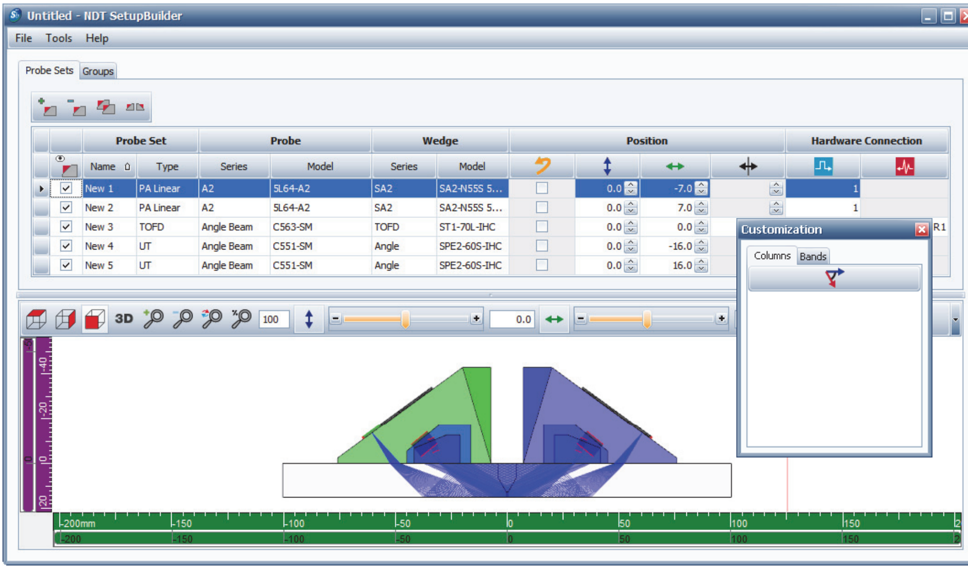


Figure 4-8 Returning a column on a tab

- To remove a column, drag the desired column header of the tab to the **Columns** tab of the **Customization** dialog box.
- To replace a previously removed column on the tab, drag the desired column header from the **Columns** tab in the **Customization** dialog box back to the tab, or double-click the column header. When you double-click a column header, it returns to its original position before being removed.
- To remove an entire section header, including all of its columns, drag the desired section header from the tab on the main window to the **Bands** tab in the **Customization** dialog box (see Figure 4-9 on page 38).

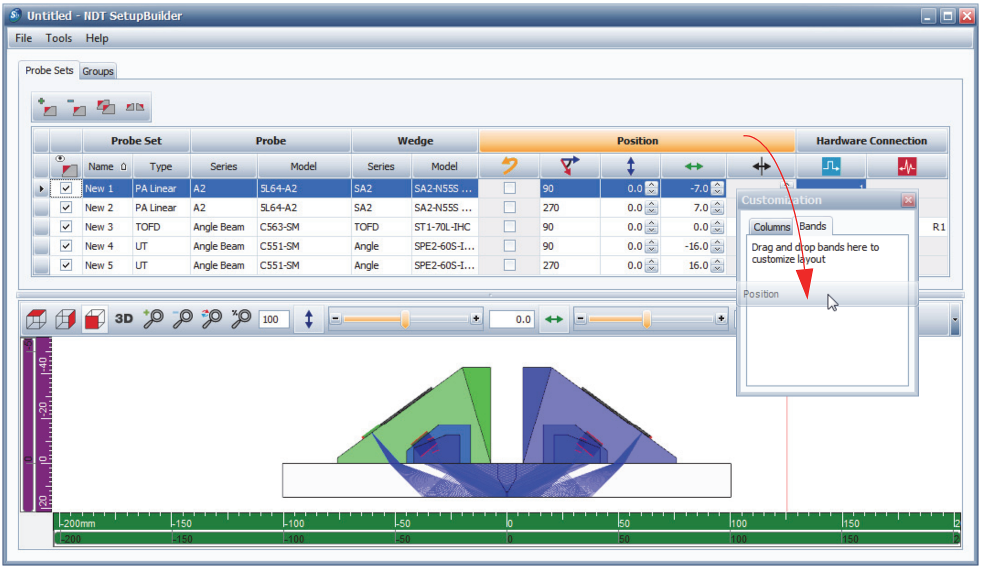


Figure 4-9 Removing a section header from a tab

- To return a section header removed from the tab, drag the desired section header from the **Bands** tab in the **Customization** dialog box back to the tab, or double-click the column header. When you double-click a column header, it returns to its original position before being removed.

NOTE

On the **Probe Sets** and **Groups** tabs, the section headers do not contain the shortcut menu with the column-configuration and data display commands.

4.6.4 Adjusting the Columns using the Best-Fit Function

The **Probe Sets** or **Groups** tab can be used to best-fit the width of individual columns. In other words, this enables you to select the best column width for data display.

To best-fit columns on a tab

◆ Right-click a column header on **Probe Sets** or **Groups** tab, and then, on the shortcut menu, click **Best-Fit** (see Figure 4-10 on page 39).

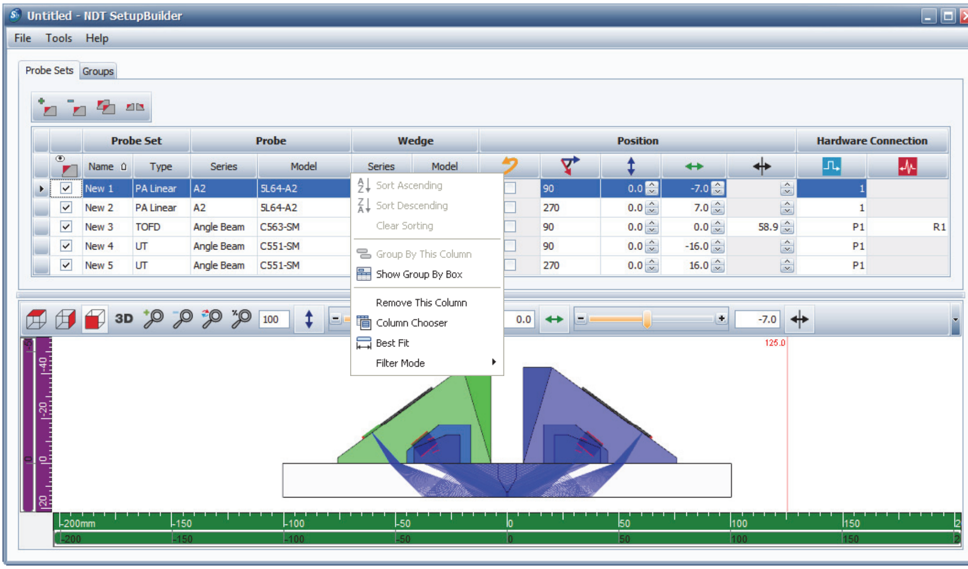


Figure 4-10 Best-fitting the data within a column on a tab

The column width is adjusted to the length of the line with the most content.

NOTE

The section headers on the **Probe Sets** and **Groups** tabs do not contain the shortcut menu with the column-configuration and data display commands.

4.7 RayTracing View

NDT SetupBuilder contains a RayTracing view that is similar to those offered in other Evident software. The RayTracing view displays graphical representations of the part and the weld, predefined or custom probe sets (probes and wedges), and the groups that are used in the inspection setups.

The RayTracing view provides standard Side (B-scan), Top (C-scan), and End (D-scan) data views with color-coded axes and an interactive 3-D view.

Table 6 on page 40 lists the basic ultrasonic views (scans) that are illustrated in Figure 4-11 on page 41. The color-coded axis content for each viewing option provided in RayTracing view is illustrated in Figure 4-12 on page 42.

Table 6 Basic data views available in RayTracing view

Point of view	View	Axis content
Top	C-scan	Scan (blue) versus Index (green)
Side	B-scan	Ultrasound (pink) versus Scan (blue)
End	D-scan	Ultrasound (pink) versus Index (green)

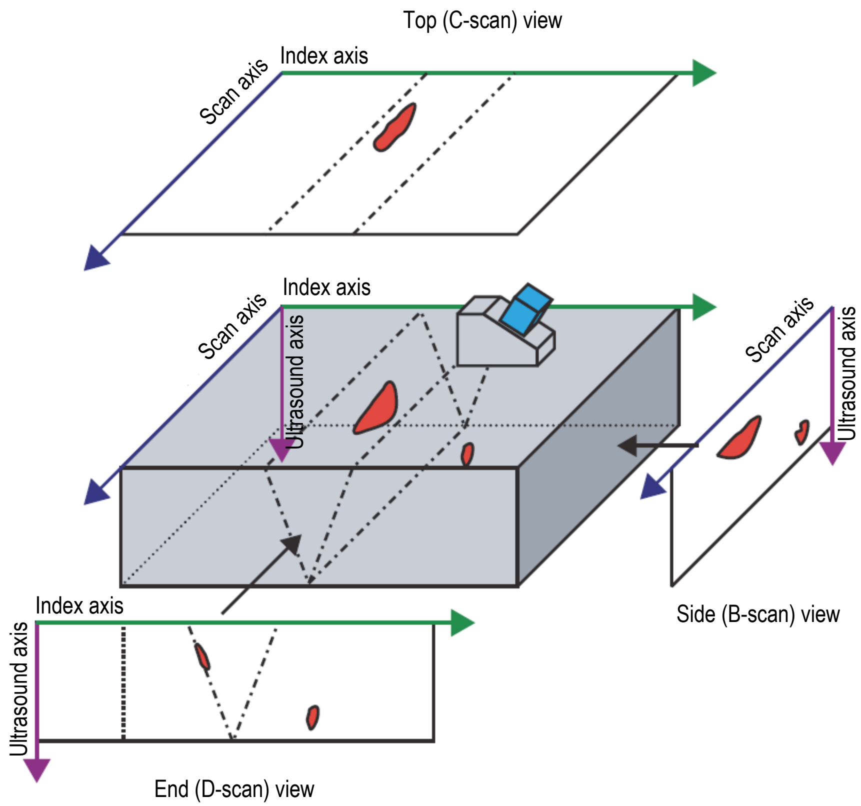


Figure 4-11 Top, Side, and End ultrasonic views with 90° probe skew angle

In the example in Figure 4-11 on page 41, if the probe skew angle is 0° (or 180°), the Side view becomes the End view, and vice versa.

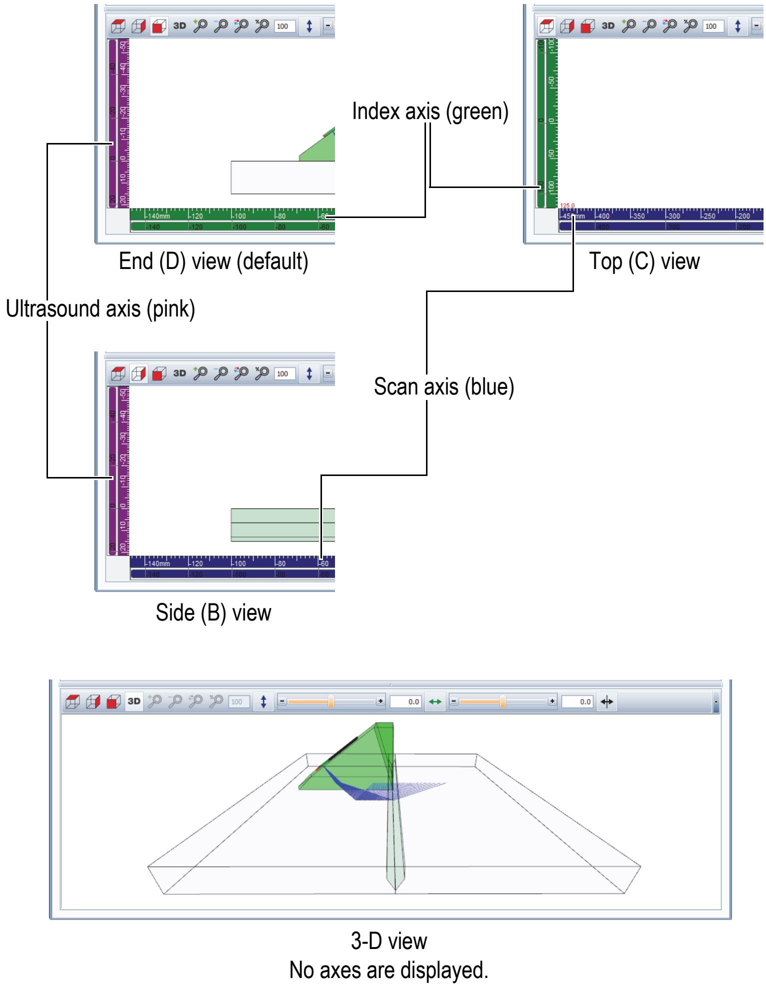


Figure 4-12 Axis content for the RayTracing view options

The RayTracing view provides the following controls and elements, which are used to view and configure inspection setups:

Toolbar

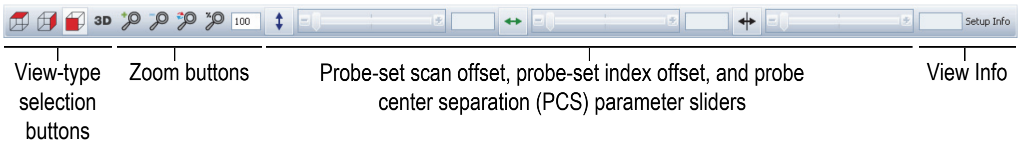


Figure 4-13 The toolbar for the RayTracing view

The RayTracing view toolbar is located in the upper-left corner. This view provides controls for adjusting the view according to your needs and for interactive modification of three probe parameters (see Figure 4-13 on page 43).

The toolbar controls can be used to perform the following tasks in RayTracing view:

— Select a view type (see “Selecting a View Type” on page 45).

— Interactively modify probe parameter values (see “Interactively Modifying Probe Parameter Values” on page 47).

— Zoom the RayTracing view (see “Zooming the RayTracing View” on page 50).

NOTE

On the RayTracing view toolbar, point the mouse to each toolbar element to display a tooltip with a short description of the function.

Rulers

The rulers are scales displayed on the left and at the bottom of the RayTracing view (see Figure 4-14 on page 44). The color of the ruler identifies the axis. For more information about the axis color codes, see “RayTracing View” on page 40.

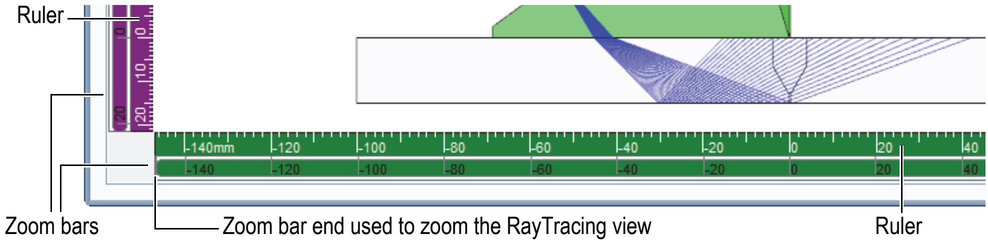


Figure 4-14 RayTracing view axis components

Zoom bars

Zoom bars appear on the left and bottom edges of the RayTracing view (see Figure 4-14 on page 44). Each zoom bar contains a zoom box that enables you to set the visible part of the view. The zoom box shows the relative position and proportion of the graphical representation currently visible in the display area relative to the contents of the entire RayTracing view. The color of the zoom bar identifies the axis on the different views.

To zoom in or out of the RayTracing view using a zoom bar, drag one end of the zoom bar to resize it. Scroll through the zoom bar to view other parts of the graphical representation (see “Zooming the RayTracing View” on page 50).

TIP

To reset a zoom bar, just double-click it.

Cursors

The cursors are thin horizontal and vertical lines used to identify points and/or a region within RayTracing view. Two cursor types are available: reference and measurement cursors. A pair of each type of cursor is displayed: one vertically and the other horizontally. Reference cursors are red and measurement cursors are blue. A label indicates the exact coordinate of each cursor along the x-axis or y-axis.

Reference and measurement cursors can be displayed quickly in RayTracing view (see “Displaying Cursors” on page 53).

In other Evident software, the cursors are also used to measure the data displayed.

In RayTracing view, the configured part, weld, probe sets (probes and wedges), groups, and group focus points are displayed at their true scale (see Figure 4-15 on page 45). The first element of a probe is identified with a red outline. Group focus points are displayed as green dots.

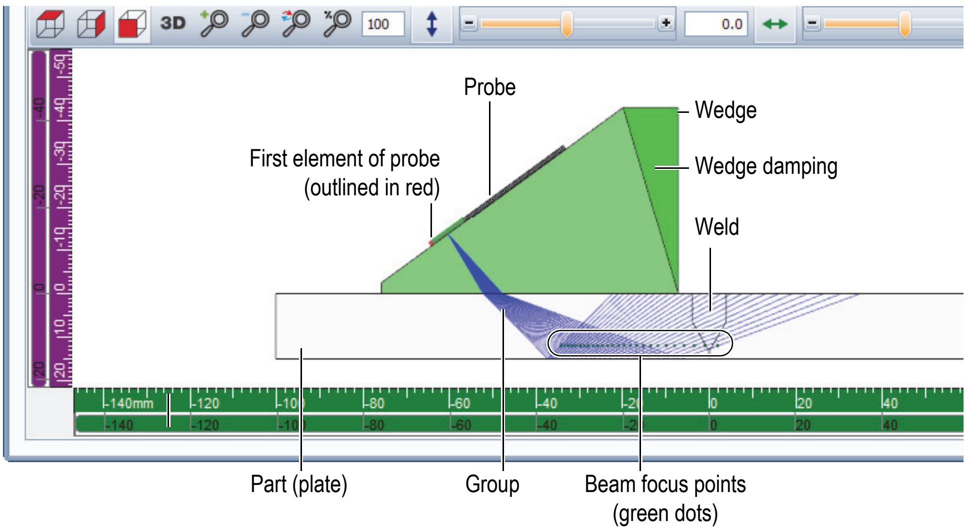


Figure 4-15 Part, weld, probe set, and group representation in RayTracing view

4.8 Selecting a View Type

The RayTracing view toolbar provides the following controls, which are used to select a point of view:

NOTE

For dual matrix-type probes, the End (D) view is selected by default. The Top (C) view, Side (B) view, and 3-D view are not available.

— **Show Top (C) View** ()

The Top (C) view is defined by the index axis (represented by the vertical axis in RayTracing view), and the scan axis (represented by the horizontal axis). The Top (C) view provides a C-scan view (see Figure 4-16 on page 46).

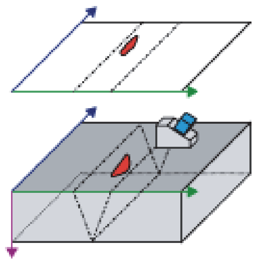


Figure 4-16 Top (C-scan) view

— **Show Side (B) View** ()
  The Side (B) view is defined by the ultrasound axis (represented by the vertical axis in RayTracing view) and the scan axis (represented by the horizontal axis). The Side (B) view provides a B-scan view (see Figure 4-17 on page 46).

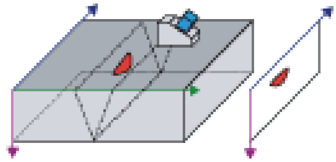


Figure 4-17 Side (B-scan) view

— **Show End (D) View** ()
  The End (D) view is defined by the ultrasound axis (represented by the vertical axis in RayTracing view) and the index axis (represented by the horizontal axis). The End (D) view provides a D-scan view (see Figure 4-18 on page 47).

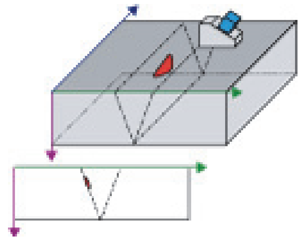


Figure 4-18 End (D-scan) view

— **Show 3-D View** ()

The 3-D view provides a three-dimensional view of the setup defined by the ultrasound axis, the scan axis, and the index axis.

For complete descriptions of these points of view, see “RayTracing View” on page 40.

To select a view type

◆ Click the appropriate point-of-view button on the RayTracing view toolbar: **Show Top (C) View**, **Show Side (B) View**, **Show End (D) View**, or **Show 3-D View**.

TIP

The RayTracing view toolbar’s points-of view commands can be activated using keyboard shortcuts. For more information, see “Accessing Commands Using Keyboard Shortcuts” on page 31.

4.9 Interactively Modifying Probe Parameter Values

The RayTracing view toolbar provides the following controls, which can be used to interactively modify the parameters of a probe set selected on the **Probe Sets** tab:

- **Probe-Set Scan Offset** slider 

  The **Probe-Set Scan Offset** slider changes the distance between the center of the probe front and the origin (zero) of the scan axis.

The parameter value for the selected probe set is interactively updated in the **Probe-Set Scan Offset** ( ) column of the **Probe Sets** tab (see “Configuring Probe-Set Position Parameters” on page 130).

- **Probe-Set Index Offset** slider

  The **Probe-Set Index Offset** slider changes the distance between the center of the probe front and the origin (zero) of the index axis.

  The parameter value for the selected probe set is interactively updated in the **Probe-Set Index Offset** ( ) column of the **Probe Sets** tab (see “Configuring Probe-Set Position Parameters” on page 130).

- **Probe Center Separation (PCS)** slider

  The **PCS** slider changes the distance between a pair of probes, the probe center separation (PCS) distance from exit point to exit point, or the distance for a TOFD-type probe set.

  On the **Probe Sets** tab for the selected probe set, the parameter value is interactively updated in the **Probe Center Separation (PCS)** ( ) column (see “Configuring Probe-Set Position Parameters” on page 130).

NOTE

The **Probe Center Separation (PCS)** slider is unavailable for a probe set of the PA linear, UT, or dual matrix type.

To interactively modify one of the three probe-set parameters using the RayTracing view toolbar

1. Select a probe set on the **Probe Sets** tab, or select a group in the **Groups** tab (the parameters of the probe set related to this group will be modified).
2. Move the **Scan Offset**, **Index Offset**, or **Probe Center Separation (PCS)** slider to the left to reduce the parameter value, or move the slider to the right to increase it.

   The value is automatically updated in the corresponding column on the **Probe Sets** tab, and the graphical representation is automatically and accurately updated in RayTracing view.

4.10 Viewing Inspection Setup Information

The RayTracing view toolbar contains a **Setup Info** button, which can be used to view specific information.

To view inspection setup information

- Click the **Setup Info** button on the right-hand side of the RayTracing view toolbar (see Figure 4-19 on page 49).

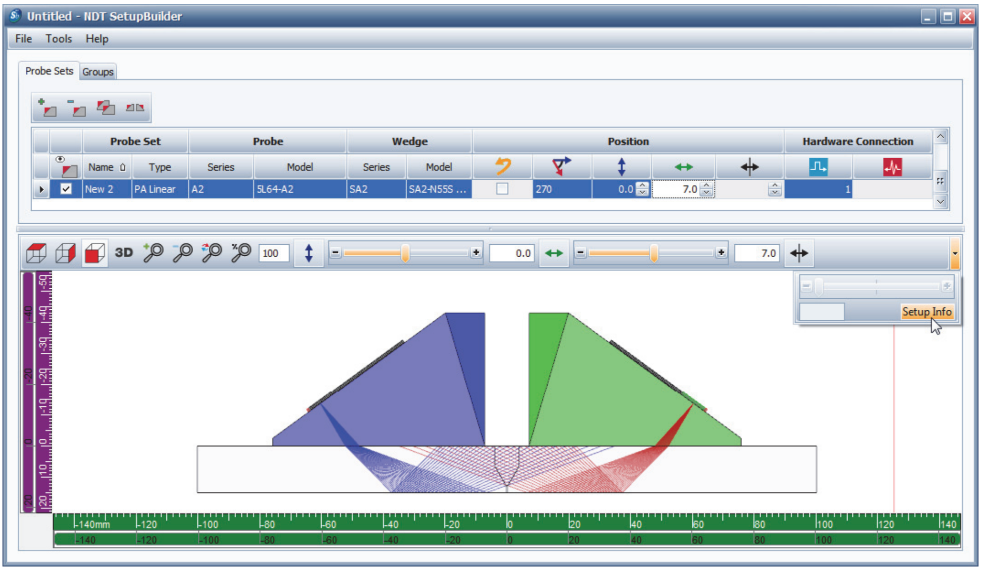


Figure 4-19 Clicking the Setup Info button on the RayTracing view toolbar

The **Setup Info** pane opens on the right-hand side of the RayTracing view. It displays information related to the items selected in the active tab, the visibility of probe sets and groups, and the part parameters (see Figure 4-20 on page 50).

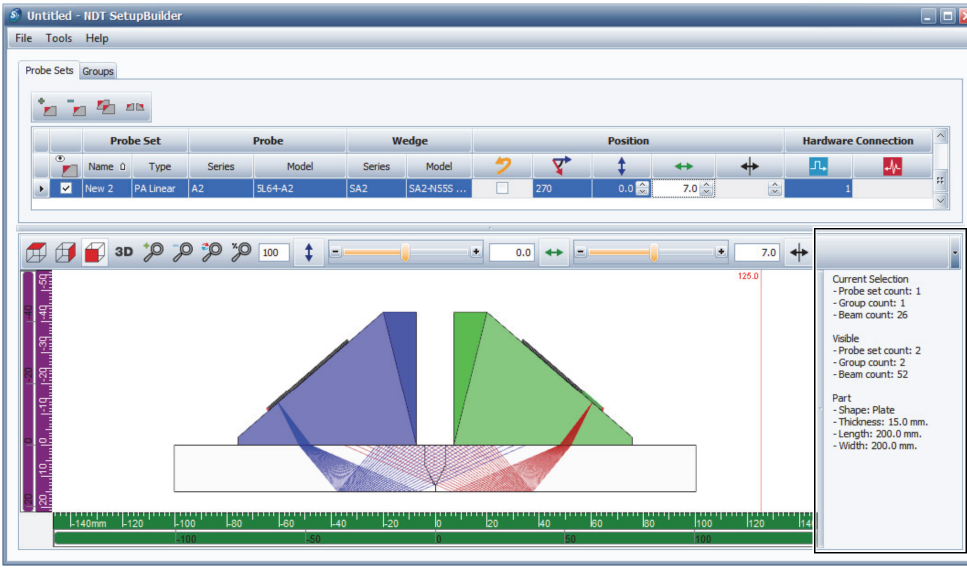


Figure 4-20 The RayTracing view with the Setup Info pane displayed

4.11 Zooming the RayTracing View

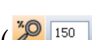
To change the zoom factor or the appearance of the RayTracing view, use the toolbar's zoom buttons, the mouse, or the zoom bars.

The RayTracing view toolbar contains the following controls, which can be used to zoom in and out:

— **Zoom In** button ()

— **Zoom Out** button ()

— **Reset Zoom** button ()

— **Custom Zoom** button ()

When using the **Zoom In** button (), the RayTracing view is zoomed in at 200 %.

When using the **Zoom Out** button (), the RayTracing view is zoomed out at 50 % of the current scale, and always from its center point. The graphical representation is scaled proportionally.

NOTE

The 100 % scale in RayTracing view shows the entire part and all the probe sets included in your inspection setup.

To zoom in on the RayTracing view

- Click the **Zoom In** button () on the toolbar.

  The graphical representation is scaled proportionally to 200 % of the initial view.

  OR

  In the box on the right-hand side of the **Custom Zoom** button (150) on the toolbar, type a value higher than 100 %, and then click the **Custom Zoom** button ().

  The graphical representation is scaled proportionally.

  OR

  In 2-D view (for Top [C], Side [B], or End [D] view, see “Selecting a View Type” on page 45), drag the mouse vertically, horizontally, or diagonally on the graphical representation in RayTracing view to define the zone of interest.

  When you release the mouse button, the RayTracing view zooms in on the selected area.

NOTE

When zooming the RayTracing view with a mouse drag, the graphical representation is not scaled proportionally. Use one of the toolbar’s zoom buttons to reset the scale factor to 1:1 in RayTracing view.

To zoom out of RayTracing view

- Click the **Zoom Out** button ( ) on the RayTracing view toolbar.
  The graphical representation is scaled proportionally to 50 % of the initial view.
  OR

  In the box on the right-hand side of the **Custom Zoom** button ( ) on the toolbar, type a value equal to or higher than 100 %, and then click the **Custom Zoom** button ( ).
  The graphical representation is scaled proportionally.

To reset the RayTracing view to 100 % scale

- Click the **Reset Zoom** button ( ) on the RayTracing view toolbar.
  The graphical representation is scaled proportionally to 100 %.
  OR

  In the box on the right-hand side of the **Custom Zoom** button ( ) on the toolbar, type the 100 % value, and then click the **Custom Zoom** button ( ).
  The graphical representation is scaled proportionally to 100 %.

To zoom in on or change the appearance of the graphical representation in 3-D view

- Drag one of the three mouse buttons on the graphical representation in RayTracing view to move it in 3-D space:
  - *Left mouse button*: Rotate the 3-D graphical representation along the x, y, and z axes.
  - *Middle mouse button*: Zoom in or out of the 3-D graphical representation.
  - *Right mouse button*: Move the 3-D graphical representation up, down, left, or right without rotating it.

To zoom the RayTracing view using the zoom bars

- Drag one end of an axis' zoom bar to interactively change the zoom factor of the RayTracing view along that axis (see Figure 4-21 on page 53).
  OR
  Enter values as follows:

a) Hold down the CTRL key and click a zoom bar to display the zoom bar parameter box.

b) Enter values in the **Start** and **Stop** boxes.

c) Press Return.

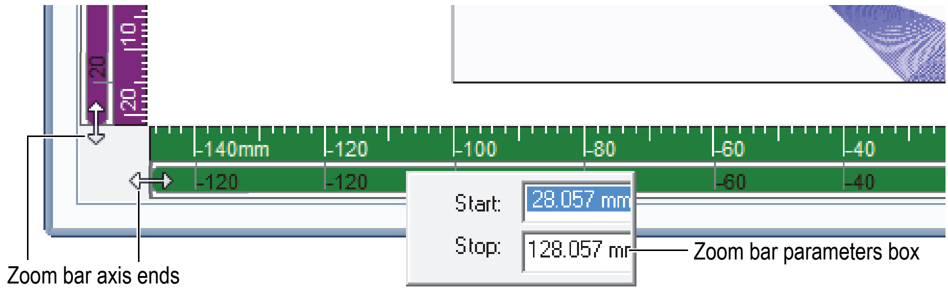


Figure 4-21 Zoom bar controls in RayTracing view

To reset the scale ratio to 1:1 in RayTracing view

- Double-click an axis zoom bar to reset the scale ratio of the RayTracing view to 1:1 along that axis (see Figure 4-21 on page 53).

  OR

  Click one of the zoom buttons (, , , or) on the RayTracing view toolbar to reset the scale ratio of the RayTracing view to 1:1 along both axes.

4.12 Displaying Cursors

Both the reference and measurement cursors can be quickly displayed in RayTracing view (see Figure 4-22 on page 54).

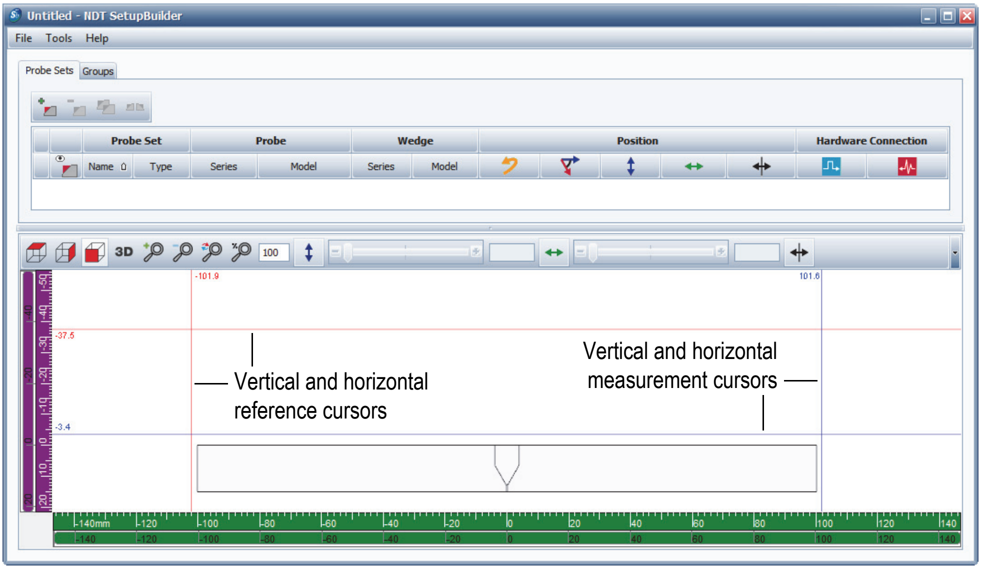


Figure 4-22 Reference and measurement cursors in RayTracing view

To display the reference cursors

- Double-click the RayTracing view.

To display the measurement cursors

- Double right-click the RayTracing view.

4.13 Modifying the Pane Size in the Main Window

The split bar in between the tabs area and the RayTracing view can be used to change the relative surface area of the tabs and the RayTracing view (see Figure 4-23 on page 55).

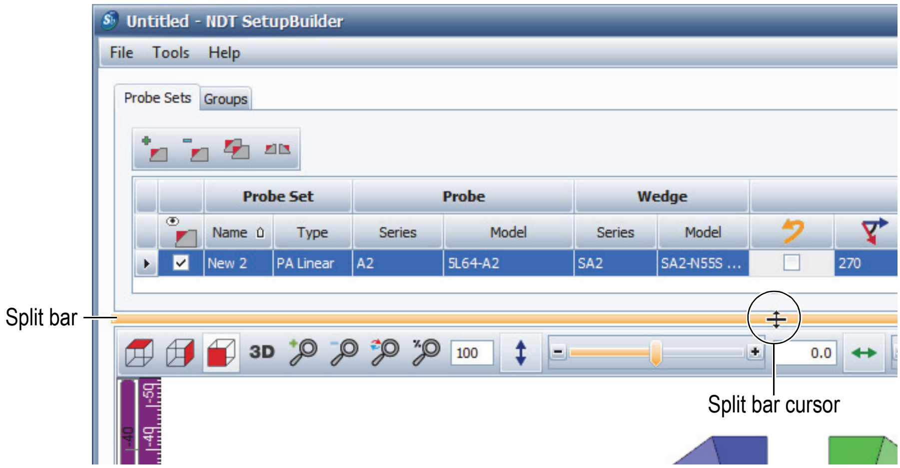


Figure 4-23 Using the split bar to modify the size of the main window's panes

To modify the pane size in the main window

- Drag the split bar up or down in between the tabs area and the RayTracing view.

4.14 Accessing the Online Help

The contents of the NDT SetupBuilder *User's Manual* are available as an interactive HTML help file directly within the software.

To access the online help

- On the **Help** menu, click **Online Help**.
  The NDT SetupBuilder HTML help window appears (see Figure 4-24 on page 56).

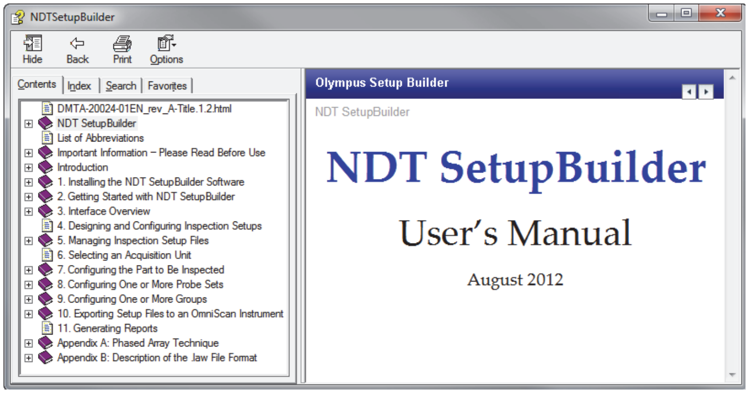


Figure 4-24 The HTML help window

Use the **Contents**, **Index**, **Search**, and **Favorites** tab tools to find relevant information.

4.15 NDT SetupBuilder Software

The **About** NDT SetupBuilder window contains the NDT SetupBuilder version number and copyright information.

To access the About NDT SetupBuilder window

- On the **Help** menu, click **About**.
  The **About** NDT SetupBuilder window appears (see Figure 4-25 on page 56).

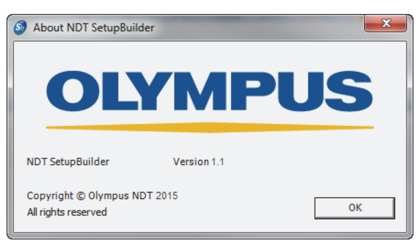


Figure 4-25 The About NDT SetupBuilder window

5. Designing and Configuring Inspection Setups

When defining an inspection setup or technique, a workspace must first be created or opened before configuring or modifying the inspection parameters.

The general procedure to create inspection setups consists of managing workspaces, selecting acquisition units, configuring the parts and welds to be inspected (as well as the probe sets and groups that are to be used for the inspection), exporting the setups to an instrument, and generating reports.

To design and configure an inspection setup

1. Create a new workspace, or open an existing one.
   For more information, see “Managing Inspection Setup Files” on page 59.
2. Select an acquisition unit.
   For more information, see “Selecting an Acquisition Unit” on page 65.
3. Configure the part to be inspected.
   For more information, see “Managing Inspection Setup Files” on page 59.
4. Configure one or more probe sets.
   For more information, see “Configuring One or More Probe Sets” on page 91.
5. Configure one or more groups.
   For more information, see “Configuring One or More Groups” on page 137.
6. Export the inspection setup to a file.
   For more information, see “Exporting Setup Files to an OmniScan Instrument” on page 161.
7. Generate a report on the inspection setup.
   For more information, see “Generating Reports” on page 169.

6. Managing Inspection Setup Files

One inspection setup can be saved per setup file. A setup file must be created for each weld inspection technique requiring configuration. Each setup file contains a workspace with a defined part and weld, and one or more required probes, wedges, and groups.

If managing multiple inspection projects, you will need to create and use several setup files. Each type of inspection technique requires one setup file, each with its own workspace.

Afterwards, the setup files containing the inspection setups will be available for transfer to the OmniScan inspection instruments. These setup files can also be used as the basis for creating other inspection setups.

6.1 Creating a Workspace

By default, a new workspace is created when you open NDT SetupBuilder. That workspace is used to design and configure a weld inspection technique. Workspace files have the .wkb extension.

Upon creation, the workspace contains a mild steel plate part with a symmetrical V weld. In place of a plate part, a tube part or bar part can be selected. Defining a weld is optional. The part and weld are fully configurable. Probes, wedges, and groups can also be selected or created, and then configured. For more information about designing and configuring a weld inspection technique, see “Designing and Configuring Inspection Setups” on page 57.

After you have completed work on an inspection setup, the workspace can be saved to a setup file. A new workspace can then be created in order to configure another setup.

To create a new workspace

- Start NDT SetupBuilder. A new workspace with a default part and weld is automatically created. For more in formation, see "Starting NDT SetupBuilder" on page 23.

  OR

  On the **File** menu, click **New**.

  A new workspace is created. If a workspace was already opened in NDT SetupBuilder, you have the option to save it before it is closed.

6.2 Opening a Workspace

It is possible to open a previously saved and closed workspace.

To open a workspace

1. On the **File** menu, click **Open**.
2. In the **Open** dialog box, select the desired workspace file (see Figure 6-1 on page 60).

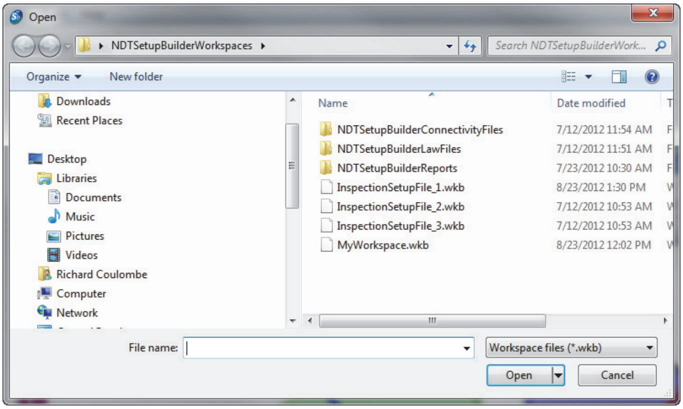


Figure 6-1 The Open dialog box used to open a workspace

3. Click **Open**.

   The selected workspace file is displayed.

The workspace can then be edited and saved (see “Saving a Workspace” on page 61), or it can be saved under a new name to serve as the basis for a new inspection technique (see “Saving a Workspace as a New File” on page 62).

6.3 Saving a Workspace

After creating or opening a workspace, the **Save** command in the **File** menu is unavailable. After you have changed at least one parameter in the workspace, the **Save** command becomes available.

When a change is made to an open workspace, an asterisk is displayed after the name of the file in the title bar until the change has been saved (see Figure 6-2 on page 61).

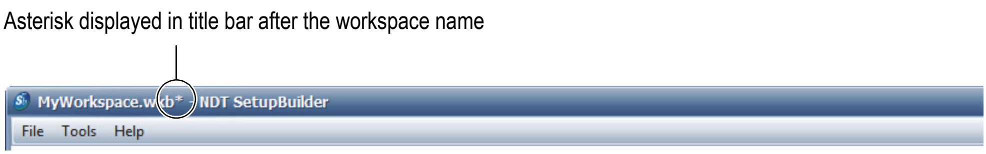


Figure 6-2 Title bar with an asterisk to indicate unsaved changes

To save a workspace

1. On the **File** menu, click **Save**.
2. In the **Save As** dialog box, choose the folder where you want to save the workspace file (see Figure 6-3 on page 62).

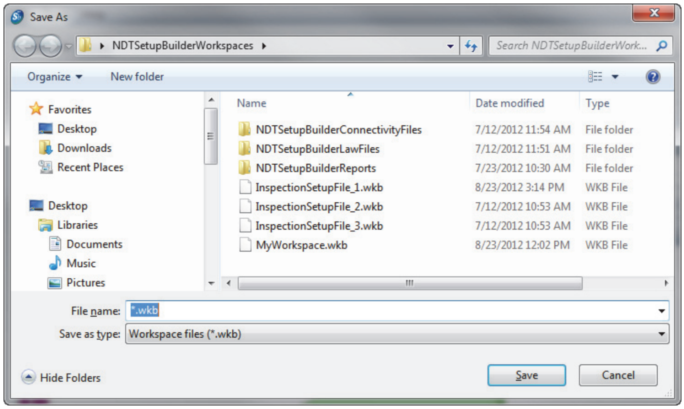


Figure 6-3 The Save As dialog box used to save a workspace

3. In the **File name** box, enter the name for the file.
4. Click **Save**.

   The workspace file is saved with the .wkb extension.

6.4 Saving a Workspace as a New File

A workspace can be saved as a new file that can be used as the basis for a new inspection setup.

To save a workspace as a new file

1. On the **File** menu, click **Save As**.
2. In the **Save As** dialog box, select the folder where you want to save the workspace file (see Figure 6-4 on page 63).

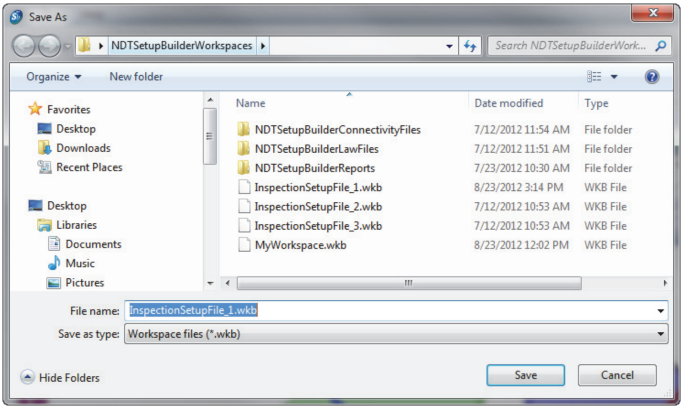


Figure 6-4 The Save As dialog box used to save a workspace as a new file

3. In the **File name** box, enter a name for the file.
4. Click **Save As**.

7. Selecting an Acquisition Unit

For every inspection setup configured in NDT SetupBuilder, you must select the target acquisition unit technology. By default, the OMNI-PA acquisition unit technology is selected, because it is the most commonly used.

NOTE

Because the choice of target acquisition unit technology has an effect on all aspects of your inspection setup, selecting the acquisition unit technology should always be the first step of the configuration procedure.

The values that depend on the target acquisition unit technology are element quantity (aperture), UT channel (pulser and receiver, found on the **Probe Sets** tab, if they are available for a unit or module), and first and last element (found on the **Groups** tab).

The **OMNI-PA** and **OMNI-UT** instrument options allow full configuration and support of OmniScan instruments, and enable saving inspections setups in the connectivity file format (.ondtsetup) or .law file format, for importing into the instruments for processing and for performing inspections.

The **Generic PA** and **Generic UT** instrument options allow generic configuration of inspections setups, which can be saved as .law files only.

To select an acquisition unit

1. On the **Tools** menu, click **Acquisition Unit**.
2. In the **Acquisition Unit** dialog box (see Figure 7-1 on page 66), in the **Instrument** list, select the target acquisition unit technology for which you want to create groups:
   — **OMNI-PA**: Phased array OmniScan for multi-element probe set.

By default, the **OMNI-PA** OmniScan target technology is selected. The **OMNI-PA** option offers support for PA linear, TOFD, and UT probe-set types. By default, 16 active elements and 128 total elements are selected.

— **OMNI-UT**: OmniScan for single element probe set.

— **Generic-PA**: Generic phased array instrument for multi-element probe set.

— **Generic-UT**: Generic instrument for single element probe set.

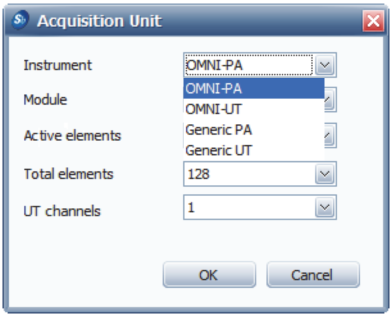


Figure 7-1 The Acquisition Unit dialog box

When the **OMNI-PA** or **Generic-PA** option is selected, the **None** option is available in the **UT channels** list, which enables you to select no channels for the inspection setup.

When the **OMNI-UT** or **Generic-UT** option is selected, the **Active elements** and **Total elements** lists become unavailable, because the UT target technology supports single-element probe sets only (see Figure 7-1 on page 66).

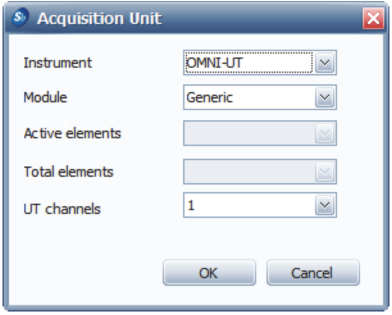


Figure 7-2 The Acquisition Unit dialog box with UT target technology

When the **Generic-PA** or **Generic-UT** option is selected, more options are displayed in the other lists available in the **Acquisition Unit** dialog box (see Figure 7-3 on page 67).

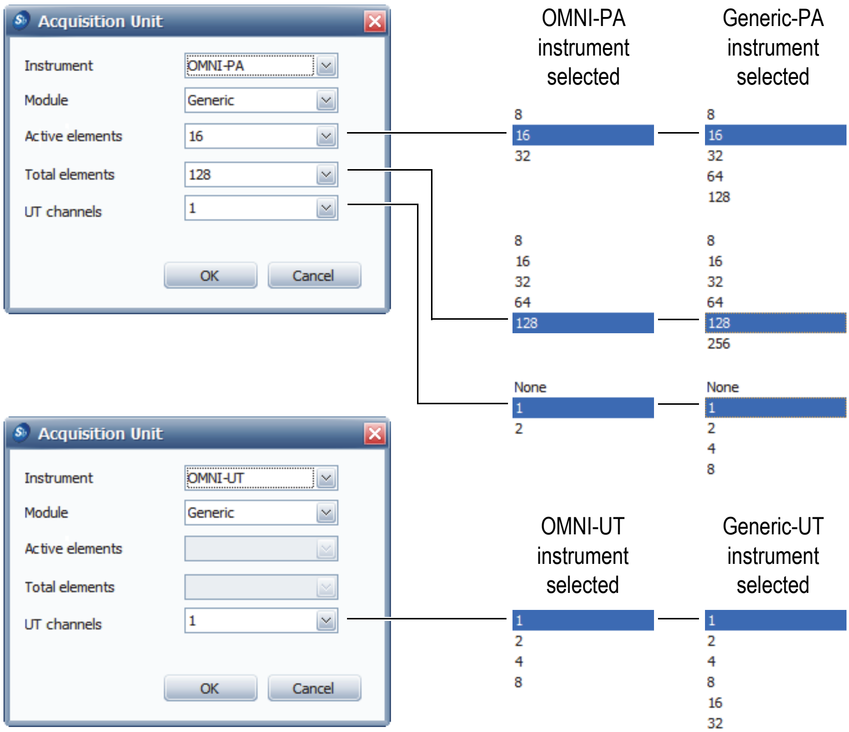


Figure 7-3 The Acquisition Unit dialog box with the various list options

3. In the **Module** list, select the module unit (see Figure 7-4 on page 68).

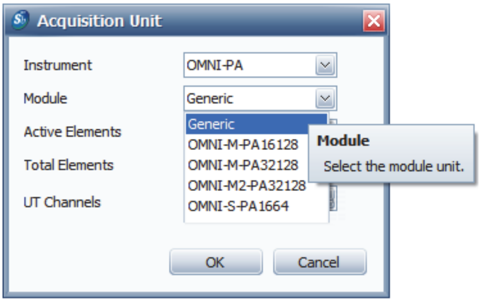


Figure 7-4 The Acquisition Unit dialog box

4. In the **Active elements** list, select the number of active elements on the probe.

   The default value is **16** elements. The number of active elements that is selected defines the maximum element quantity (aperture) for the groups.

   When the number of active elements selected in the **Active elements** list is not supported by the selected instrument technology, the value in the **Active elements** list is automatically adjusted based on the module's capacity.

5. In the **Total elements** list, select the total number of elements on the probe.

   The default value is **128** elements.

   When the number of total elements selected in the **Total elements** list is smaller than the number of active elements selected in the **Active elements** list, the value in the **Total elements** list is automatically adjusted based on the module's capacity.

   When the number of total elements selected in the **Total elements** list does not correspond with the number of total elements supported by the selected instrument technology, the value in the **Total elements** list is automatically adjusted based on the module's capacity.

6. In the **UT channels** list, select the number of channels for the inspection setup.

   The default value is **1** channel.

   When the number of channels selected in the **UT channels** list does not correspond with the number of channels supported by the selected instrument technology, the value in the **UT channels** list is automatically adjusted based on the module's capacity.

NOTE

The number of channels selected when defining the acquisition unit technology become available in the **Pulser** and **Receiver** column lists from the **Hardware Connection** section on the **Probe Sets** tab, when a probe set of the TOFD or UT type is selected. For more information, see “Setting Up the Probe-Set Hardware Connection” on page 133.

7. Click **Save**.

When you change the target acquisition unit technology for an inspection setup, the system verifies the configured groups, and if one or more groups are not accurate for this technology (that is, the element quantity (aperture), the first and last element, and the UT channel values are higher than the value selected in the workspace), the system identifies the invalid values and offers to delete them or to cancel the operation.

8. Configuring the Part to Be Inspected

You can use the workspace to configure a specific part to be inspected. The part can be a plate, a tube, or a bar. A weld can also be configured, but this is optional.

8.1 Configuring the Part

When NDT SetupBuilder initially creates it, a workspace has a default mild steel plate with a simple symmetrical V weld (see Figure 8-1 on page 71).

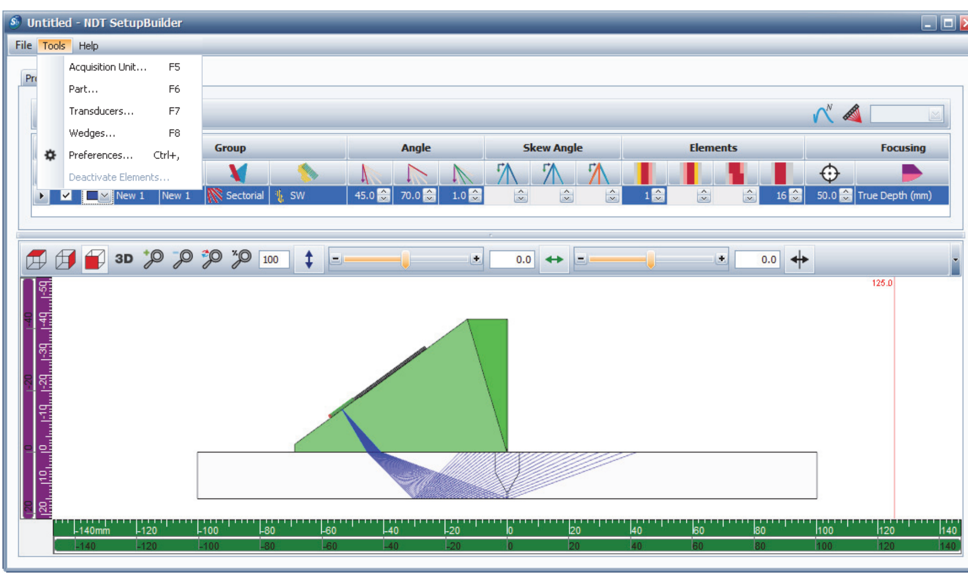


Figure 8-1 Default plate part with simple symmetrical V weld in RayTracing view

To configure the part to be inspected

1. On the **Tools** menu, click **Part**.
2. In the **Part Definition** dialog box, configure the part and weld parameters (see Figure 8-2 on page 72):

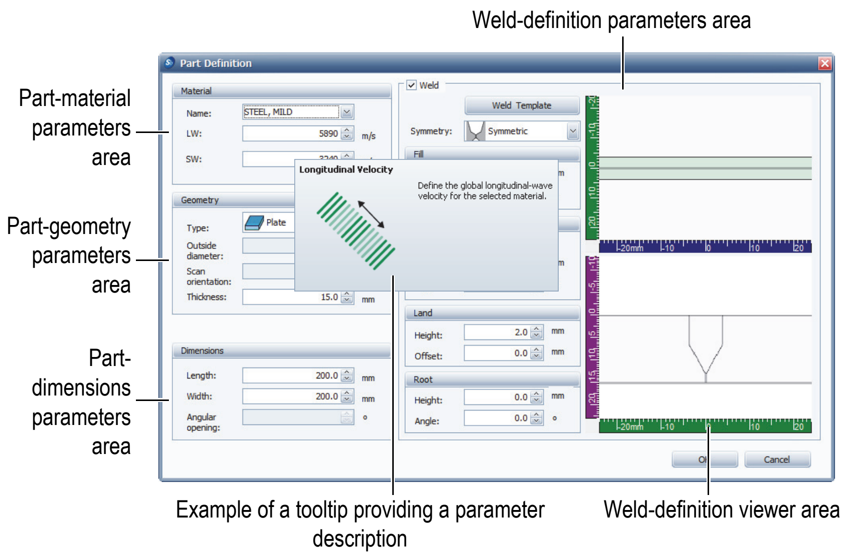


Figure 8-2 The Part Definition dialog box with default parameters

a) Define the part parameters (see “Configuring the Part” on page 71):

(1) Define the part material (see “Configuring the Part Material” on page 73).

(2) Define the part geometry (see “Configuring the Part Geometry” on page 74).

(3) Define the part size in RayTracing view (see “Configuring the Part Dimensions” on page 76).

b) Define the weld parameters (see “Defining the Weld to Be Inspected” on page 78).

NOTE

In the **Part Definition** dialog box, point the mouse to each parameter to display a tooltip with a short description.

3. Click **OK**.

After the part and weld have been designed and configured, they appear in RayTracing view (see “RayTracing View” on page 40).

8.2 Configuring the Part Material

The part material parameters are fundamental to the inspection technique, because they determine the characteristics of ultrasound wave propagation in the part during nondestructive testing.

The **Part Definition** dialog box can be used to configure the part material parameters. For more information about accessing the dialog box, see “Configuring the Part to Be Inspected” on page 71.

The **Material** area offers the following parameters:

Name

Click the **Name** list to select the material of which the part is composed. The default value is **STEEL, MILD** (mild 1020 steel).

Type a portion of or the complete name within the **Name** list in order to reduce the number of items displayed in the list.

When a material is selected, the appropriate ultrasound longitudinal wave (longitudinal) velocity and shear wave (transverse) velocity values are automatically configured in the **LW** and **SW** boxes. These values can be manually modified.

LW

In the **LW** box, enter the longitudinal wave (longitudinal) velocity of the ultrasound waves in the material (either in meters per second [m/s] or in inches per microsecond [in./µs], depending on the measurement unit selected in the NDT SetupBuilder preferences).

Longitudinal velocity is the speed of the ultrasound waves that have the same direction of vibration as their direction of travel, which means that the movement of the medium is in the same or opposite direction as the motion of the wave.

SW

In the **SW** box, enter the shear wave (transverse) velocity of the ultrasound waves in the material (either in meters per second [m/s] or in inches per microsecond [in./µs], depending on the measurement unit selected in the NDT SetupBuilder preferences).

Transverse velocity is the speed of the ultrasound waves that have a direction of vibration that is perpendicular to their direction of travel, which means that the movement of the medium is perpendicular to the direction of wave propagation.

NOTE

The material, longitudinal velocity, and transverse velocity parameters of a part can be customized in order to create a custom material. The custom material is available only in the workspace where it has been defined. After a material's parameters have been modified, an asterisk is displayed to the right of the **Material** list.

8.3 Configuring the Part Geometry

In a new workspace, the default part type is a plate. The following two options are available: tube and bar. Depending on the selected part type, you can configure the scan orientation (which is generally the same as the weld orientation, and only available for a tube or bar part), the part thickness, and/or the part diameter.

The **Part Definition** dialog box can be used to configure the part geometry parameters. For more information about accessing the dialog box, see "Configuring the Part to Be Inspected" on page 71.

The part geometry parameters have an influence on the probe set/wedge/group configuration. The **Geometry** area contains the following parameters:

Type 

Click the **Type** list to select the part shape type. Choose **Plate**, **Tube**, or **Bar** (see Figure 8-3 on page 75). The part type selected by default is **Plate**.

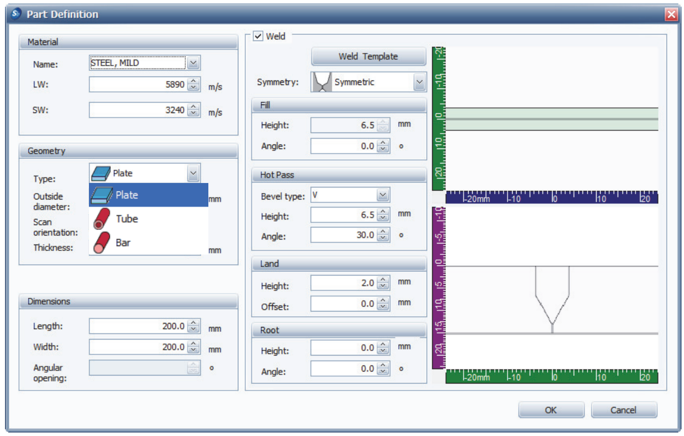
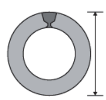
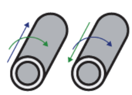


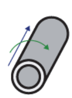
Figure 8-3 The Type list in the Part Definition dialog box

Outside diameter 

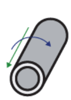
When **Tube** or **Bar** is selected in the **Type** list, in the **Outside diameter** box**,** type the outside diameter (OD) value of the part. The default value is 200 mm. The **Diameter** box is unavailable when **Plate** is selected in the **Type** (Geometry) list.

Scan orientation 

When **Tube** or **Bar** is selected in the **Type** list, click the **Scan orientation** list to select the orientation of the scan axis on a tube or bar part. The scan axis is defined by the scanner movement. The following options are available:

— **Circumferential** 

Choose **Circumferential** to inspect a path that runs along the circumference of the tube or bar part.

— **Axial** 

Choose **Axial** to inspect a path that runs along the length of the tube or bar part.

The weld orientation determines the scan axis on a tube or bar part for the inspection technique. The **Scan orientation** list is unavailable when **Plate** is selected from the **Type** list.

Thickness 

In the **Thickness** box, enter a value for the thickness of the part. The thickness is constant for the entire part, and is defined by the shortest distance between the outer and inner surfaces of a part. The default thickness value is 15 mm. The **Thickness** box is unavailable when **Bar** is selected from the **Type** list.

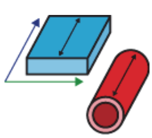
8.4 Configuring the Part Dimensions

The **Part Definition** dialog box can be used to configure the part dimension parameters.

When the RayTracing view is set to Top (C) view, Side (B) view, or End (D) view, the part dimension parameters define the length of the RayTracing view axes. When the RayTracing view is set to 3-D view, the part display-size parameters define the dimensions of the part in the 3-D view.

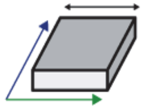
For more information about accessing the **Part Definition** dialog box, see "Configuring the Part to Be Inspected" on page 71.

The following parameters can be configured for the part display size:



Length

In the **Length** box, enter a value for the length of the part in RayTracing view. For a plate part, the length is parallel to the scan axis. For a tube or bar part, the length is measured along the cylinder's axis. In 2-D view, the part's length value will be split in half between negative and positive values on the RayTracing view axes. The default value is 200 mm.

Width 

When **Plate** is selected in the **Type** list (under **Geometry**), in the **Width** box, type a value for the total index axis width in RayTracing view. In 2-D view, the width of the index axis will be split in half between negative and positive values on the RayTracing view axes. The default value is 200 mm.

The **Width** box is unavailable when **Tube** or **Bar** is selected in the **Type** list under **Geometry** (see Figure 8-4 on page 77).

NOTE

When the RayTracing view is set to 3-D view, the values entered in the **Length** and **Width** boxes determine the size of the displayed plate part.

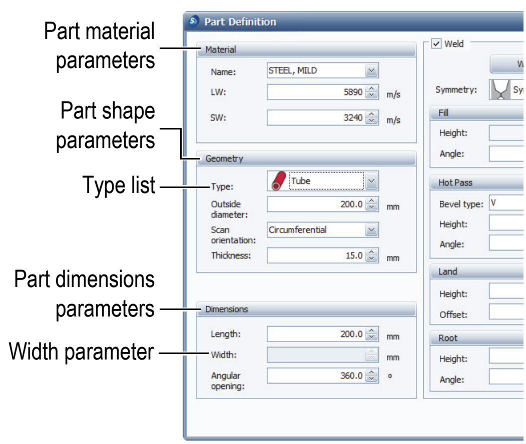
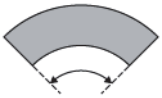


Figure 8-4 RayTracing view automatic Width parameter adjustment example

Angular opening 

In the **Angular opening** box, for a tube part, enter a value in degrees for the portion of the tube section that is to be displayed in RayTracing view. The default value is 360°, which means the entire section of the tube part is displayed.

The **Angular opening** box is unavailable when **Plate** or **Bar** is selected in the **Type** list under **Geometry**.

NOTE

When the RayTracing view is set to 3-D view, the values entered in the **Length** and **Angular opening** boxes determine the size of the displayed tube part.

8.5 Defining the Weld to Be Inspected

If you want to configure the weld to be inspected using a technique defined in the workspace, there are six predefined configurations available in the **Weld Template** list. Three symmetry options are available in the **Symmetry** list. The weld dimension parameters can be modified individually for the inspection technique's requirements. The weld representation is dynamically updated in the graphical representation zone located on the right-hand side of the **Part Definition** dialog box.

8.5.1 Weld Regions

NDT SetupBuilder provides setup parameters that can be used to precisely define the different regions of a weld. The weld regions and their parameters are defined as follows (see Figure 8-5 on page 79):

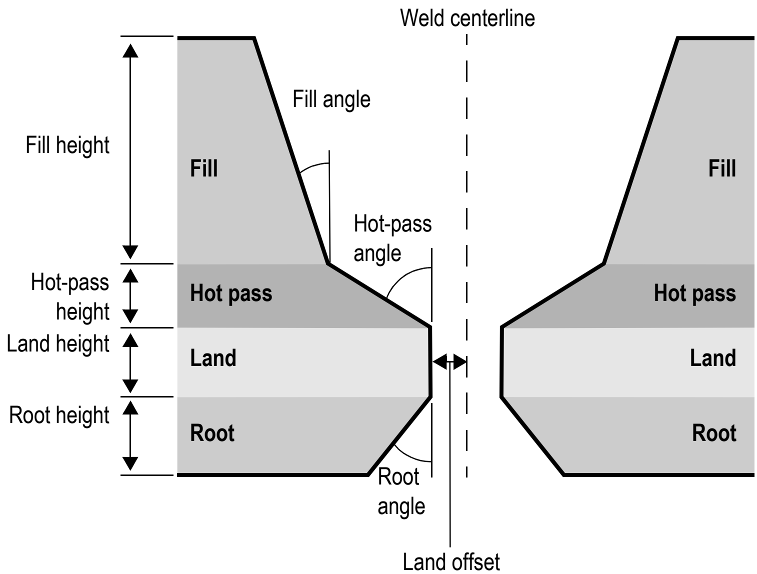


Figure 8-5 The parts and parameters of a weld definition

- **Fill**: The topmost region of a weld. For a tube, this region is closest to the outside diameter (OD). This region extends at an angle from the hot pass to the top of the piece. A fill with a zero-degree (0°) angle relative to the weld centerline is vertical.

  In NDT SetupBuilder, the parameters that define the fill region are the fill **Height** and **Angle**.

NOTE

In some actual weld definitions, the fill may be subdivided into fill and upper fill regions, or it may be subdivided into several regions, each with a specific fill angle and fill height. The upper fill region is also known as the cap. NDT SetupBuilder supports only one fill.

- **Hot pass**: The region just below the fill of the weld. This region receives the second pass during the welding process. The hot pass can be of two bevel types (see Figure 8-6 on page 80):

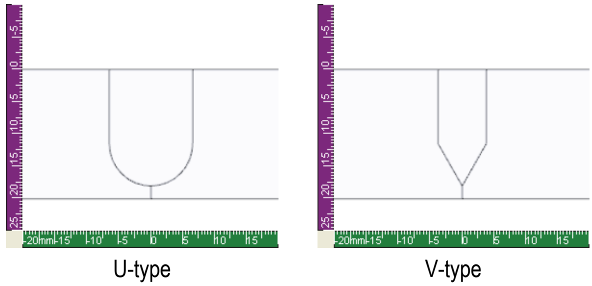


Figure 8-6 Hot-pass types

- **U-type**: Defined by a radius measured between the hot pass and the fill. The radius extends to the point where it becomes tangential to the fill region (where it crosses, or touches the fill region's angle). In NDT SetupBuilder, the fill walls are vertical when the angle is zero degrees (0°) relative to the weld centerline. The U-type hot pass as defined here is also known as a J-type hot pass.
- **V-type**: The V-type hot pass is comprised of an angled wall, the height of which is measured between the hot pass and the fill.

In NDT SetupBuilder, the parameters that define the hot-pass region are the hot-pass **Bevel type**, **Height**, and **Angle**.

- **Land**: This is the vertical region above and below the area in which the first pass is placed during the welding process. The land region itself is not welded. The weld walls join together in this region. An offset or a gap may optionally separate the walls of the land region, and may include an empty space when the weld is completed.

  In NDT SetupBuilder, the parameters that define the land region are the land **Height** and **Offset**.

NOTE

In CRC-type bevels, the land region is known as *lack of cross-penetration (LCP)*.

- **Root**: This region extends at an angle from below the land region to the bottom of the piece. For a tube, this region is closest to the inside diameter (ID). This is the region where the root pass is placed during the welding process. The root is not always part of a weld definition.

In NDT SetupBuilder, the parameters that define the root region are the root **Height** and **Angle**.

NOTE

The root of the weld is also known as the root fill.

8.5.2 Basic Weld Shape Templates

NDT SetupBuilder provides the following six basic weld shape templates (see Figure 8-7 on page 81):

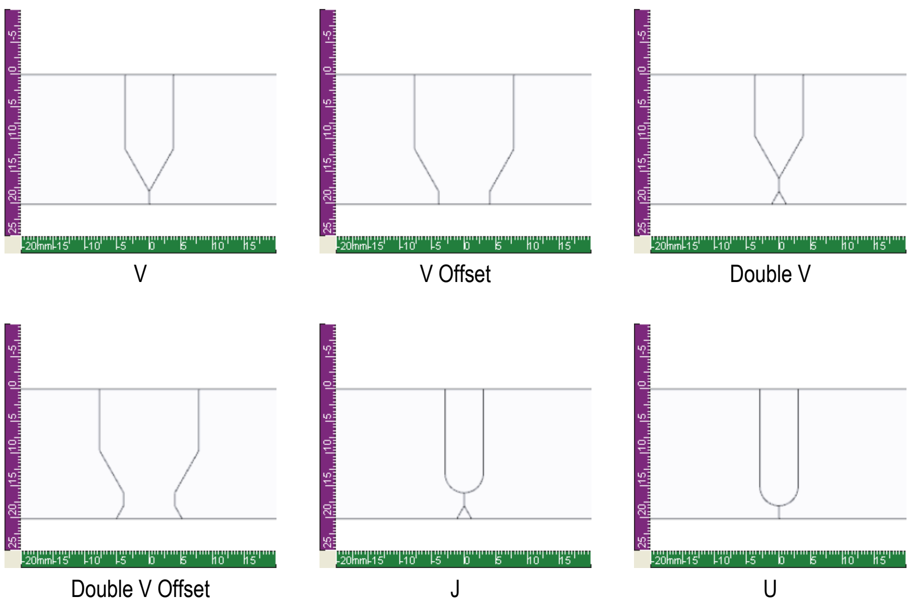


Figure 8-7 Available weld shape templates

- **V**: A weld type comprised of a fill, a V-shaped hot pass, and a land. There is no predefined root.
- **V Offset**: A weld type comprised of a fill, a V-shaped hot pass, and a land with an offset or gap. There is no predefined root.

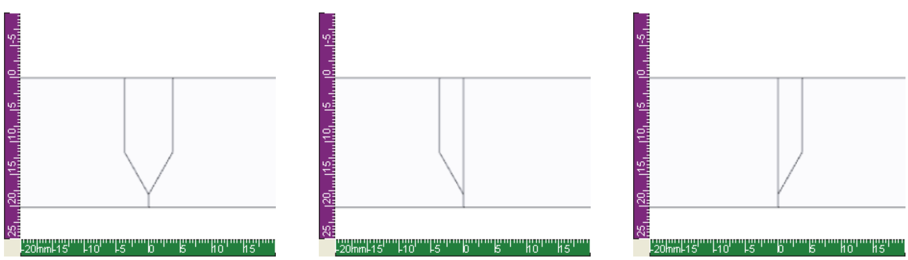
- **Double V**: A weld type comprised of a fill, a V-shaped hot pass, a land, and a V-shaped root.
- **Double V Offset**: A weld type comprised of a fill, a V-shaped hot pass, a land with an offset or gap, and a V-shaped root.
- **J**: A weld type comprised of a fill with facing vertical walls, a J-shaped hot pass, a land, and a V-shaped root.
- **U**: A weld type made of a fill, a U-shaped hot pass, and a land. There is no predefined root.

NOTE

The **Double V**, **Double V Offset**, and **J** weld templates provide a predefined root region.

8.5.3 Weld Symmetry

In NDT SetupBuilder, there are three different types of weld symmetry (see Figure 8-8 on page 82):



Symmetrical   Flat Right-Hand Side (RHS)   Flat Left-Hand Side (LHS)

Figure 8-8 Available weld symmetry options

- **Symmetrical**: Weld type with symmetrical wall configurations along the weld center line.
- **Flat Right-Hand Side (RHS)**: Asymmetrical weld type, with one vertical flat wall parallel to the weld center line on the right-hand side, and appropriate fill, hot pass, land, and root walls on the left-hand side.

- **Flat Left-Hand Side (LHS)**: Asymmetrical weld type, with one vertical wall parallel to the weld center line on the left-hand side, and appropriate fill, hot pass, land, and root walls on the right-hand side.

NOTE

The flat right-hand side (RHS) and flat left-hand side (LHS) weld types are also known as K-type welds.

8.5.4 Weld Configuration

In NDT SetupBuilder, the weld is an optional parameter under **Part Definition**. This means that you are able to configure an inspection setup with a part that does not include a weld.

The weld is defined by configuring the parameters for the different weld regions (see “Weld Regions” on page 78) in the **Part Definition** dialog box (see Figure 8-9 on page 84). For more information about accessing the dialog box, see “Configuring the Part to Be Inspected” on page 71.

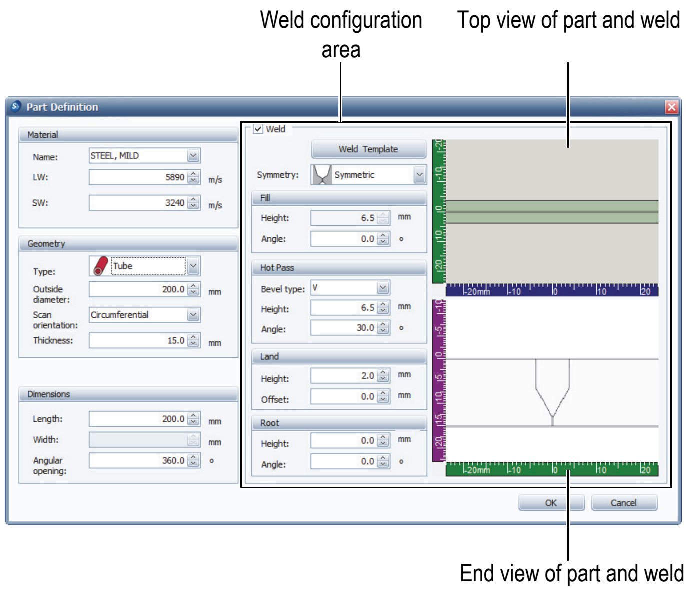


Figure 8-9 The Part Definition dialog box with the Weld area

In the **Part Definition** dialog box, two views (Top view and an End view) display the shape of the configured weld, and are updated dynamically as parameters are set or modified.

NOTE

Only positive values can be entered for weld parameters. Regions with a height parameter value set to 0 (zero) are not displayed in the weld definition.

Configure the following parameters for the weld regions applicable to the inspection setup:

Weld

Select the **Weld** check box if you want to include a weld definition for the part defined in the inspection setup, and to make the weld parameters available in the **Part Definition** dialog box. By default, the **Weld** check box is selected for a plate or a tube part.

Clear the **Weld** check box to define a part without a weld. When the **Weld** check box is cleared, the weld parameters are unavailable in the **Part Definition** dialog box.

Weld Template

In the **Weld Template** list, select a weld template with the appropriate weld symmetry for your inspection setup (see Figure 8-10 on page 85):

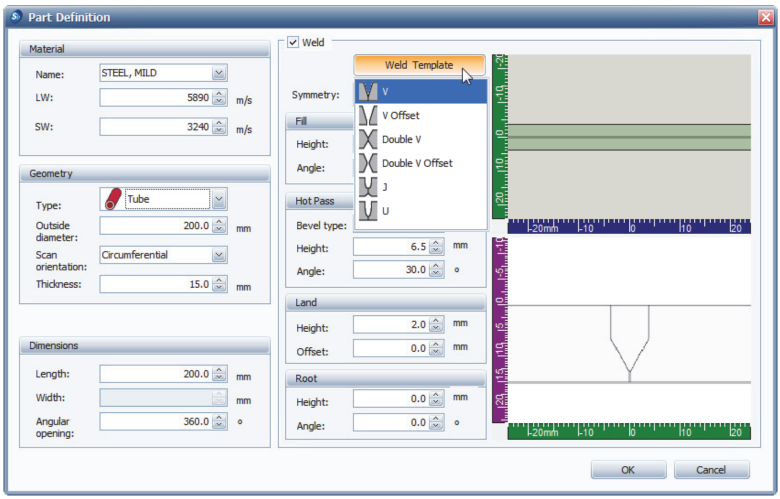


Figure 8-10 The Weld Template list

— **V**
— **V Offset**
— **Double V**
— **Double V Offset**
— **J**
— **U**

The weld templates available in NDT SetupBuilder correspond to those available with OmniScan instruments. For more information about the available weld templates, see "Basic Weld Shape Templates" on page 81.

By default, the **V** weld template is selected. Selecting a weld template sets the original values of all of the weld parameters, and provides a basis on which to refine your weld configuration in line with the inspection setup.

NOTE

By default, only the **Double V**, **Double V Offset**, and **J** weld templates offer a predefined root region.

Symmetry

In the **Symmetry** list, select the appropriate weld symmetry:

— **Symmetrical**

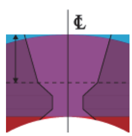
— **Flat Right-Hand Side (RHS)**

— **Flat Left-Hand Side (RHS)**

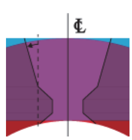
By default, the **Symmetrical** type is selected in the **Symmetry** list. For more information about the available weld symmetry types, see “Weld Symmetry” on page 82.

Fill

In the **Fill** area, the following two parameters define the fill region:

Height 

To configure the fill region’s height, enter a value in the **Height** box. This measurement is always taken parallel to the weld center line.

Angle 

To configure the wall angles of the fill region, enter a value in the **Angle** box. The fill angle is defined as the angle between the fill bevel and a line parallel to the center line.

Hot Pass

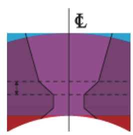
The the **Hot Pass** area contains the following three parameters, which define the hot-pass region:

Bevel Type 

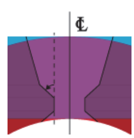
In the **Bevel Type** list, select the appropriate hot-pass bevel type: choose **U** or **V**. For more information about hot-pass bevel types, see “Weld Regions” on page 78.

NOTE

A hot-pass bevel type is originally set according to the weld type selected in the **Weld template** list. The **Bevel Type** list provides additional options that can be used to define the hot pass more precisely.

Height 

To configure the hot-pass region’s height, enter a value in the **Height** box. This measurement is always taken parallel to the weld center line.

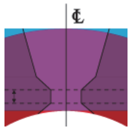
Angle 

To configure the wall angles of the hot-pass region, enter a value in the **Angle** box. The hot-pass angle is defined as the angle between the hot-pass bevel and a line parallel to the weld center line. The maximum usable value for the hot-pass region wall angle is 89.9°, and the minimum height value is 0.0 mm. A hot-pass height value of 0.0 mm means that no hot pass is defined. A minimal value greater than 0.0° is needed to define the hot pass angle.

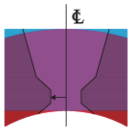
When the bevel type selected in the **Bevel Type** list is **U**, the **Angle** box is unavailable.

Land

In the **Land** area, the following two parameters define the land region:

Height 

To configure the land region's height, enter a value in the **Height** box. This measurement is always taken parallel to the weld center line.

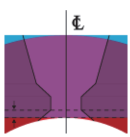
Offset 

To configure the gap between one of the land region's walls and the weld center line, enter a value in the **Offset** box. The total gap measurement between the two walls of the land is two times the value displayed in the **Offset** box. Specify the minimal distance between either side of the weld and the center line. This measurement is always taken perpendicular to the weld center line.

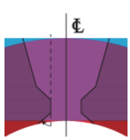
When the bevel type selected in the **Bevel Type** list is **U**, the **Offset** box is unavailable.

Root

In the **Root** area, the following two parameters define the root region:

Height 

To configure the root region's height, enter a value in the **Height** box. This measurement is always taken parallel to the weld center line.

Angle 

To configure the wall angles of the root region, enter a value in the **Angle** box.

NOTE

By default, in the **Root** area, the **Height** box and the **Angle** box parameters have a value of 0, except when the template selected in the **Weld Template** list is **Double V**, **Double V Offset**, or **J**. Only the **Double V**, **Double V Offset**, and **J** weld templates offer a predefined root region.

9. Configuring One or More Probe Sets

In NDT SetupBuilder, the probe sets for a given inspection setup are configured from the **Probe Sets** tab. The **Probe Sets** tab provides tools and parameters that can be used to configure the probes and wedges included in the probe sets.

The **Probe Sets** tab contains a toolbar that provides basic probe-set definition commands (see Figure 9-1 on page 91).

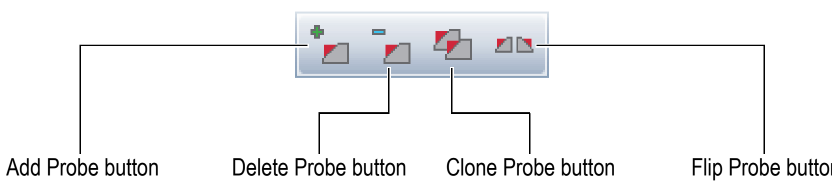


Figure 9-1 The Probe Sets tab toolbar

TIP

Several of the **Probe Sets** tab's commands can be activated using keyboard shortcuts. For more information, see "Accessing Commands Using Keyboard Shortcuts" on page 31.

The **Probe Sets** tab columns provide parameters that can be used to configure the probe sets:

- To define probe sets, see:
  — "Setting Probe-Set Visibility" on page 97.

— “Configuring Probe-Set Position Parameters” on page 130.
- To define a probe set’s probes, see:
  — “Selecting a Probe Series and Model” on page 100.
  — “Managing Custom Probe Models” on page 103.
- To define a probe set’s wedge, see:
  — “Selecting a Wedge Series and Model” on page 114.
  — “Managing Custom Wedge Models” on page 116.

NOTE

On the **Probe Sets** tab, point the mouse to each column header or toolbar element to display a tooltip with a short description of the applicable parameter or function.

RayTracing view can be used to visualize the probe sets being configuring for a 2-D or 3-D inspection setup, with the exception of dual matrix-type probe sets, which are not displayed. For more information, see “RayTracing View” on page 40.

When a probe set is selected on the **Probe Sets** tab, it becomes highlighted in RayTracing view. For more information, see “Identifying Selected Probe Sets in RayTracing View” on page 32.

The tools needed to configure probe-set groups are located on the **Groups** tab. For more information about configuring groups, see “Configuring One or More Groups” on page 137.

9.1 Adding a Probe Set

The **Probe Sets** tab toolbar can be used to insert one or more probe sets in an inspection setup. The probe-set parameters can also be configured from the **Probe Set** section.

To add a probe set

◆ Click the **Add Probe** button () on the **Probe Sets** tab toolbar.

A new probe set is added in the **Probe Sets** tab table. This table can be used to set or modify the following probe-set definition parameters: **Visible**, **Name**, and

Type. For more information about these parameters, see “Configuring the Probe-Set Section Parameters” on page 99.

When a new probe set of the TOFD or UT type is added, an associated group is also automatically created on the **Groups** tab. Adding a probe set of the PA linear or dual matrix type does not automatically create a group. For more information about creating and configuring groups, see “Configuring One or More Groups” on page 137.

9.2 Deleting a Probe Set

The **Probe Sets** tab toolbar can be used to delete one or more probe sets in an inspection setup.

To delete a probe set

1. In the list under the **Probe Sets** tab, select one or more probe sets you want to delete (see Figure 9-2 on page 93).

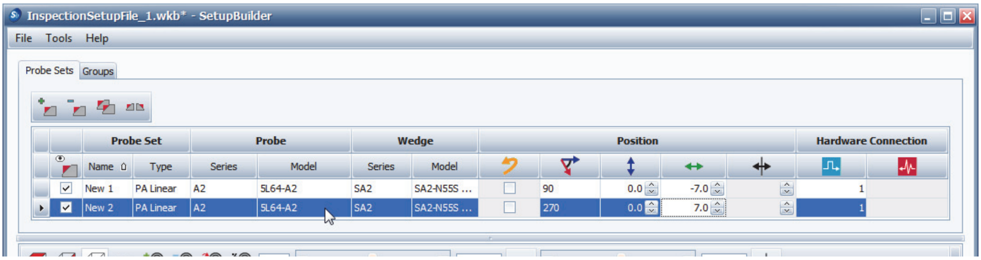


Figure 9-2 Selecting a probe set on the Probe Sets tab for deletion

2. Click the **Delete Probe** button ( ) on the **Probe Sets** tab toolbar.

   The selected probe sets are deleted from the list under the **Probe Sets** tab.

NOTE

When you delete a probe set, all the groups associated with it are simultaneously deleted from the **Groups** tab.

9.3 Cloning a Probe Set

The **Probe Sets** tab toolbar can be used to clone one or more probe sets in an inspection setup. When cloning a probe set, the groups associated with it are also cloned on the **Groups** tab.

To clone a probe set

1. In the list under the **Probe Sets** tab, select the probe set or sets you want to clone (see Figure 9-3 on page 94).

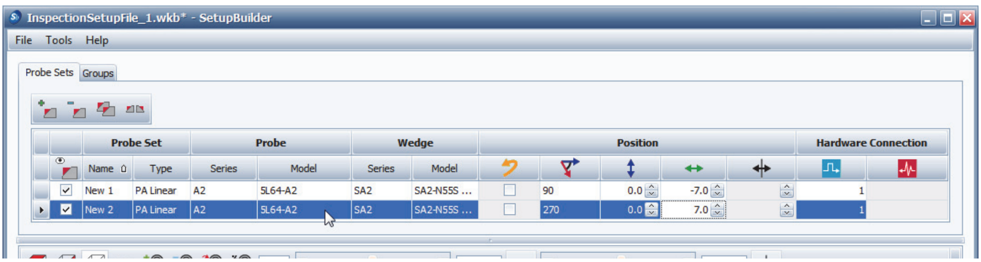


Figure 9-3 Selecting a probe set on the Probe Sets tab for cloning

2. Click the **Clone Probe** button () on the **Probe Sets** tab toolbar.

   The cloned probe sets are added to the list under the **Probe Sets** tab.

TIP

Use the **Clone Probe** button () in combination with the **Flip Probe** button () on the **Probe Sets** tab toolbar to create a pair of opposing probe sets. For more information about the **Flip Probe** button, see “Flipping a Probe Set” on page 95.

9.4 Flipping a Probe Set

The **Probe Sets** tab toolbar can be used to flip one or more probe sets in an inspection setup. Flipping a probe set switches the skew parameter values between 0° and 180°, or 90° and 270°, and inverts the sign of the index offset parameter value. For more information about the index offset parameter, see “Configuring Probe-Set Position Parameters” on page 130.

To flip a probe set

1. In the list under the **Probe Sets** tab, select one or more probe sets you want to flip (see Figure 9-4 on page 95).

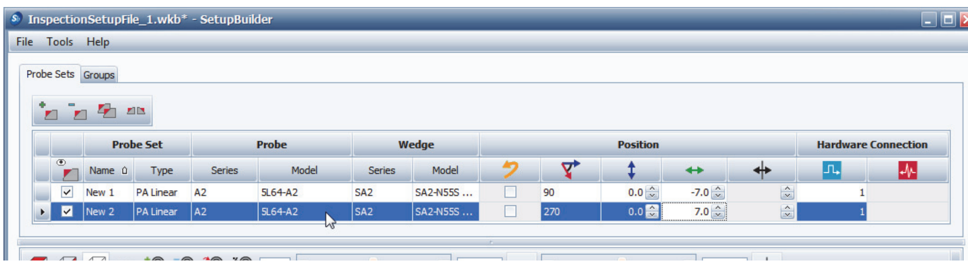


Figure 9-4 Selecting a probe set on the Probe Sets tab for flipping

2. Click the **Flip Probe** button ( ) on the **Probe Sets** tab toolbar.

   The selected probe sets’ skew and index offset parameters are modified in the list under the **Probe Sets** tab.

TIP

Use the **Flip Probe** button ( ) in combination with the **Clone Probe** button ( ) on the **Probe Sets** tab toolbar to create a pair of opposing probe sets. For more information about the **Clone Probe** button, see “Cloning a Probe Set” on page 94.

9.5 Configuring the Parameters of Probe Sets

When defining or modifying a probe set on the **Probe Sets** tab, it is also possible to configure the available parameters identified on the tab's column headers (see Figure 9-5 on page 96).

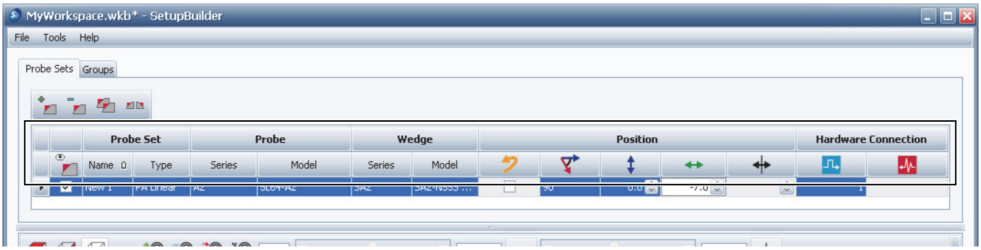


Figure 9-5 The Probe Sets tab column headers

To configure probe-set parameters

1. From the list under the **Probe Sets** tab, select the probe set you want to configure (see Figure 10-12 on page 146).

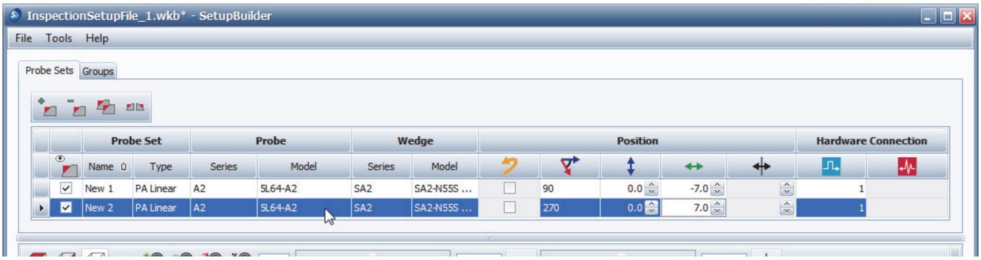


Figure 9-6 Selecting a probe set on the Probe Sets tab to configure parameters

2. In the **Probe Set** section, set the probe-set visibility. For more information, see "Setting Probe-Set Visibility" on page 97.
3. In the **Probe Set** section, configure the probe-set name and type parameters. For more information, see "Configuring the Probe-Set Section Parameters" on page 99.

4. In the **Probe** section, select the probe-set probe series and models. For more information, see “Selecting a Probe Series and Model” on page 100.
5. In the **Wedge** section, select the wedge series and models. For more information, see “Selecting a Wedge Series and Model” on page 114.
6. In the **Position** section, configure the probe-set position parameters. For more information, see “Configuring Probe-Set Position Parameters” on page 130.
7. In the **Hardware Connection** section, configure the probe-set hardware connection parameters. For more information, see “Setting Up the Probe-Set Hardware Connection” on page 133.

9.5.1 Setting Probe-Set Visibility

In RayTracing view, the probe-set visibility can be set using the check box in the **Visible** column () on the left side of the probe-set parameters table under the **Probe Sets** tab. By default, a probe set is visible and the check box selected upon creation (see Figure 9-7 on page 98). If a probe set’s **Visible** check box is cleared, the groups associated with the probe set also become invisible, and they cannot be made visible in RayTracing view unless the probe set is displayed.

It is possible to independently set the visibility of the groups associated with a visible probe set. For more information, see “Setting Group Visibility” on page 144.

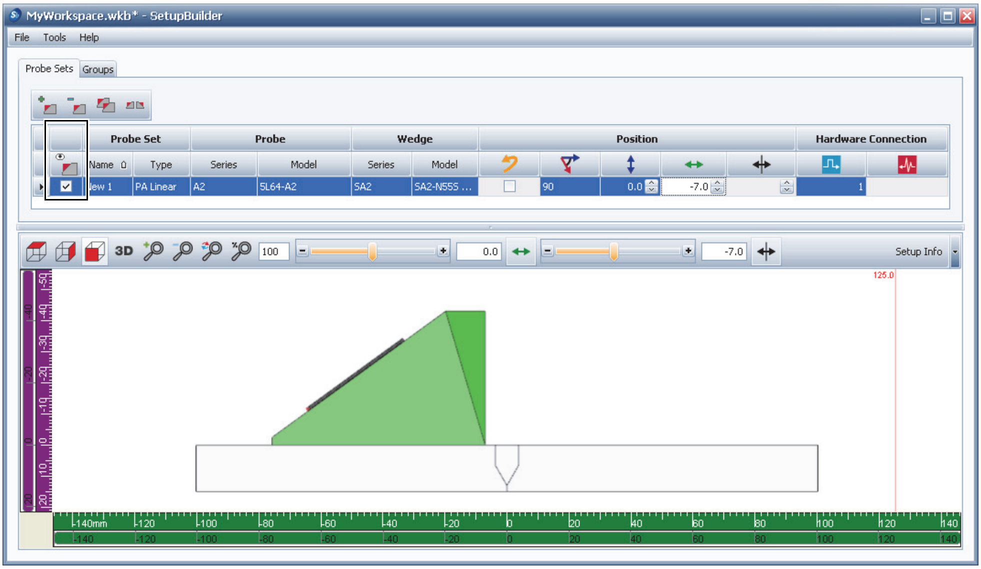


Figure 9-7 The Probe Sets tab with the Visible check box selected

To set the probe-set visibility

- Do one of the following:
  - To show the probe set and all related groups in RayTracing view, select the **Visible** column ( ) check box (see Figure 9-7 on page 98).
  - To hide the probe set and all related groups in RayTracing view, clear the **Visible** column ( ) check box (see Figure 9-8 on page 99).

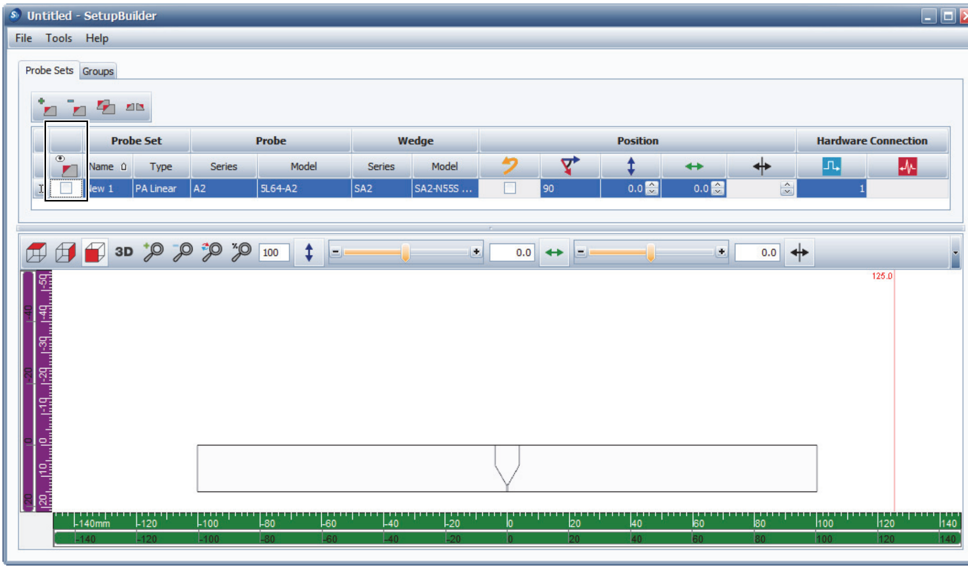


Figure 9-8 A probe set with the Visibility check box cleared on the Probe Sets tab

9.5.2 Configuring the Probe-Set Section Parameters

The **Probe Sets** tab can be configured using the probe-set parameters in the **Probe Set** section.

NOTE

The items available in the **Type** list in the **Probe Set** section change based on the acquisition unit selected for the inspection setup. For more information about selecting an acquisition unit, see "Selecting an Acquisition Unit" on page 65.

To configure the probe-set section parameters

◆ On the **Probe Sets** tab, under **Probe Set**, configure the following parameters:

Name

In the **Name** box, enter a name for the probe set. By default, the name **New *n*** is automatically generated. The probe-set name can contain a maximum of 15 characters.

Type

In the **Type** list, select the appropriate type of probe set to be used in the inspection setup:

- **PA Linear**: For phased array inspection. By default, **PA Linear** is selected in the list.
- After creating a PA linear-type probe set, you must manually create one or more groups and associate them with the probe set.
- **TOFD**: For time-of-flight diffraction inspection.
- When you create a TOFD-type probe set, a pitch-catch group is automatically created and associated with the probe set.
- **UT**: For ultrasound inspection.
- When you create a UT-type probe set, a single pulse-echo group is automatically created and associated with the probe set.
- **Dual Matrix**: for phased array dual inspection.
- When you create a dual matrix-type probe set, you must manually create one or more groups and associate them with the probe set.

The type of probe set selected in the **Type** list determines which items are displayed in the **Series** list under **Probe**.

NOTE

When you change the **Type** of a probe set, the groups that are associated with it are deleted from the **Groups** tab.

For more information about creating and configuring groups, see “Configuring One or More Groups” on page 137.

9.5.3 Selecting a Probe Series and Model

NDT SetupBuilder provides a list of available Evident probe series and models, and it is possible to select a probe series and model from the predefined lists. The list of available probe series and models only displays items that are compatible.

If the probe series and models available in the predefined lists do not accurately reflect the probe required for your inspection setup, you can define a customized probe. Custom probes are created using the **Probes** command from the **Tools** menu (see “Managing Custom Probe Models” on page 103). Custom probes are displayed separately in the list.

NOTE

In the **Series** list, select **Custom** to display the list of custom probes available in the **Model** list under the **Probe Sets** tab. You can then select the appropriate existing custom probe to edit it.

To select a probe series and model

- On the **Probe Sets** tab, under **Probe**, select a probe series and model from the following lists:

  Series

  In the **Series** list, select the appropriate series of probe to be used in the probe set:

  - **Angle Beam**: Select **Angle Beam** to display the list of existing angle-beam probes in the **Model** list.
  - **Immersion**: Select **Immersion** to display the list of existing immersion probes in the **Model** list.
  - **Contact**: Select **Contact** to display the list of existing contact probes in the **Model** list.
  - **Dual**: Select **Dual** to display the list of existing pitch-catch (pulser-receiver) probes in the **Model** list.
  - **Straight Beam**: Select **Straight Beam** to display the list of straight-beam probes in the **Model** list.
  - **Integrated Wedge**: Select **Integrated Wedge** to display the list of existing integrated-wedge probes in the **Model** list.
  - **Custom**: Select **Custom** to display the list of existing custom probes in the **Model** list. To edit a custom probe or use it in the inspection setup, select it from the **Model** list.

  The probe-series list displays only those series that are compatible with the probe-set type selected in the **Type** list under **Probe Set**.

NOTE

For PA linear-type and dual matrix-type probe sets, the **Series** list provides a complete listing of the different series of probe mounts available in the Evident probe and wedge catalog.

Model 

In the **Model** list, select the appropriate probe model to be used in the probe set.

The list of probe models displays only those models that are compatible with the probe-set type selected in the **Type** list under **Probe Set** and the probe series selected in the **Series** list under **Probe**.

9.5.4 Identifying the First Element on a Probe

The first element of a probe is visually identified by a red outline in RayTracing view (see Figure 9-9 on page 103). You can reverse the position of the first element of a probe in conjunction with certain wedges from the wedge series. For more information, see “Configuring Probe-Set Position Parameters” on page 130.

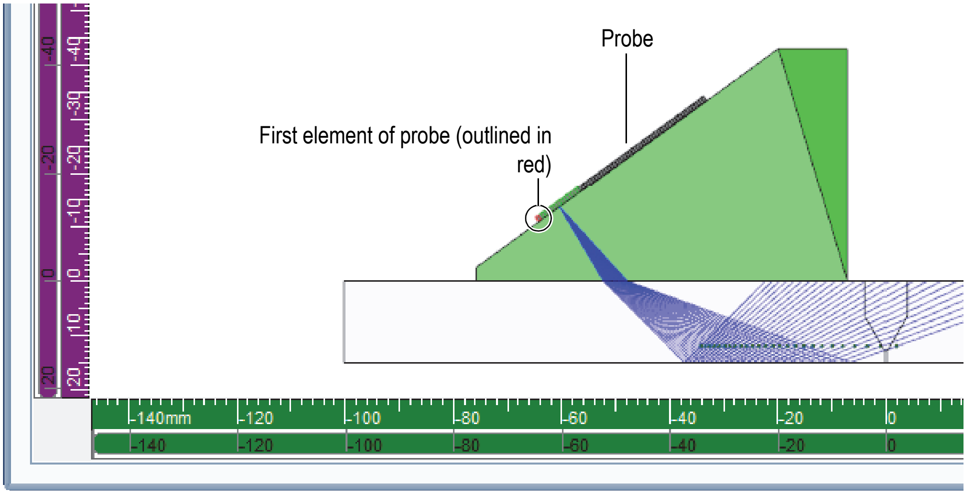


Figure 9-9 The first element of a probe is identified in RayTracing view

9.5.5 Managing Custom Probe Models

The NDT SetupBuilder workspace allows you to define custom probes that accurately comply with your inspection setup, or delete custom probes. On the **Probe Sets** tab, under **Probe**, existing custom probe models are displayed in the **Model** list when **Custom** is selected in the **Series** list. The custom probes created are then saved to the open workspace.

To manage custom probe models

1. On the **Tools** menu, click **Transducers** to access the probe settings (see Figure 9-10 on page 104).

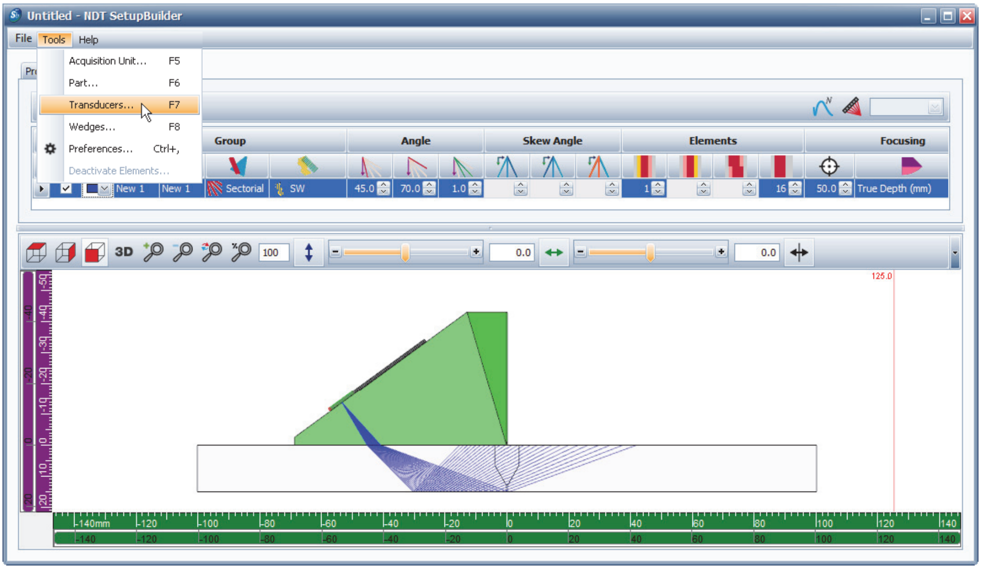


Figure 9-10 Accessing the probe settings on the Tools menu

The **Manage Probes** dialog box appears (see Figure 9-11 on page 105). This dialog box contains the following three areas:

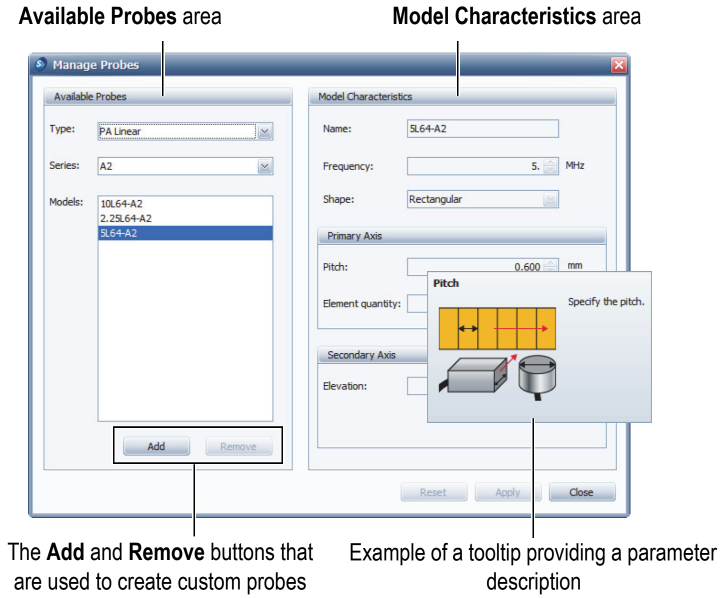


Figure 9-11 The Manage Probes dialog box

NOTE

In the **Manage Probes** dialog box, point the mouse to each parameter to display a tooltip with a short description.

— **Available Probes**: This area provides lists that allow you to select a probe from predefined probe-set types, probe series, and probe models. The **Add** and **Remove** buttons can be used to create a custom probe based on a predefined model selected from the lists or to delete an existing custom probe.

— **Model Characteristics**: This area displays the parameters of the probe model selected in the **Available Probes** area. For a predefined probe, these parameters are unavailable. When you create a custom probe model, the parameters become editable.

— Probe parameters diagram: This area identifies the probe's primary axis, secondary axis, width, and pitch parameters.

2. Under **Available Probes**, select a probe to use as the basis for your custom probe:

   Type

   In the **Type** list, select the appropriate type of probe set you want to use in the inspection setup:

   - **PA Linear**: For phased array inspection. By default, **PA Linear** is selected in the list.
   - **UT**: For ultrasound inspection.
   - **TOFD**: For time-of-flight diffraction inspection.

   The type of probe set selected in the **Type** list determines the probe options displayed in the **Series** list.

   Series

   In the **Series** list, select the appropriate probe series to be used:

   - **Angle Beam**: Select **Angle Beam** to display the list of existing angle-beam probes in the **Models** list.
   - **Contact**: select **Contact** to display the list of existing contact probes in the **Model** list.
   - **Dual**: Select **Dual** to display the list of existing pitch-catch (pulser-receiver) probes in the **Models** list.
   - **Immersion**: Select **Immersion** to display the list of existing immersion probes in the **Models** list.
   - **Straight Beam**: Select **Straight Beam** to display the list of straight-beam probes in the **Models** list.
   - **Integrated Wedge**: Select **Integrated Wedge** to display the list of existing integrated wedge probes in the **Models** list.
   - **Custom**: Select **Custom** to display the list of existing custom probes in the **Models** list. You can then edit the appropriate custom probe by selecting it from the **Models** list.

     The **Custom** probe series will not display unless there is at least one existing custom probe model saved within the workspace. For more information, see “Configuring Custom Probe Model Parameters” on page 108.

   The **Series** list displays only those options that are compatible with the probe type selected in the **Type** list.

NOTE

For PA linear-type probe sets, the **Series** list provides a complete listing of the different series of probe mounts available in the Evident probe and wedge catalog, and the default probe set series is **A2**.

Models 

In the **Models** list, select the appropriate probe model to be used.

The list of probe models displays only those models that are compatible with the probe-set type selected in the **Type** list, and the probe series selected in the **Series** list.

3. Do one of the following:

   — To delete a custom probe, select it in the **Models** list under **Available Probes**, and then click **Remove** (see Figure 9-12 on page 107).

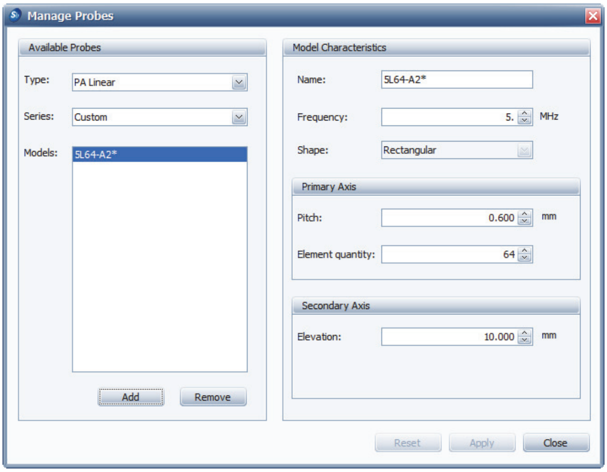


Figure 9-12 The Manage Probes dialog box and a new probe

The selected custom probe is deleted from the **Models** list.

- To modify a custom probe, select it in the **Models** list under **Available Probes**.

  The parameters of the selected custom probe will now be available under **Model Characteristics**.
- To add a custom probe, click **Add**.

  When a custom probe is selected in the **Models** list, it is automatically populated in the **Name** box under **Model Characteristics** (see Figure 9-12 on page 107). The name of the new probe is marked by an asterisk and it can be edited. The parameters under **Model Characteristics** become available.

4. Configure the probe parameters according to the specifications for the inspection setup.

   For details on configuring probe parameters when creating or modifying a custom probe model, see “Configuring Custom Probe Model Parameters” on page 108.
5. Perform one of the following actions:
   - Click **Reset** to reset the selected custom probe model parameters in the **Models** list under **Available Probes**.
   - Click **Apply** to save the custom probe model in the **Models** list under **Available Probes**.
   - Click **Close** to save the custom probe model in the **Models** list under **Available Probes** and close the dialog box.

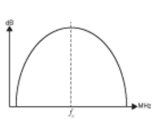
9.5.6 Configuring Custom Probe Model Parameters

The **Manage Probes** dialog box can be used to create, modify, or delete custom probes. For more information about managing probes and accessing the dialog box, see “Managing Custom Probe Models” on page 103.

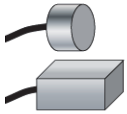
The following parameters can be configured in the **Manage Probes** dialog box:

Name

In the **Name** box, enter the name of the custom probe. By default, the name **New *n*** is automatically generated for custom probes. The probe name can contain a maximum of 15 characters.

Frequency 

In the **Frequency** box, enter the frequency of the custom probe.

Shape 

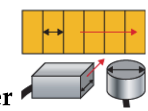
For a UT-type probe set only, in the **Shape** list, select the custom probe's element shape: **Rectangular** or **Circular**. By default, **Rectangular** is selected in the list.

NOTE

For a probe set of the PA linear or TOFD type, the **Shape** list is unavailable, and its value is set to **Rectangular**.

Primary Axis

In the **Primary Axis** area, configure the **Pitch/Side 1 length/Diameter** and **Element quantity** settings for your custom probe model definition:

Pitch/Side 1 length/Diameter 

The name in this box changes according to the type of probe set selected in the **Type** list under **Available Probes**. For TOFD or UT probe-set types, the box's name also changes according to the shape of the probe displayed in the **Shape** box under **Model Characteristics**.

For a probe set of the PA linear type, in the **Pitch** box, enter the element pitch, which is the spacing (center-to-center distance) between consecutive probe elements on the primary axis of the custom probe model.

In the **Side 1 length** box, for a probe set of the TOFD or UT type with a rectangular shape, enter the length of the element on the primary axis of the custom probe model.

In the **Diameter** box, for a probe set of the TOFD or UT type with a circular shape, enter the diameter of the custom probe model.

NOTE

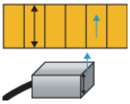
In NDT SetupBuilder, the width value of a single element of a probe is used as the element pitch value.

Element quantity

In the **Element quantity** box, enter the number of elements that comprise the probe on the primary axis of the custom probe model.

Secondary Axis

In the **Secondary Axis** area, configure the **Elevation/Side 2 length** setting for your custom probe definition:

Elevation/Side 2 length 

The name in this box changes according to the type of probe set selected in the **Type** list under **Available Probes**. For TOFD or UT probe-set types, the box's name also changes according to the shape of the probe displayed in the **Shape** box under **Model Characteristics**.

For a PA linear-type probe set, in the **Elevation** box, enter the elevation (the *element width*) of the elements of the custom probe model along the secondary axis. For a probe set of the TOFD or UT type with a circular shape, enter the diameter of the custom probe model.

For a probe set of the TOFD or UT type with a rectangular shape, in the **Side 2 length** box, enter the length of the element on the secondary axis of the custom probe model.

9.5.7 Deactivating and Activating Phased Array Probe Elements

When configuring an inspection setup for a PA linear- or dual matrix-type probe set, NDT SetupBuilder provides functions to individually deactivate or activate elements from the PA linear or dual matrix probes already created in your workspace. The functions for deactivating or reactivating probe elements are unavailable for the other types of probes.

Deactivating probe elements is especially useful when one or more of a probe's elements have become nonfunctional, or *dead*. The nonfunctional elements can be deactivated from within the inspection setup configuration using the **Deactivate Elements** command from the **Tools** menu.

NOTE

In the **Deactivate Elements** dialog box, point the mouse to each parameter to display a tooltip with a short description.

To deactivate or activate phased array probe elements

1. On the **Tools** menu, click **Deactivate Elements** (see Figure 9-13 on page 111).

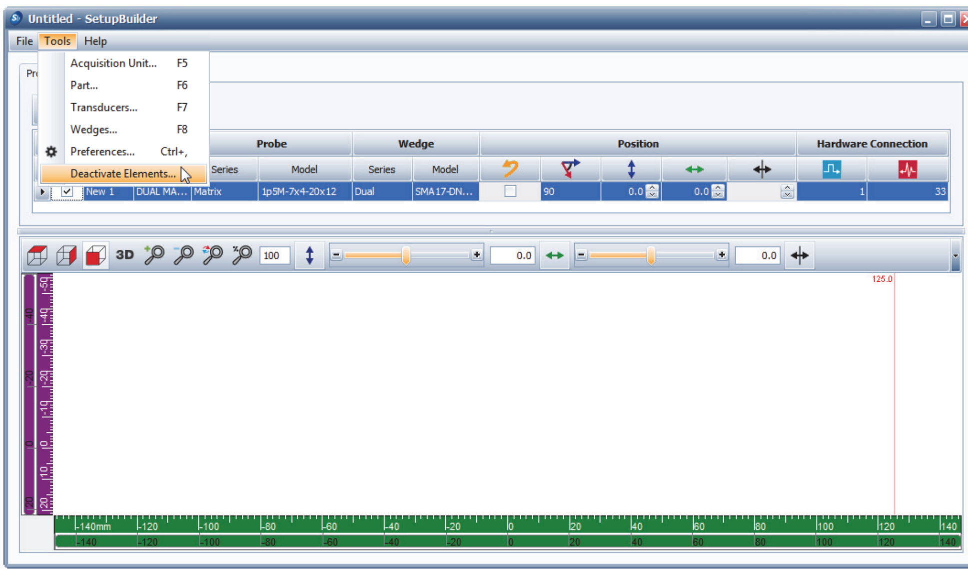


Figure 9-13 Selecting the Deactivate Elements command from the Tools menu

The **Deactivate Elements** dialog box appears (see Figure 9-14 on page 112 and Figure 9-15 on page 112). The dialog box contains the following parameters:

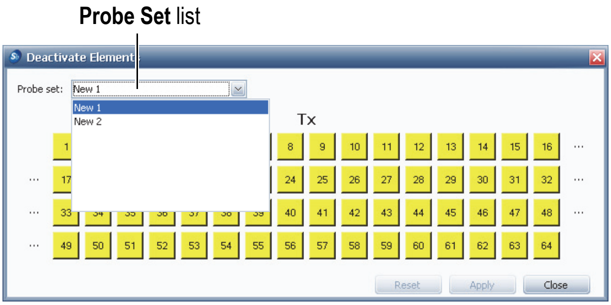


Figure 9-14 The Deactivate Elements dialog box for a PA linear probe

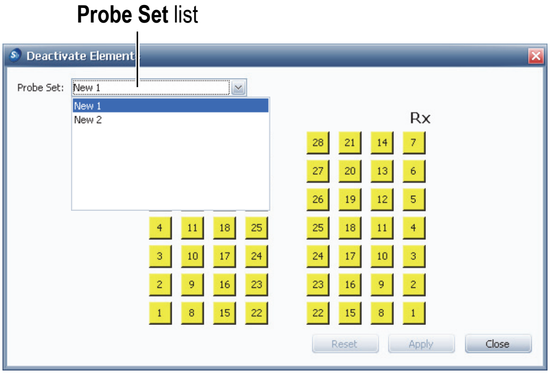


Figure 9-15 The Deactivate Elements dialog box for a dual matrix probe

2. From the **Probe Set** list, select the appropriate probe set.

   All the PA linear- or dual matrix-type probe sets available in your workspace are displayed in the list.

3. To deactivate an element, click it in the grid (see Figure 9-16 on page 113 and Figure 9-17 on page 113).

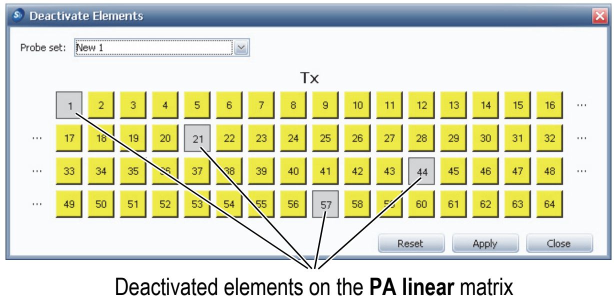


Figure 9-16 The Deactivate Elements dialog box for a PA linear probe

For a PA linear-type probe, all the elements are in sequential order from top to bottom in the rows displayed under **Tx** (see Figure 9-16 on page 113).

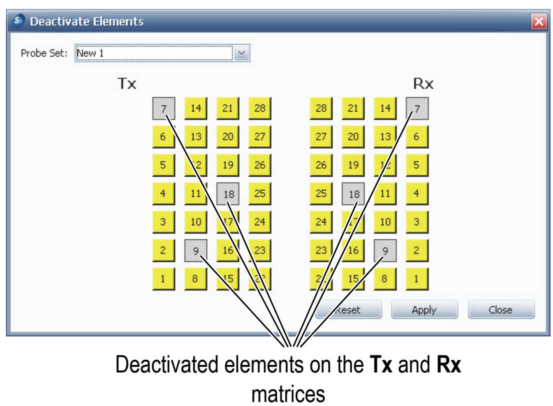


Figure 9-17 The Deactivate Elements dialog box for a dual matrix probe

For a dual matrix probe, the elements from the pulser probe are regrouped under **Tx**, and the elements from the receiver probe are regrouped under **Rx** (see Figure 9-17 on page 113).

4. Perform one of the following actions:
   — Click **Reset** to reset the deactivated elements.
   — Click **Apply** to save the deactivated elements for the selected probe set.

— Click **Close** to save the deactivated elements for the selected probe set and close the dialog box.

NOTE

An inspection setup that contains deactivated elements can only be sent to an OmniScan instrument as one or more .law files. For more information on exporting an inspection setup to a .law file, see "Exporting a Workspace as a .law File" on page 164.

9.5.8 Selecting a Wedge Series and Model

NDT SetupBuilder provides a list of wedge series and models available for the Evident ultrasound and phased array probe sets. Wedge series and models can be selected from the predefined lists. The lists of available wedge series and models display only those items that are compatible with the selected probe series and model.

If the wedge series and models available in the predefined lists do not accurately describe the inspection setup, custom probes, which are displayed separately in the list, can also be defined (see "Managing Custom Probe Models" on page 103).

To select a wedge

◆ On the **Probe Sets** tab, under **Wedge**, select a wedge series and model from the following lists:

Series

In the **Series** list, select the appropriate series of the wedge to be used for the probe set:

- **Angle**: Select **Angle** to display the list of existing angle-beam wedges in the **Model** list. This series is selected by default for UT-type probe sets.
- **Contact**: Select **Contact** to display the list of existing contact wedges in the **Model** list.
- **Integrated**: Select **Integrated** to display the list of existing integrated wedges in the **Model** list.
- **SA2**: Select **SA2** to display the list of existing SA2-series wedges in the **Model** list. This series is selected by default for PA linear-type probe sets. For the PA linear probe-set type, additional wedge series may be available; for example, the following wedge series may be displayed: **ABWX-MWUX**, **CustomSA**, **SA*n***, **SAW*n***, **SI*n***, **SNW*n***, and **SPWZ*n***.

NOTE

For PA linear-type and dual matrix-type probe sets, the **Series** list provides a complete listing of the different series of available wedges categorized by probe mount, as detailed in the Evident probe and wedge catalog.

- **TOFD**: Select **TOFD** to display the list of existing TOFD-series wedges in the **Model** list. This series is selected by default for a probe set of the TOFD type.
- **Wedge**: Select **Wedge** to display the list of existing wedge-series wedges in the **Model** list. This wedge series is available only when **PA Linear** is selected in the **Type** list for the probe set.
- **Custom**: Select **Custom** to display the list of existing custom wedges in the **Model** list. The appropriate custom wedge can then be selected from the **Model** list for use in the inspection setup, or it can be edited.

  The **Custom** wedge series is not displayed unless there is at least one existing custom-wedge model saved in the workspace. For more information, see “Managing Custom Wedge Models” on page 116.

NOTE

Not all wedge series are available for all probe-set types. For more information about the wedge series available for each type of probe set, see Table 7 on page 115.

Table 7 Available wedge series on the Probe Sets tab

Probe-set type Wedge series	PA linear	TOFD	UT	Dual matrix
Angle	—	✓	✓ [a]	—
Contact	✓	✓	✓	—
Integrated	—	✓	✓	—
SA2, ABWX-MWUX, CustomSA, SA*n*, SAW*n*, SI*n*, SNW*n*, and SPWZ*n* [b]	✓ [c]	—	—	✓

Table 7 Available wedge series on the Probe Sets tab *(continued)*

Probe-set type / Wedge series	PA linear	TOFD	UT	Dual matrix
TOFD	—	✓[d]	✓	—
Wedge	✓	—	—	—
Custom[e]	✓	✓	✓	—

a. **Angle** is the default wedge series for the UT probe-set type.
b. Additional wedge series are available for the PA linear probe-set type only. The following series of wedges may be displayed: **SA2**, **ABWX-MWUX**, **CustomSA**, **SA*n***, **SAWn**, **SI*n***, **SNW*n***, and **SPWZ*n***. The list of different series of available wedges is categorized by probe mount, as detailed in the Evident probe and wedge catalog.
c. **SA2** is the default wedge series for the PA linear probe-set type.
d. **TOFD** is the default wedge series for the TOFD probe-set type.
e. The **Custom** wedge-series option becomes available when at least one custom wedge model has been saved for that wedge series.

Model 

In the **Model** list, select the appropriate wedge model to be used for the probe set. The list of displayed wedge models is determined by the probe-set type that has been selected in the **Type** list under **Probe Set** and the wedge series selected in the **Series** list under **Wedge**.

9.5.9 Managing Custom Wedge Models

In your workspace, NDT SetupBuilder allows you to define custom wedges that accurately correspond to your inspection setup or delete existing custom wedges. When **Custom** is selected in the **Series** list on the **Probe Sets** tab under **Wedge**, custom wedge models are displayed in the **Model** list. The custom wedges you create are saved exclusively in the open workspace.

To manage custom wedge models

1. On the **Tools** menu, click **Wedges** (see Figure 9-18 on page 117).

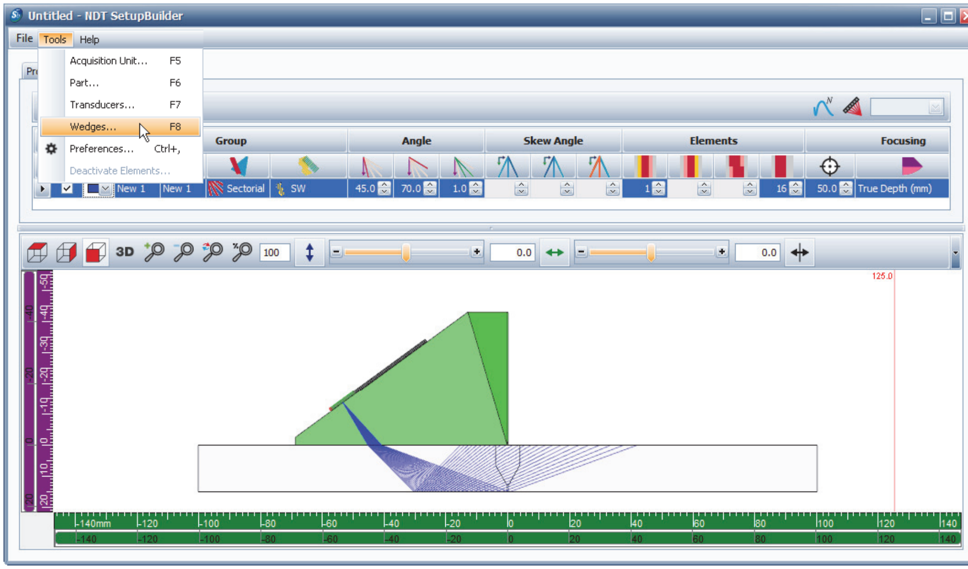


Figure 9-18 Selecting the Wedges command from the Tools menu

The **Manage Wedges** dialog box appears (see Figure 9-19 on page 118). This dialog box contains the six following areas:

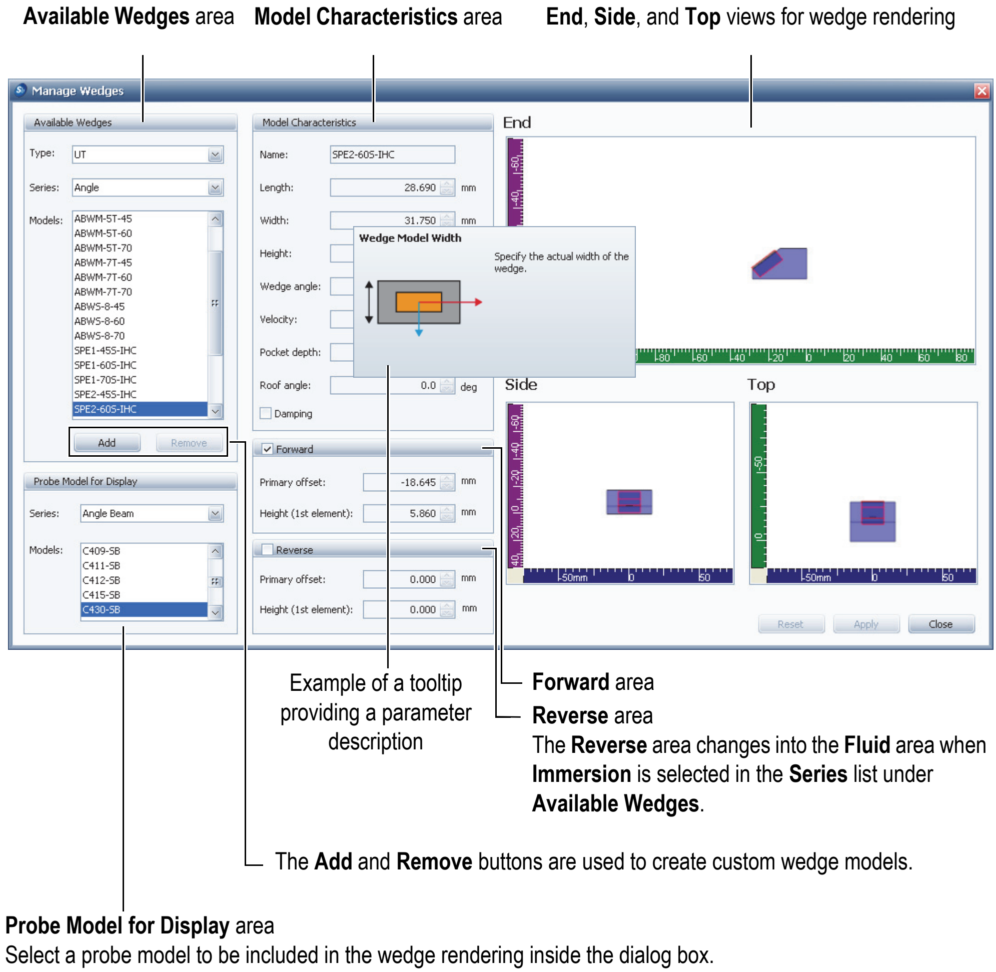


Figure 9-19 The Manage Wedges dialog box

NOTE

In the **Manage Wedges** dialog box, point the mouse to the each parameter to display a tooltip with a short description.

— **Available Wedges**: This area provides lists that allow you to select a wedge model from among predefined probe-set types, wedge series, and wedge models. The **Add** and **Remove** buttons can be used to create a custom wedge based on a predefined model selected using the lists or to delete an existing custom wedge.

— **Probe Model for Display**: This area provides a list that can be used to select a probe model to be included in the graphical representations of the wedge displayed in the dialog box's **End**, **Side**, and **Top** views.

NOTE

The probe model displayed in the main window's RayTracing view is the same probe model selected in the **Model** list under **Probe** on the **Probe Sets** tab. The probe model selected in the **Probe Model for Display** area in the **Manage Wedges** dialog box does not change the selection on the **Probe Sets** tab.

— **Model Characteristics**: This area displays the wedge-model parameters selected in the **Available Wedges** area. For a predefined wedge, these parameters are unavailable. When you create a custom wedge model, the parameters become editable.

— **Forward**: This area displays the wedge-model parameters indicating that the probe can be mounted with a zero-degree (0°) orientation on the wedge. These parameters are unavailable for a predefined wedge. When you create a custom wedge model, the parameters become editable.

— **Reverse**: This area displays the wedge-model parameters indicating that the probe can be mounted with a 180° orientation on the wedge. These parameters are unavailable for a predefined wedge. When you create a custom wedge model, the parameters become editable.

  Refer to the actual wedge-model technical specifications to ensure that the probes can be mounted with a 180° orientation on the wedge.

— **End**, **Side**, and **Top** views: This area provides graphical representations of the selected wedge model.

2. Under **Available Wedges**, select a predefined wedge to use as the basis for your custom wedge:

   Type

   In the **Type** list, select the appropriate probe-set type to be used in the inspection setup:

- **PA Linear**: For phased array inspection. By default, **PA Linear** is selected in the list.
- **TOFD**: For time-of-flight diffraction inspection.
- **UT**: For ultrasound inspection.

The type of probe set selected in the **Type** list determines the options displayed in the **Series** list described below.

Series

In the **Series** list, select the appropriate wedge series to be used for the probe set:

- **Angle**: Select **Angle** to display the list of existing angle-beam wedges in the **Model** list. This series is selected by default for a UT-type probe set.
- **Integrated**: Select **Integrated** to display the list of existing integrated wedges in the **Model** list.
- **SA2**: The **SA2** series is selected by default for a probe set of the PA linear type. For the PA linear probe-set type, additional series of wedges are available. For example, the following wedge series may be available: **ABWX-MWUX**, **CustomSA**, **SA*n***, **SAW*n***, **SI*n***, **SNW*n***, and **SPWZ*n***.
- **TOFD**: The **TOFD** series is selected by default for a probe set of the TOFD type.
- **Wedge**: Select **Wedge** to display the list of existing wedges-series wedges in the **Model** list. This wedge series is only available when **PA Linear** is selected in the **Type** list for the probe set.
- **Custom**: Select **Custom** to display the list of existing custom wedges in the **Model** list. The appropriate custom wedge can then be selected from the **Model** list for use in the inspection setup, or it can be selected and then edited.

NOTE

Not all wedge series are available for all probe-set types. For more information about the wedge series available for each type of probe set, see Table 8 on page 121.

Table 8 Available wedge series in the Manage Wedges dialog box

Probe-set type	PA linear	TOFD	UT
Wedge series[a]			
Angle	—	✓	✓[b]
Integrated	—	✓	✓
SA2, ABWX-MWUX, CustomSA, SA*n*, SAW*n*, SI*n*, SNW*n*, and SPWZ*n*[c]	✓[d]	—	—
TOFD	—	✓	✓
Wedge	✓	—	—
Custom[e]	✓	✓	✓

a. In the **Manage Wedges** dialog box, the **Contact** wedge series is not included in the **Series** list under **Available Wedges**. The **Contact** wedge series is only available in the **Series** list under **Wedges** on the **Probe Sets** tab (see “Selecting a Wedge Series and Model” on page 114).
b. **Angle** is the default wedge series for the UT probe-set type.
c. Additional wedge series are available for the PA linear probe-set type only. The following series of wedges may be displayed: **SA2**, **ABWX-MWUX**, **CustomSA**, **SA*n***, **SAWn**, **SI*n***, **SNW*n***, and **SPWZ*n***. The list of different series of available wedges is categorized by probe mount, as detailed in the Evident probe and wedge catalog.
d. **SA2** is the default wedge series for the PA linear probe-set type.
e. The **Custom** wedge-series option becomes available when at least one custom wedge model has been saved for that series.

Models 

In the **Models** list, select the appropriate wedge model to be used for the probe set. The wedge-series list that is displayed is determined by the probe-set type selected in the **Type** list under **Available Wedges** and by the wedge series selected in the **Series** list.

3. Under **Probe Model for Display**, select a probe series and model to be included in the graphical representations of the wedge that are displayed in the dialog box’s **End**, **Side**, and **Top** views:

Series

In the **Series** list, select the appropriate probe series to be displayed in the dialog box's three views:

- **Angle Beam**: Select **Angle Beam** to display the list of angle-beam probes in the **Model** list.
- **Contact**: Select **Contact** to display the list of contact probes in the **Model** list.
- **Integrated Wedge**: Select **Integrated Wedge** to display the list of integrated-wedge probes in the **Model** list.
- **Straight Beam**: Select **Straight Beam** to display the list of straight-beam probes in the **Model** list.
- **Custom**: Select **Custom** to display the list of existing custom probes in the **Model** list. The appropriate custom probe can then be selected from the **Model** list for use in the inspection setup, or it can be selected and edited.

  The **Custom** probe series is not displayed unless there is at least one existing custom probe model saved in the workspace. For more information, see "Configuring Custom Probe Model Parameters" on page 108.

The probe-series list displays only those series that are compatible with the probe-set type selected in the **Type** list under **Available Wedges**.

Model 

In the **Model** list, select the appropriate probe model to be displayed in the dialog box's three views.

The probe-models list displays only those models that are compatible with the probe series selected in the **Series** list under **Probe Model for Display**.

4. Under **Model Characteristics**, configure the parameters according to the wedge specifications for the inspection setup.

   For details about how to configure wedge parameters when defining a new wedge model, see "Configuring Custom Wedge Model Parameters" on page 124.
5. Perform one of the following actions:

   — To delete a custom wedge, select it in the **Models** list under **Available Wedges**, and then click **Remove** (see Figure 9-12 on page 107).

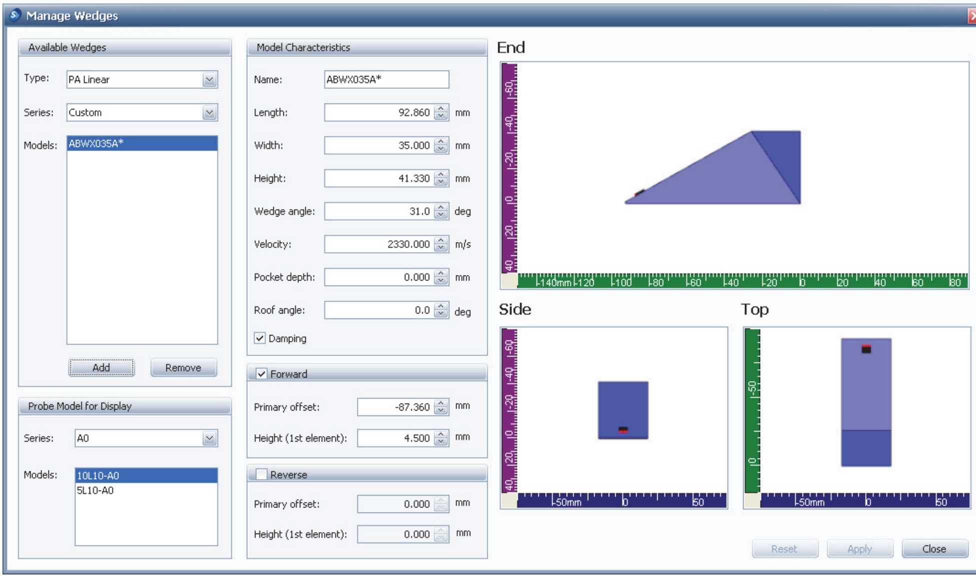


Figure 9-20 The Manage Probes dialog box with a new probe

The selected custom wedge is deleted from the **Models** list.

— To modify a custom wedge, select it in the **Models** list under **Available Wedges**.

The parameters for the selected custom wedge will now be available under **Model Characteristics**.

— To add a custom wedge, click **Add**.

When adding a custom wedge under **Model Characteristics**, a wedge with the name and parameters of the model selected from the **Models** list is added (see Figure 9-12 on page 107). The name of the new wedge, marked by an asterisk, is displayed in the **Name** box, and it can be edited. The parameters under **Model Characteristics** are also now available.

6. Configure the wedge parameters according to the specifications of the inspection setup.

   For details on how to configure wedge parameters when creating or modifying a custom wedge model, see "Configuring Custom Wedge Model Parameters" on page 124.

7. Perform one of the following actions:

- Click **Reset** to reset the selected custom wedge model parameters in the **Models** list under **Available Wedges**.
- Click **Apply** to save the custom wedge model in the **Models** list under **Available Wedges**.
- Click **Close** to save the custom wedge model in the **Models** list under **Available Wedges** and close the dialog box.

9.5.10 Configuring Custom Wedge Model Parameters

The **Manage Wedges** dialog box can be used to create, modify, or delete custom wedges. For more information about managing wedges and accessing the dialog box, see “Managing Custom Wedge Models” on page 116.

The parameters from the **Manage Wedges** dialog box are unavailable for predefined wedges. When you create a custom wedge model, the parameters become editable.

NOTE

When you select the predefined immersion wedge series, some of the wedge parameters become editable in the **Model Characteristics**, **Forward**, and **Reverse** areas.

The **Manage Wedges** dialog box contains the following four sections, which can be used to configure the custom wedge parameters (see Figure 9-21 on page 125):

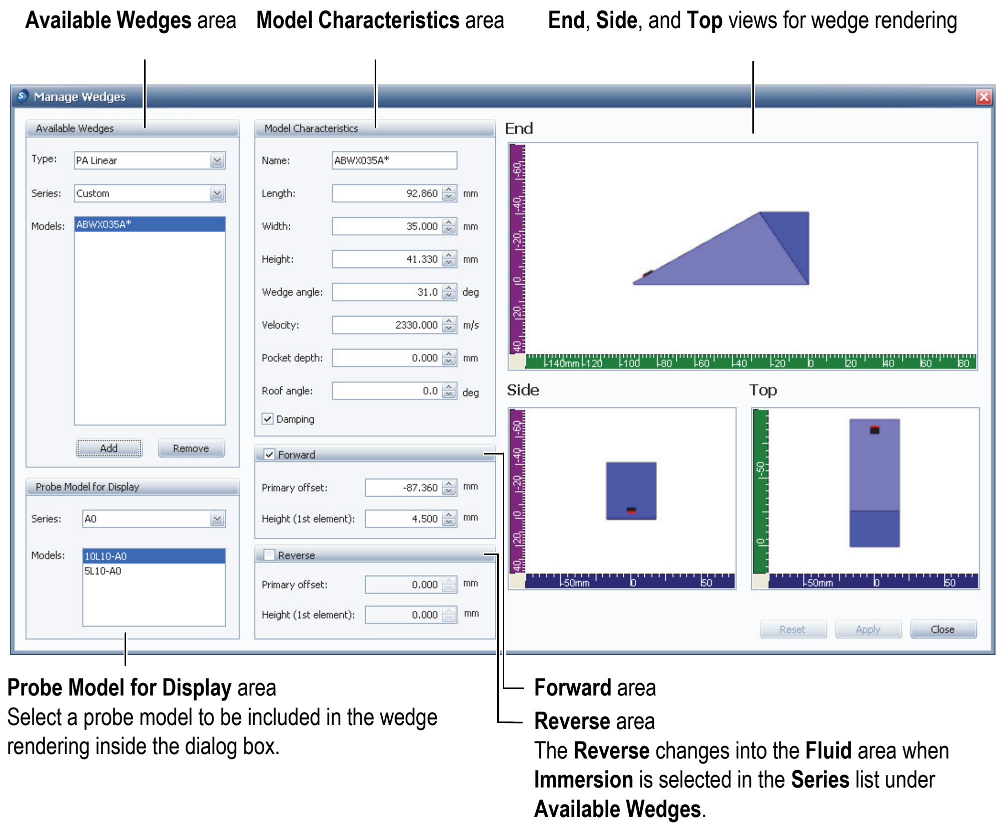


Figure 9-21 The Manage Wedges dialog box for custom wedge configuration

- **Model Characteristics**: This area displays the parameters of the wedge model selected in the **Available Wedges** area.
- **Forward**: This area displays the wedge-model parameters indicating that the probe can be mounted on the wedge with a zero-degree (0°) orientation.
- **Reverse**: This area displays the wedge-model parameters indicating that the probe can be mounted on the wedge with a 180° orientation. The **Reverse** area changes into the **Fluid** area when **Immersion** is selected in the **Series** list under **Available Wedges**.

  Refer to actual wedge-model technical specifications to ensure that probes can be mounted on the wedge with a 180° orientation.

- **End**, **Side**, and **Top** views: This area provides graphical representations of the wedge model being configured. These representations are dynamically updated when the wedge parameters are configured.

Several of the custom wedge model parameters are configured according to wedge-specific conventions.

The following parameters can be configured in the **Manage Wedges** dialog box:

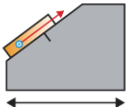
Model Characteristics area

Name

Enter a name for the custom wedge model. By default, for a new custom wedge, the name of the currently selected wedge model marked by an asterisk ([Wedge Model Name]*) is automatically generated. The name of a wedge model can contain a maximum of 15 characters.

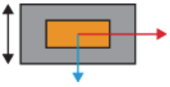
NOTE

It is recommended to include the name of the probe model for which the wedge model has been configured in the custom wedge's name.

Length 

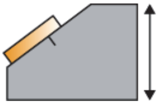
Enter a value indicating the actual length of the wedge.

For a cylindrical part with a curvature along the primary axis, the wedge length represents the distance between the contact points of the wedge.

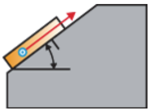
Width 

Enter a value indicating the actual width of the wedge.

For a cylindrical part with a curvature along the secondary axis, the wedge width represents the distance between the contact points of the wedge.

Height 

Enter a value indicating the actual height of the wedge model.

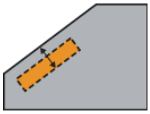
Wedge angle[1] 

Enter a value in degrees indicating the angle of the wedge.

The *wedge angle* is the angle between the element surface (when it is fixed onto the wedge) and the surface of the part (or the tangential plane in relation to the surface of the part in the case of a cylindrical geometry). This angle is obtained upon rotation around the secondary axis of the probe, and it can have values between 0° and 89.9°.

Velocity[1]

Enter a value in meters per second or inches per microsecond (m/s or in./µs) indicating the ultrasound longitudinal-wave velocity in the wedge.

Pocket depth 

Enter a value indicating the pocket depth of the wedge perpendicular to the probe surface.

Roof angle[1] 

Enter a value in degrees indicating the roof angle of the wedge. The *roof angle* is the rotation angle around the primary axis of the probe set, and can have values between −89.9° and 89.9°. For a probe skew of 0°, a positive roof angle will generate beams with total skew angles between 0° and 180°.

NOTE

An inspection setup that contains a wedge with a roof angle value other than 0 (zero) can only be exported as a .law file. For more information, see “Exporting a Workspace as a .law File” on page 164.

---

1. For a predefined wedge model, when a wedge series of the **Immersion** type is selected, the **Wedge angle**, **Velocity**, **Roof angle**, **Primary offset**, **Height (1st element)**, and **Fluid** parameters become editable.

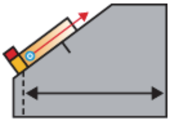
Damping

Select the **Damping** check box to show the damping material on the wedge. Clear the **Damping** check box to hide the damping material on the wedge.

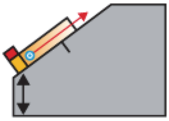
Forward area

Select the **Forward** check box to specify that the probe can be mounted with its primary axis pointing in the same direction as the probe set's primary axis, which indicates that the probe has a zero-degree (0°) orientation on the wedge.

The following parameters can be configured for the probe location on the wedge:

Primary offset[1] 

Enter a value indicating the primary axis offset at the middle of the first element of the probe-set probe relative to the back of the wedge, with a 0° orientation on the wedge. The offset is always measured along a straight line and normally has positive values.

Height (1st element)[1] 

Enter a value indicating the height (tertiary offset) at the middle of the first element of the probe-set probe relative to the material surface, with a 0° orientation on the wedge (see Figure 9-22 on page 128).

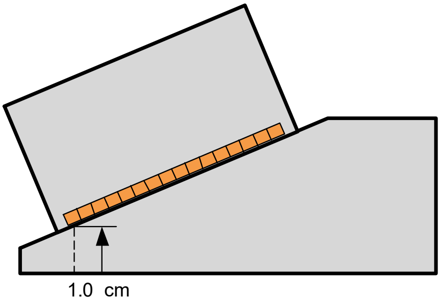


Figure 9-22 The height of the first element

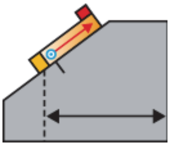
1. For a predefined wedge model, when a wedge series of the **Immersion** type is selected, the **Wedge angle**, **Velocity**, **Roof angle**, **Primary offset**, **Height (1st element)**, and **Fluid** parameters become editable.

For a cylindrical part, the height is measured relative to the flat wedge surface obtained by drawing a line between the contact points of the wedge, and its value is always positive.

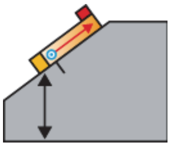
This parameter is usually provided by the wedge manufacturer.

Reverse area

Select the **Reverse** check box to specify that the probe can be mounted with its primary axis pointing in the opposite direction of the probe set's primary axis, which indicates that the probe has a 180° orientation on the wedge:

Primary offset 

Enter a value indicating the primary axis offset of the middle of the first reverse element of the probe-set probe relative to the back of the wedge, with a 180° orientation on the wedge. The offset is always measured along a straight line, and normally has positive values.

Height (1st element) 

Enter a value indicating the height (tertiary offset) of the middle of the first reverse element of the probe-set probe relative to the material surface, with a 180° orientation on the wedge (see Figure 9-22 on page 128).

For a cylindrical part, the height is measured relative to the flat wedge surface obtained by drawing a line between the contact points of the wedge, and this value is always positive.

This parameter is usually provided by the wedge manufacturer.

NOTE

The **Reverse** column's check box is only available for certain series of wedges. Refer to your probes' and/or wedges' technical specifications to verify whether the probes can be reversed when mounted on the wedges.

9.5.11 Configuring Probe-Set Position Parameters

Several probe-set position parameters for custom wedge models are defined according to specific probe conventions.

To configure probe-set position parameters

- On the **Probe Sets** tab, under **Position**, configure the following parameters to position the probe set relative to the weld to be inspected:

  Reverse column ()

  Under **Reverse**, select the check box to mount the probe backwards on the wedge (see Figure 9-23 on page 130), to reverse the position of the probe on the wedge, and to invert the order of the element numbers in the calculation of the beams (focal laws). When the **Reverse** column check box is selected, the lowest element is attributed the highest element number.

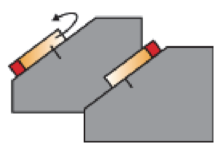


Figure 9-23 Example of a probe mounted backwards on a wedge

NOTE

The **Reverse** column check box is only available for some series of wedges. Refer to your probes' and/or wedges' technical specifications to verify whether the probes can be reversed when mounted on the wedges.

  Probe-Set Skew Angle column ()

  In the list under **Probe-Set Skew Angle**, select the angle of the ultrasonic beam relative to the scan axis. The available options are: **0°**, **90°**, **180°**, and **270°** (see Figure 9-24 on page 131).

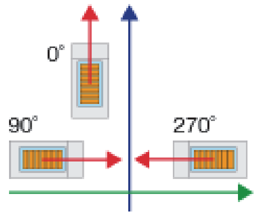


Figure 9-24 Examples of available probe-set angles

The *probe-set skew angle* is defined as the angle between the primary axis of the probe set and the plane defined by the scan axis and the ultrasound 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 in the clockwise direction.

Probe-Set Scan Offset column ()

In the box under **Probe-Set Scan Offset**, enter the scan offset of the probe set. The *probe-set scan offset* represents the distance between the probe set's primary axis and the origin of the scan axis (see Figure 9-25 on page 131), or the distance on the scan axis between the actual beam emergence point of the probe set and the reference point on the probe.

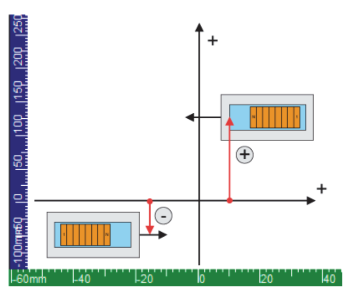


Figure 9-25 Probe-set scan offset

Probe-Set Index Offset column ()

In the box under **Probe-Set Index Offset**, enter the mechanical index offset of the probe set. The *probe-set index offset* represents the distance between the front of the wedge and the origin of the index axis (see Figure 9-26 on page 132), or the distance on the index axis between the actual beam emergence point of the probe and the reference point of the probe.

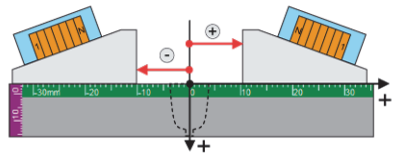


Figure 9-26 Probe-set index offset

Probe Center Separation (PCS) column ()

In the box under **Probe Center Separation (PCS)**, enter the PCS distance of the selected TOFD probe set (see Figure 9-27 on page 132). The PCS is defined as the distance between the beam exit points of the two probes of a TOFD probe set.

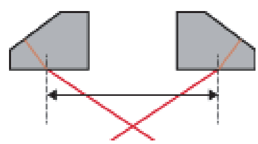


Figure 9-27 Probe center separation measurement

TIP

The values of the **Probe-Set Scan Offset**, **Probe-Set Index Offset**, and **Probe Center Separation (PCS)** parameters can be interactively modified using controls on the toolbar from RayTracing view. For more information, see “Interactively Modifying Probe Parameter Values” on page 47.

9.5.12 Setting Up the Probe-Set Hardware Connection

The **Hardware Connection** section on the **Probe Sets** tab provides parameters that can be used to identify the first wired element of the pulser group and the first wired element of the receiver group on the probe-set probes.

For more information about probe-set pulser and receiver elements, see "Physical Principles" on page 173.

To set up the probe-set hardware connection

◆ On the **Probe Sets** tab, under **Hardware Connection**, configure the following parameters in order to identify the first element of the pulser and/or receiver group on the probe-set probes:

Pulser column ()

In the list under **Pulser**, select the channel number of the acquisition unit to be connected to the pulsing probe's first element (see Figure 9-28 on page 133). By default, for a PA linear or dual matrix probe set, element **1** is selected in the list.

For a UT or TOFD probe set, channel **P1** is selected in the list. The values that contain a P (P1, P2, etc.) identify pulser channels instead of probe elements.

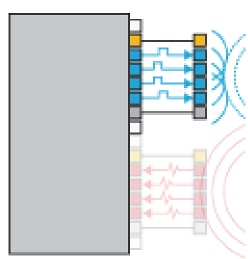


Figure 9-28 Pulsing probe with first element marked in yellow

When the pulsing probe and the receiving probe are the same, the receiver channel is identical to the pulser channel.

Receiver column()

In the list under **Receiver**, select the channel number of the acquisition unit to be connected to the receiving probe's first element (see Figure 9-29 on page 134).

For a TOFD probe set, channel **R1** is selected in the list. The values that contain a P (P1, P2, etc.) identify pulser channels instead of probe elements. The values that contain an R (R1, R2, etc.) identify receiver channels instead of probe elements. When a channel has been selected in the **Pulser** column list, it is not available in the **Receiver** column list.

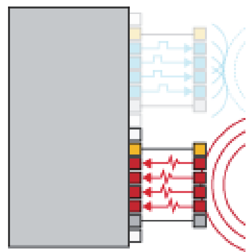


Figure 9-29 Receiving probe with the first element marked in yellow

When the pulsing probe and receiving probe are the same, the receiver channel is identical to the pulser channel.

The **Receiver** column list is unavailable when the probe type selected is PA Linear or UT.

NOTE

When using only one probe, you should set the value of these parameters to 1 (see Figure 9-30 on page 134).

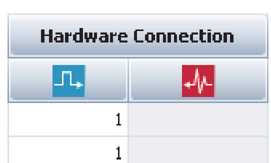


Figure 9-30 Example pulser and receiver configuration for a single probe

Use the **Pulser** and **Receiver** column parameters when:

- Two UT probes will be connected in a symmetric configuration for measurement.

- Two phased array probes will be connected to a splitter box (such as the OMNI-A-ADP05 shown in Figure 9-31 on page 135) for measurement in a symmetric configuration.

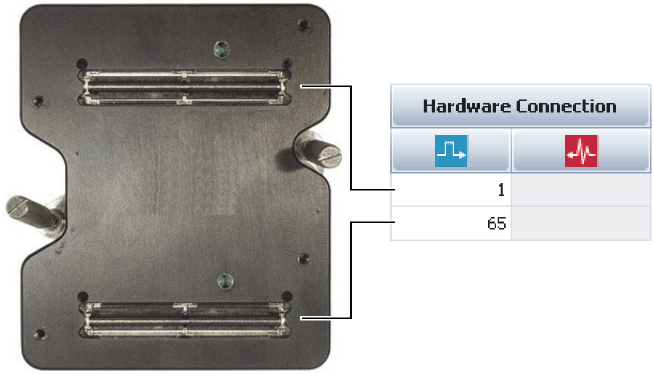


Figure 9-31 Example pulser and receiver configuration for two 128-element probes

In this case, you need to separately calculate the beams for each probe using the same values for all the NDT SetupBuilder parameters, with the exception of the **Pulser connection** and **Receiver connection** parameters. The **Pulser connection** and **Receiver connection** parameters for a given probe must have the same value. If you are working with 128-element probes, the value can be set between 1 and 64 for the first probe and between 65 and 128 for the second probe (see Figure 9-31 on page 135).

NOTE

In the case of TOFD probes, you need to connect the pulser and receiver on different connectors and note the element numbers used in the respective parameters.

---

10. Configuring One or More Groups

In NDT SetupBuilder, the groups used in an inspection setup are configured on the **Groups** tab. The **Groups** tab provides tools and parameters for modifying groups that were automatically created with probe sets, creating and configuring new groups, deleting groups, and setting group visibility.

Before adding groups to your inspection setup on the **Groups** tab, you must create the required probe sets using the **Probe Sets** tab. When creating a group, you are required to select a probe set from the list of currently available probe sets in your workspace. For more information, see “Adding a Probe Set” on page 92.

The group’s width is measured at the 6 dB attenuation level. As such, the beam spread at −6 dB is displayed in RayTracing view. The beam overlap displayed in RayTracing view for a TOFD probe set is also −6 dB.

The **Groups** tab contains a toolbar that provides basic beam-definition commands (see Figure 10-1 on page 138).

TIP

Several of the **Groups** tab’s commands can be activated using keyboard shortcuts. For more information, see “Accessing Commands Using Keyboard Shortcuts” on page 31.

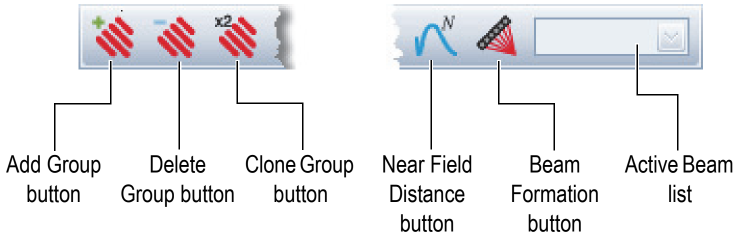


Figure 10-1 Groups tab toolbar

The **Groups** tab columns provide parameters that can be used to configure the group visibility, angles, elements, and focusing.

NOTE

On the **Groups** tab, point the mouse to each column header or toolbar element to display a tooltip with a short description of the pertaining parameter or function.

The groups being configured for an inspection setup can be viewed in 2-D or 3-D in RayTracing view, except for groups associated with probe sets of the dual matrix type, which are only displayed in End (D) view. For more information, see “RayTracing View” on page 40.

When you select a group on the **Groups** tab, the probe set it is associated with is highlighted in RayTracing view. For more information, see “Identifying Selected Probe Sets in RayTracing View” on page 32.

The tools needed to configure the probe sets associated with the groups can be found on the **Probe Sets** tab. For more information about configuring groups, see “Configuring One or More Probe Sets” on page 91.

10.1 Adding a Group

For probe sets of the PA linear or dual matrix type, the **Groups** tab toolbar can be used to add one or more groups in an inspection setup. Groups cannot be added to probe sets of the TOFD or UT types, each of which are created with an associated group.

Before being able to add groups to your inspection setup on the **Groups** tab, you must have created the required PA linear- or dual matrix-type probe sets on the **Probe Sets** tab. When creating a group, you are required to select a probe set from the list of currently available probe sets in your workspace. For more information, see “Adding a Probe Set” on page 92.

To add a group

1. Click the **Groups** tab.

2. Click the **Add Group** button ( ) on the toolbar, and then select the desired PA linear- or dual matrix-type probe set in the list that appears immediately below the **Add Group** button (see Figure 10-2 on page 139).

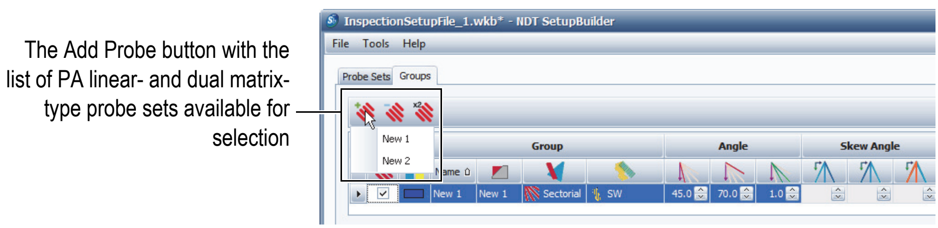


Figure 10-2 The Add Probe button with the Probe Sets list displayed

A new group is added to the **Groups** tab table. This table can be used to configure or modify the available group definition parameters. However, the probe set associated with the group cannot be changed after it has been created. For more information, see “Configuring the Parameters of Groups” on page 145.

10.2 Deleting a Group

For probe sets of the PA linear or dual matrix type, the **Groups** tab toolbar can be used to delete one or more groups in an inspection setup. The groups associated with probe sets of the TOFD or UT type cannot be deleted.

To delete a group

1. In the list under the **Groups** tab, select one or more groups you want to delete (see Figure 10-3 on page 140).

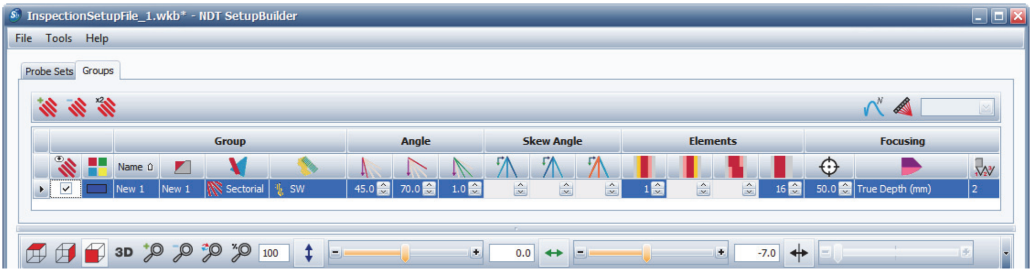


Figure 10-3 Selecting a group on the Groups tab for deletion

2. Click the **Remove Group** button () on the **Groups** tab toolbar.

   The selected groups are deleted from the list of groups on the **Groups** tab.

10.3 Cloning a Group

For probe sets of the PA linear or dual matrix type, the **Groups** tab toolbar can be used to clone one or more groups in an inspection setup. The groups associated with probe sets of the TOFD or UT types cannot be cloned.

To clone a group

1. In the list under the **Groups** tab, select one or more groups you want to clone (see Figure 10-4 on page 140).

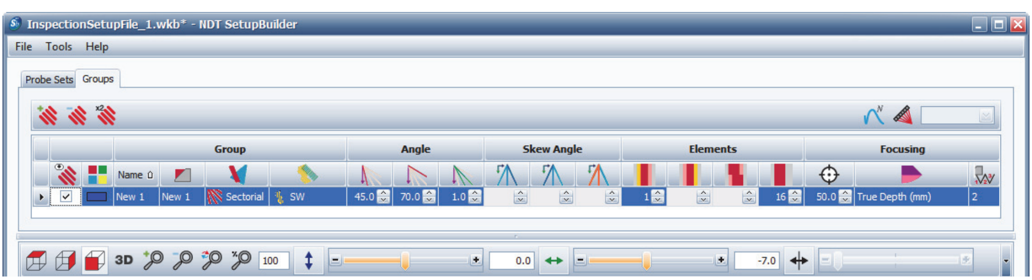
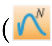
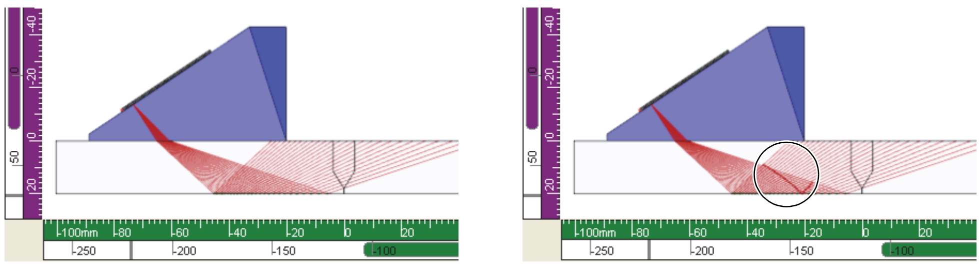


Figure 10-4 Selecting a group on the Groups tab for cloning

2. Click the **Clone Group** button () on the **Groups** tab toolbar.
   The cloned groups are added to the list of groups on the **Groups** tab.

10.4 Setting a Group’s Near-Field Visibility

The **Groups** tab toolbar can be used to set the visibility of a group’s near field. The **Near Field Distance** button () is used to toggle a function that displays the near fields of all the groups visible in RayTracing view (see Figure 10-5 on page 141).



Group in RayTracing view with near-field visibility off (default)

Group in RayTracing view with near-field visibility on

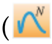
Figure 10-5 Near-field visibility examples

NOTE

Only within an inspection setup configured for a plate part, the near field for groups associated to probe-sets of the PA linear, TOFD or UT types can be displayed. When an inspection setup is configured for a tube or bar part, the near field for groups cannot be displayed.

The near-field visibility is set in RayTracing view simultaneously for all the visible groups of compatible inspection setup configurations.

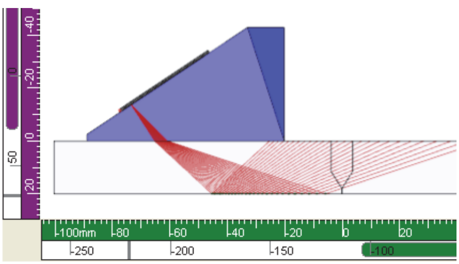
To set a group's near-field visibility

◆ Click the **Near Field Distance** button () on the **Groups** tab toolbar.

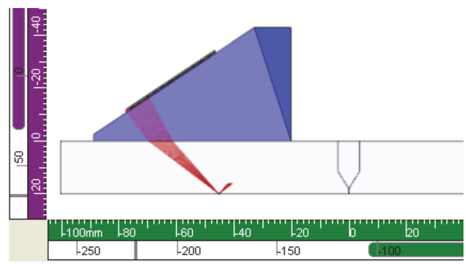
Dots are displayed on the beams of all the visible groups in RayTracing view, with the end of the near field displayed for each beam. The **Near Field Distance** button toggles the beams' near-field visibility on and off.

10.5 Setting Beam (Focal Law) Formation Visibility for a Group

The **Groups** tab toolbar can be used to set the visibility of beam formation (the focal laws) for a group. The **Beam Formation** button () is used to toggle a function that displays either all of the beams from a group or one of the beams that form that group in RayTracing view (see Figure 10-6 on page 142).



RayTracing view with beam (focal law) formation visibility off (default).



RayTracing view with beam (focal law) formation visibility on.

Figure 10-6 Group visibility examples

NOTE

Only within an inspection setup configured for a plate part, the beam formation visibility for groups associated to probe-sets of the PA linear or dual matrix types can be configured. When an inspection setup is configured for a tube or bar part, the beam formation for groups cannot be configured.

The beam formation is set in RayTracing view for only one group at a time for groups of compatible inspection setup configurations.

To set the beam-formation visibility of a group

1. In the list under the **Groups** tab, select the group for which you want to set the beam-formation visibility (see Figure 10-7 on page 143).

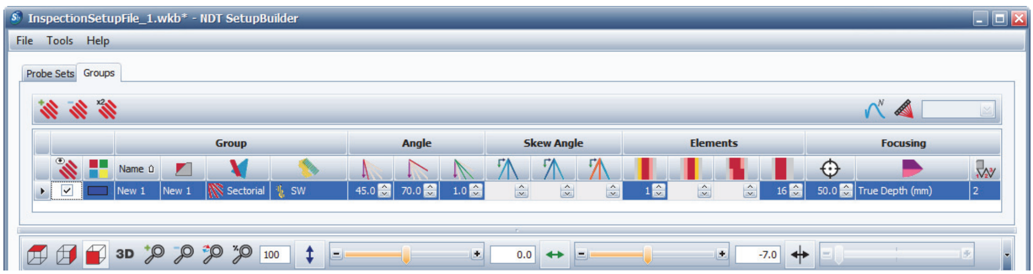


Figure 10-7 Setting the beam-formation visibility on the Groups tab

2. Click the **Beam Formation** button () on the **Groups** tab toolbar. The **Beam Formation** button toggles the beam-formation visibility on and off.

   When more than one beam is generated for the group, the **Active Beam** list to the right of the **Beam Formation** button becomes available (see Figure 10-8 on page 143).

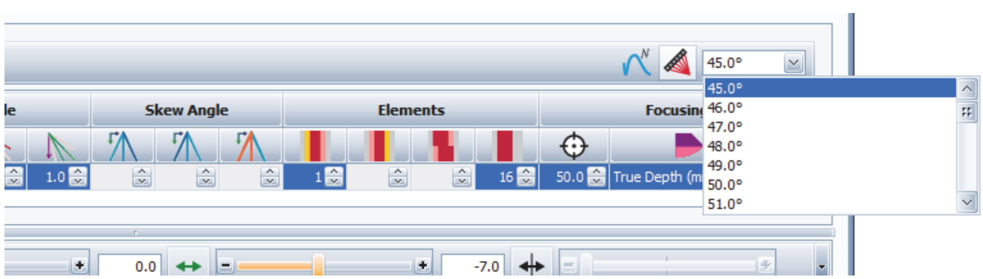


Figure 10-8 Selecting the displayed beam in the Active Beam list

3. From the **Active Beam** list, select the angle of the beam to be displayed in RayTracing view.

10.6 Setting Group Visibility

On the **Groups** tab, a group's visibility can be set independently from the probe-set visibility by using the check box in the **Visible** column () on the left-hand side of the parameter table (see Figure 10-9 on page 144). By default, a group is visible upon creation.

NOTE

When you hide a probe set, all of the groups associated with it are automatically hidden. Visibility can only be set for the groups of a visible probe set. For more information about setting a probe set's visibility, see "Setting Probe-Set Visibility" on page 97.

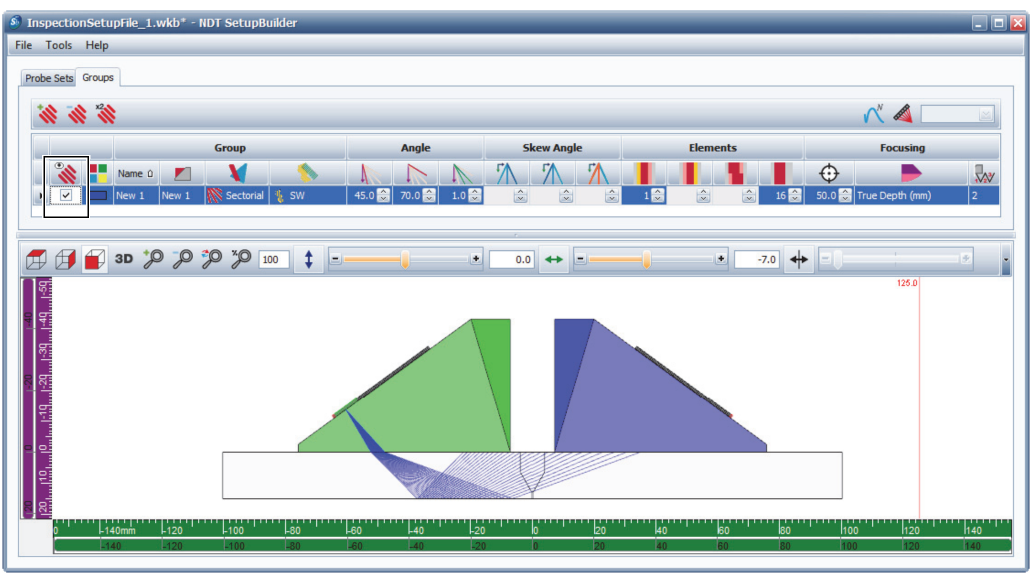


Figure 10-9 The Groups tab with a group's visibility check box selected (in the far-left column)

To set group visibility

1. In the list under the **Groups** tab, select the group for which you want to set the visibility (see Figure 10-9 on page 144).
2. Perform one of the following actions:
   — To show the group in RayTracing view, select the check box under **Visible** ().
   The group is displayed in RayTracing view.
   — To hide the group in RayTracing view, clear the check box under **Visible** ().
   The group is hidden in RayTracing view (Figure 10-10 on page 145).

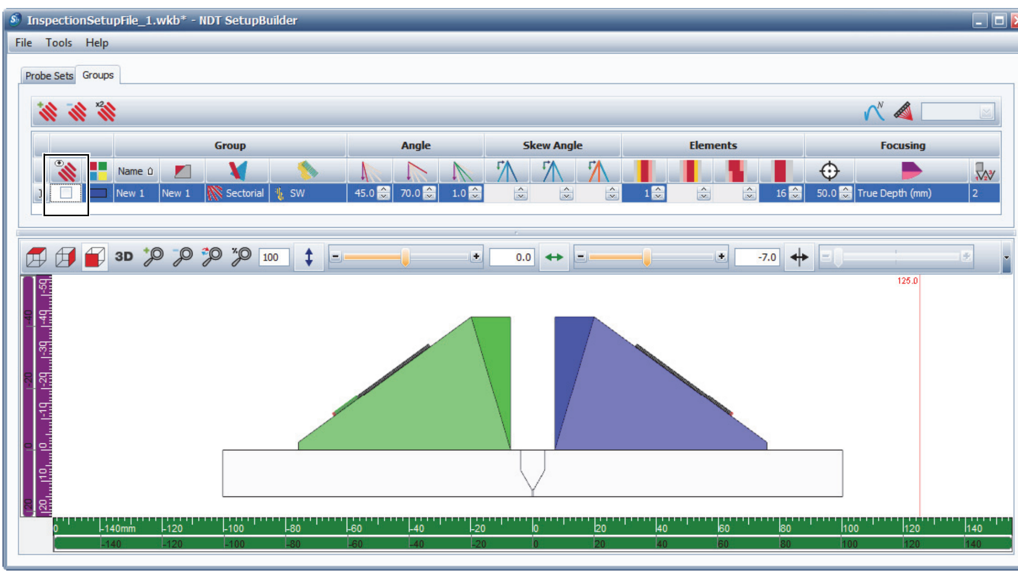


Figure 10-10 The Groups tab with a group's visibility check box cleared

10.7 Configuring the Parameters of Groups

When defining or modifying a group on the **Groups** tab, the available parameters identified on the tab's column headers can also be configured (see Figure 10-11 on page 146).

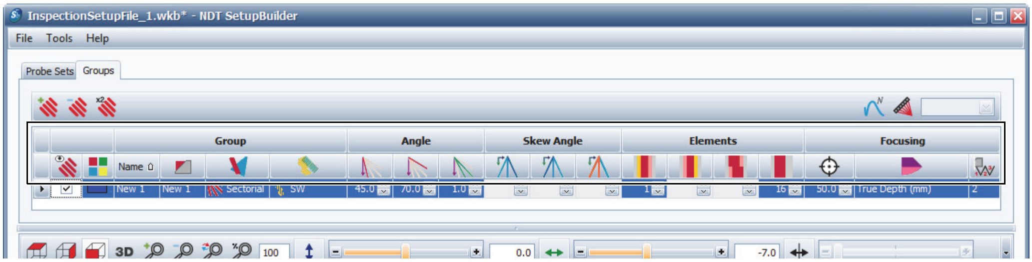


Figure 10-11 The Groups tab column headers

To configure the parameters of groups

1. From the **Groups** tab list, select the group you want to configure (see Figure 10-12 on page 146).

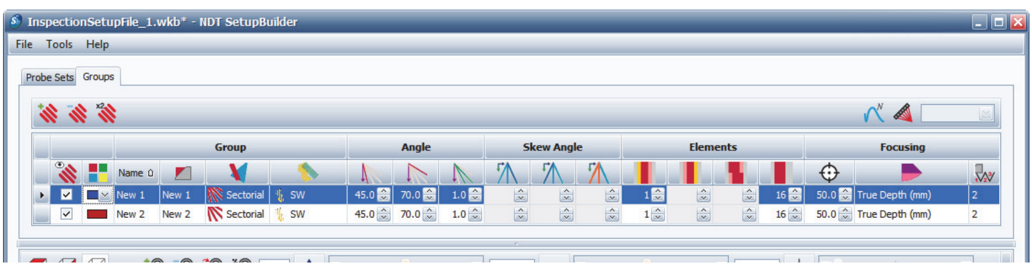


Figure 10-12 Selecting a group on the Groups tab for configuring parameters

2. On the left-hand side of the parameters table, set the group visibility and display color parameters. For more information, see “Setting the Group Visibility and Display Parameters” on page 147.
3. In the **Group** section, configure the group parameters. For more information, see “Configuring the Group Section Parameters” on page 148.
4. In the **Refracted Angle** section, configure the group refracted angle parameters. For more information, see “Configuring Refracted Angle Parameters” on page 149.
5. In the **Skew Angle** section, configure the group skew angle parameters. For more information, see “Configuring Skew Angle Parameters” on page 152.

6. In the **Elements** section, configure the group element step and element quantity (aperture) parameters. For more information, see “Configuring Element Parameters” on page 154.
7. In the **Focusing** section, configure the group focusing parameters. For more information, see “Configuring Focusing Parameters” on page 157.

10.7.1 Setting the Group Visibility and Display Parameters

The group visibility and display parameters can be set on the left-hand side of the **Groups** tab for the selected group.

To set the group visibility and display parameters

◆ On the **Groups** tab, on the left-hand side of the parameter table, set the group visibility and display color:

Visible column ( )

In the **Visible** column ( ) box, select or clear the check box to show or hide the group in RayTracing view. The visibility of the group is set independently from the probe-set visibility. For more information about setting probe-set visibility, see “Setting Probe-Set Visibility” on page 97.

NOTE

Visibility cannot be set for the groups of an invisible probe set.

Color column ( )

In the **Color** column list, click the color indicator to display the color picker, and then select the group’s color in RayTracing view (see Figure 10-13 on page 148). The default color for a group is blue.

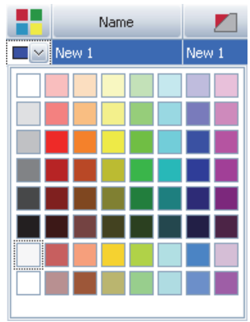


Figure 10-13 Color column's color picker

10.7.2 Configuring the Group Section Parameters

The **Groups** tab can be used to configure the group parameters in the **Group** section.

To configure group parameters

◆ In the **Group** section, configure the following parameters:

Name column

In the **Name** column box, type the name of the group. By default, the name **New *n*** is displayed.

Probe column ( )

In the **Probe** column box, the name of the probe set associated with the group is displayed.

The probe set associated with a group cannot be changed. A probe set of the PA linear or dual matrix type is associated with a group at the time of the group's creation. A group is automatically created and associated with a probe set of the TOFD or UT type.

Group Type column ( )

In the **Group Type** column list, select the appropriate type for the group. The following options are available:

- **Linear**
- **Sectorial**
- **Compound**

- ○ **Single**

Depending on the probe-set type selected in the **Probe** column list, the options available in the list change as follows:

- ○ PA: **Linear**, **Sectorial**, **Compound**, and **Single**
- ○ TOFD: **Single**
- ○ UT: **Single**
- ○ Dual Matrix: **Sectorial**

Wave Type column ( )

In the **Wave Type** column list, select the wave type for the group: **LW** or **SW** (see Figure 10-14 on page 149).

Figure 10-14 LW wave-type and SW wave-type representations

When a probe set of the TOFD type is selected for a group, **LW** is selected by default in the list, and **SW** is also available for selection.

For probe sets of the PA linear or UT type, **SW** is selected by default, and **LW** is also available for selection.

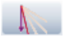
When a probe set of the dual matrix type is selected for a group, **LW** is selected by default in the list, and it is the only option available.

10.7.3 Configuring Refracted Angle Parameters

The **Groups** tab can be used to configure the group refracted angle parameters in the **Refracted Angle** section.

To configure group refracted angle parameters

◆ In the **Group** section, configure the following parameters:

Refracted Angle Start column ()

In the **Refracted Angle Start** column box, configure the angle of the first beam of the group (see Figure 10-15 on page 150). By default, the value is **45°** for probe sets of the PA linear or dual matrix type.

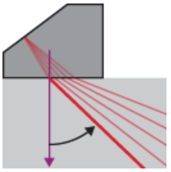


Figure 10-15 Refracted angle start measurement

For probe sets of the PA linear or dual matrix type, the value can also be edited. When the start refracted angle value is confirmed, the value in the **Refracted Angle Stop** column box is automatically updated if required, as follows:

- If the start angle value is greater than the stop angle value, the stop angle value is updated to equal the start angle value.
- If steps have been configured, they are added to the start angle value, so the updated stop angle value is closest to, but no greater than, the currently set stop angle value. For example, for a start angle value set to 20°, a stop angle value of 21°, and steps of 0.3°, the angles will be 20°, 20.3°, 20.6°, and 20.9°.

When the stop angle value is automatically updated, it is always calculated from the last one computed. If you modify the start angle value several times, the values computed between the start angle and stop angle values will drift toward the start angle value.

NOTE

For probe sets of the UT or TOFD type, the start refracted angle value is computed internally according to the probe set's wedge angle and the inspected part's material velocity by means of Snell's law. However, the start refracted angle value is not displayed, and the **Refracted Angle Start** box is not available.

Refracted Angle Stop column ()

In the **Refracted Angle Stop** column box, configure the angle of the last beam of the group (see Figure 10-16 on page 151). By default, the value is **70°** for probe sets of the PA linear or dual matrix type.

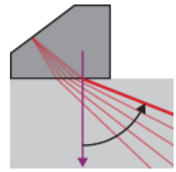


Figure 10-16 Refracted angle stop measurement

NOTE

For probe sets of the UT or TOFD type, or for linear or single beam sets, the **Refracted Angle Stop** column box is unavailable.

For probe sets of the PA linear or dual matrix type, the value can also be edited. When the stop refracted angle value is confirmed, the angle value in the **Refracted Angle Start** column box is automatically updated if required, as follows:

- If the start angle value is greater than the stop angle value, the start angle value is updated to equal the stop angle value.
- If steps have been configured, they are subtracted from the stop angle value. As such, the updated start angle value is closest to, but not smaller than, the currently set value. For example, for a start angle value set to 20°, a stop angle value of 21°, and steps of 0.3°, the angles will be 20°, 20.3°, 20.6°, and 20.9°.

When the start angle value is updated, it is always calculated from the last computed start angle value. If you modify the stop refracted angle value several times, the values computed between the start angle and stop angle values will drift toward the stop angle value.

Refracted Angle Step column ()

In the **Refracted Angle Step** column box, configure the angle of a sectorial group that is to be used as the stepping value to create all the beams in the group (see Figure 10-17 on page 152). The **Refracted Angle Step** column box is not available for single type groups.

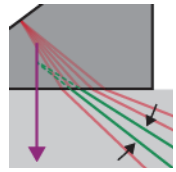


Figure 10-17 Refracted angle resolution

For a sectorial or a compound group, this value represents the angle that must be added to create each beam from the start refracted angle to the stop refracted angle. For a linear group, this value represents the number of elements that must be added to the center of each beam in order to create the next beam in the group. The default value is **1.0**.

10.7.4 Configuring Skew Angle Parameters

When the selected probe-set type is dual matrix, the **Groups** tab can be used to configure the skew angle parameters in the **Skew Angle** section.

NOTE

For PA linear, TOFD, and UT probe-set types, the Skew Angle section is unavailable. Also, the Skew Angle section is unavailable for some dual matrix probe configurations (for example, A27 with 2 × 16 elements).

To configure group skew angle parameters

◆ From the **Skew Angle** section, configure the following parameters:

Skew Start column ()

In the **Skew Start** column box, configure the skew angle value for the first beam of the group (see Figure 10-18 on page 153). The default value is **0°** (zero degrees).

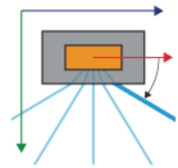


Figure 10-18 Start skew angle

When the skew start angle value is confirmed, the value in the **Skew Stop** column box is automatically updated if required, as follows: if the stop angle value is lower than the start angle value, the skew stop angle value is updated to equal the skew start angle value.

Skew Stop column ()

In the **Skew Stop** column box, configure the stop skew angle value for the last beam of the group (see Figure 10-19 on page 153). The default value is zero degrees (**0**°).

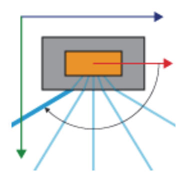


Figure 10-19 Stop skew angle

When the stop skew angle value is confirmed, the value in the **Refracted Angle Start** column box is automatically updated if required, as follows: if the start skew angle value is greater than the stop skew angle value, the start skew angle value is updated to equal the stop skew angle value.

Skew Step column ()

From the **Skew Step** column box, configure the skew angle value between the beams in the group (see Figure 10-20 on page 154). The skew angle step value is added to the start skew angle or each generated beam, up to the stop skew angle. The default value is **5**°.

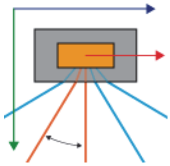


Figure 10-20 Skew angle step

10.7.5 Configuring Element Parameters

On the **Groups** tab, the element parameters can be configured in the **Elements** section.

To configure group element parameters

◆ In the **Elements** section, configure the following parameters:

First Element column ()

In the **First Element** column box, select the number of the first element to be used for the first beam (the first focal law) of the group (see Figure 10-21 on page 154). The default value is **1**.

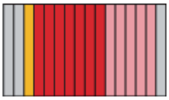


Figure 10-21 The first element displayed in yellow

When the first element value is modified, the last element value in the **Last Element** column box is automatically updated if required, as follows:

- If the first element value is greater than the last element value, the last element value is updated to equal the first element value.
- If steps have been configured, they are added to the first element value. As such, the updated last element value is closest to, but not smaller than, the currently set value. For example, if the first element value is set to 1, the last element value is set to 6, and the step value is set to 2 elements, the start elements will be 1, 3, and 5.

For a group of the linear type, when the value in the **First Element** column box is modified, the value in the **Element Quantity** column box is automatically updated if required. If adding the element quantity (aperture) value to the first element value results in a last element value greater than the currently configured last element value, the element quantity value is automatically reduced to correspond to the configured last element value. When the last element value is updated, it is always calculated based on the last computed value. If you modify the first element value several times, the values computed for the last element value will drift toward the first element value.

NOTE

For dual matrix groups, the value of the First Element parameter depends on the probe element configuration. If a probe configuration is 4 × 7, the First Element parameter is 1, 8, 15, or 22.

Last Element column ()

For a group of the linear type, in the **Last Element** column box, select the number of the last element to be used for the last beam (focal law) of the group (see Figure 10-22 on page 155). The default value is **16**. The **Last Element** column box is not available for a group of the **Sectorial** or **Single** type.

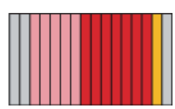


Figure 10-22 The last element displayed in yellow

When the last element value is modified, the first element value in the **First Element** column box is automatically updated if required, as follows:

- If the first element value is greater than the last element value, the first element value is updated to equal the last element value.
- If steps have been configured, they are subtracted from the last element value. As such, the updated first element value is closest to, but not smaller than, the currently set value.

When the value in the **First Element** column box is modified, the value in the **Element Quantity** column box is automatically updated if required. If adding the element quantity (aperture) value to the first element value results in a last element value greater than the currently configured last element value, the element quantity value is automatically reduced to correspond to the configured last element value.

When the last element value is updated, it is always calculated from the last computed value. If you modify the first element value several times, the values computed will drift toward the last element value.

Element Step column ()

For a group of the PA linear type, in the **Element Step** column box, configure the element step value, the number of elements between the first element of a beam (focal law), and the first element of the next beam in the group (see Figure 10-23 on page 156). The element step value is added to the start element for each beam (focal law) generated.

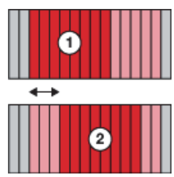


Figure 10-23 The element step between the first elements of beams

Element Quantity column ()

In the **Element Quantity** column box, select the number of elements that comprise the element quantity (aperture) for the beams (focal laws) of the group (see Figure 10-24 on page 156). The default value is **16** elements.

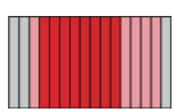


Figure 10-24 The element quantity

The element quantity value is combined with the step, the first element, and the last element values to generate the beams (focal laws) for the group.

For a group of the linear type, when the value in the **First Element** column box is modified, the value in the **Element Quantity** column box is automatically updated if required. If adding the element quantity value to the first element value results in a last element value greater than the currently configured value, the element quantity value is automatically reduced to correspond to the configured last element value.

NOTE

For dual matrix groups, the value of the Element Quantity parameter depends on the probe element configuration. If a probe configuration is 4 × 7, the Element Quantity parameter is 7, 14, 21, or 28.

10.7.6 Configuring Focusing Parameters

The **Focusing** section on the **Groups** tab can be used to configure the group parameters.

To configure group focusing parameters

◆ In the **Focusing** section, configure the following parameters:

Distance column ()

In the **Distance** column box, configure the distance (in true depth for a plate part or in half path for a tube or a bar part) at which the beams (focal laws) that form the group will intersect; in other words, the distance at which focusing should occur (see Figure 10-25 on page 157).

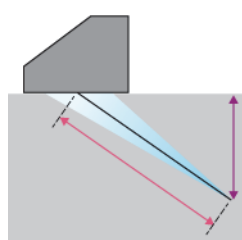


Figure 10-25 Focusing distance

This focusing distance is defined based on the type selected in the **Focusing Type** column. The default value is **50** mm with the true-depth focusing type for a group associated with a probe set of the PA linear type.

NOTE

Beam focus points are displayed as green dots in RayTracing view (see Figure 10-26 on page 158).

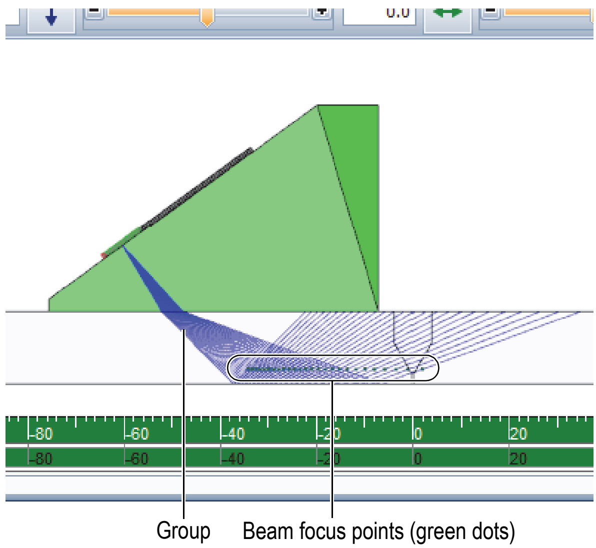


Figure 10-26 Example of beam focus points represented in RayTracing view

Focusing Type column ()

In the **Focusing Type** column box, choose one of the three available types of focusing distance measurement: **True depth (mm** or **in.)**, **True depth (% of thickness)**, or **Half path** (see Figure 10-27 on page 159).

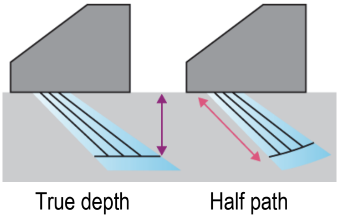


Figure 10-27 Focusing type

Legs column ( )

In the **Legs** column box, configure the number of legs, or half skips, that are displayed in RayTracing view for the group (see Figure 10-28 on page 159 and Figure 10-29 on page 159).

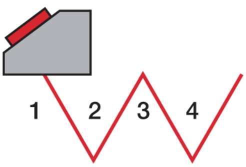


Figure 10-28 Legs

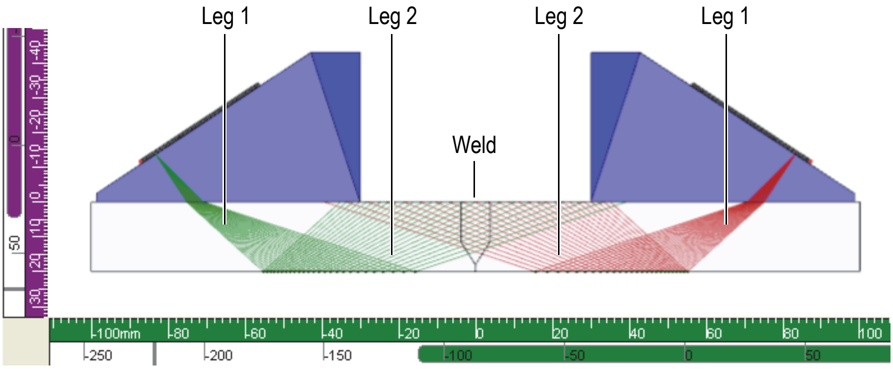


Figure 10-29 The legs for groups of two probe sets with 90° and 270° skew angles

The default value is **2** legs for groups associated with probe sets of the PA linear and UT types, and **1** leg for groups associated with probe sets of the TOFD and dual matrix types.

NOTE

For probe sets of the TOFD and dual matrix types, the **Legs** column box value cannot be modified.

11. Exporting Setup Files to an OmniScan Instrument

In order to efficiently manage inspection setups for OmniScan instruments, setup files can be exported from NDT SetupBuilder to OmniScan MX2 and OmniScan SX instruments.

To export setup files from NDT SetupBuilder to an OmniScan, the computer must be equipped with either a USB port or an internal or external SD card reader.

The workspace configuration can be exported in one of the following two formats:

- Connectivity (.ondtsetup)

  The resulting connectivity file contains the group configurations from the NDT SetupBuilder workspace file, which can be imported into an OmniScan instrument. The groups are then converted into groups in the OmniScan. The beams (focal laws) are automatically calculated according to the hardware available in the OmniScan.

  For more information about exporting a connectivity file from your workspace, see “Exporting a Workspace As a Connectivity File” on page 162.
- LAW (.law)

  The resulting .law file contains the group configurations from the NDT SetupBuilder workspace file, which can be imported into an OmniScan instrument.

  To create a .law file, your NDT SetupBuilder workspace file must contain at least one PA linear probe set and one PA linear group. For more information about the .law file format, see “Description of the .law File Format” on page 179.

  For more information about exporting .law files from your workspace, see “Exporting a Workspace as a .law File” on page 164.

The setup file can be saved on a USB key or SD card. The media file containing the setup file you want to import is then inserted into the OmniScan.

When a workspace configuration is not supported by the OmniScan (such as long seam, compound scan, dual matrix, or PA linear pitch-catch), NDT SetupBuilder allows files to be exported in the .law format. In such case, this is the only available option, because the export to connectivity option is disabled.

11.1 Exporting a Workspace As a Connectivity File

Your workspace can be exported from NDT SetupBuilder as a connectivity file, which has the .ondtsetup extension. The exported file can then be imported into an OmniScan instrument using a USB key or SD card. On import into the OmniScan, the parameters configured in the workspace are verified. If the configured parameters are valid, they are computed and groups are generated.

NOTE

Only visible groups associated with visible probe sets are exported. The entire setup configuration will be verified and computed after being imported into an OmniScan instrument. At this stage, the configuration may be declared invalid if it is not entirely supported by the OmniScan.

For more information about importing .ondtsetup files to the OmniScan, refer to the user's manual for the OmniScan MXU 3.2 software, or a more recent version.

Only a limited number of inspection setup configurations are supported within NDT SetupBuilder for exporting an inspection setup as a connectivity file. For more information, see Table 9 on page 163.

Table 9 Supported setup configurations for exporting to MXU 3.2 software using connectivity files

Probe skew	PA linear[a]	UT	TOFD	Focusing (true depth)[b]
Plate inspection				
0°	✓	✓	✓	✓
90°	✓	✓	✓	✓
180°	✓	✓	✓	✓
270°	✓	✓	✓	✓
Pipe and bar — circumferential inspection				
0°	—	—	—	—
90°	✓	✓	✓	✓
180°	—	—	—	—
270°	✓	✓	✓	✓
Pipe and bar — axial inspection				
0°	✓	✓	✓	✓
90°	—	—	—	—
180°	✓	✓	✓	✓
270°	—	—	—	—

a. Single, linear, and sectorial scans
b. When a connectivity .ondtsetup file that was exported from an inspection setup is imported in an OmniScan instrument, only true depth focusing is supported.

NOTE

If an inspection setup contains deactivated probe elements or a wedge with a roof angle value other than 0 (zero), it can only be exported to an OmniScan instrument as one or more .law files. For more information on exporting a setup as a .law file, see “Exporting a Workspace as a .law File” on page 164.

For more information about deactivating probe elements, see “Deactivating and Activating Phased Array Probe Elements” on page 110.

For more information about configuring a roof angle on a custom wedge, see “Configuring Custom Wedge Model Parameters” on page 124.

To export a workspace as an .ondtsetup file

1. On the **File** menu, point to **Export**, and then click **Connectivity**.
2. In the **Save As** dialog box, choose the folder where you want to save the file (see figure Figure 11-1 on page 164).
3. Click **Save**.

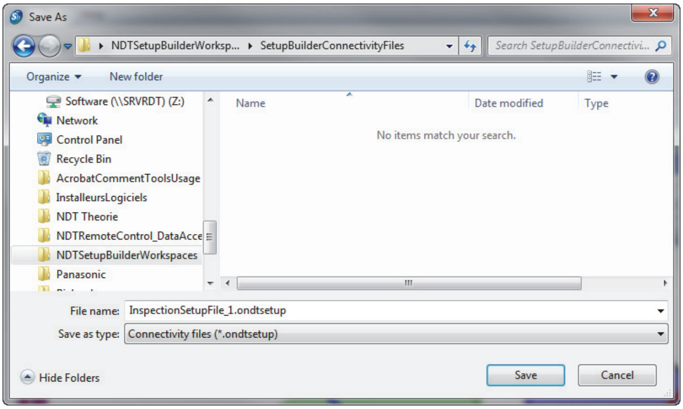


Figure 11-1 The Save As dialog box when saving a workspace as an .ondtsetup file

11.2 Exporting a Workspace as a .law File

NDT SetupBuilder can be used to export groups from your workspace into a .law file. The exported file can then be imported into an OmniScan instrument using a USB key or SD card. The groups imported into an OmniScan can be executed without the need for additional computation within the instrument, making it possible to conduct inspections that OmniScan instruments cannot perform natively.

To export a group, it must be selected in the **Groups** tab, following which it will be exported as a .law file. For more information about the .law file format, see “Description of the .law File Format” on page 179.

The following parameters are set using the following values:

- Channel gain: 0
- Voltage: 40
- Channel pulse width: 50
- Cycles: 1
- Filter: 0

NDT SetupBuilder automatically creates the file name, which is the name of the selected group.

For more information about importing a .law file to the OmniScan, refer to the user’s manual for the OmniScan MXU 3.2 software, or a more recent version.

NOTE

An inspection setup that contains deactivated probe elements or a wedge with a roof angle value other than 0 (zero) can only be sent to an OmniScan instrument as one or more .law files. For more information on exporting a setup as a .law file, see “Exporting a Workspace as a .law File” on page 164.

For more information about deactivating probe elements, see “Deactivating and Activating Phased Array Probe Elements” on page 110.

For more information about configuring a roof angle on a custom wedge, see “Configuring Custom Wedge Model Parameters” on page 124.

Only a limited number of inspection setup configurations are supported within NDT SetupBuilder for exporting an inspection setup as a .law file. For more information, see Table 10 on page 166.

Table 10 Supported setup configurations for exporting to MXU 3.2 software using .law files

Probe skew	PA linear[a]	UT	TOFD	Focusing[b]	Element deactivation	Roof angle	Dual matrix
Plate inspection							
0°	✓	✓	✓	✓	✓	✓	—
90°	✓	✓	✓	✓	✓	✓	✓
180°	✓	✓	✓	✓	✓	✓	—
270°	✓	✓	✓	✓	✓	✓	—
Pipe and bar — circumferential inspection							
0°	✓	✓	✓	✓	✓	✓	—
90°	✓	✓	✓	✓	✓	✓	✓
180°	✓	✓	✓	✓	✓	✓	—
270°	✓	✓	✓	✓	✓	✓	—
Pipe and bar — axial inspection							
0°	✓	✓	✓	✓	✓	✓	—
90°	✓	✓	✓	✓	✓	✓	—
180°	✓	✓	✓	✓	✓	✓	—
270°	✓	✓	✓	✓	✓	✓	—

a. Single, linear, compound, and sectorial scans
b. Half path and true depth

To export the current group as a .law file

1. On the **File** menu, point to **Export**, and then click **Laws**.
2. In the **Save As** dialog box, select the folder where you want to save the file (see Figure 11-1 on page 164).
3. Click **Save**.

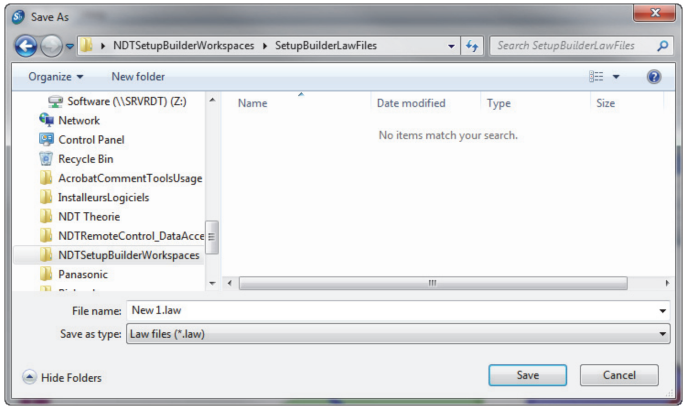


Figure 11-2 The Save As dialog box when saving a workspace as a .law file

12. Generating Reports

NDT SetupBuilder allows you to generate an HTML report of your inspection setup, which is automatically displayed in your web browser. Reports generated by NDT SetupBuilder use the same format as reports generated by the OmniScan.

The report contains all the probe sets and groups of the inspection setup defined in the workspace.

To generate a report

- On the **File** menu, click **Generate Report**.

When generated, the report automatically opens in your default web browser (see Figure 12-1 on page 170). In the browser, you can save the report as an HTML or PDF file or send it to a printer.

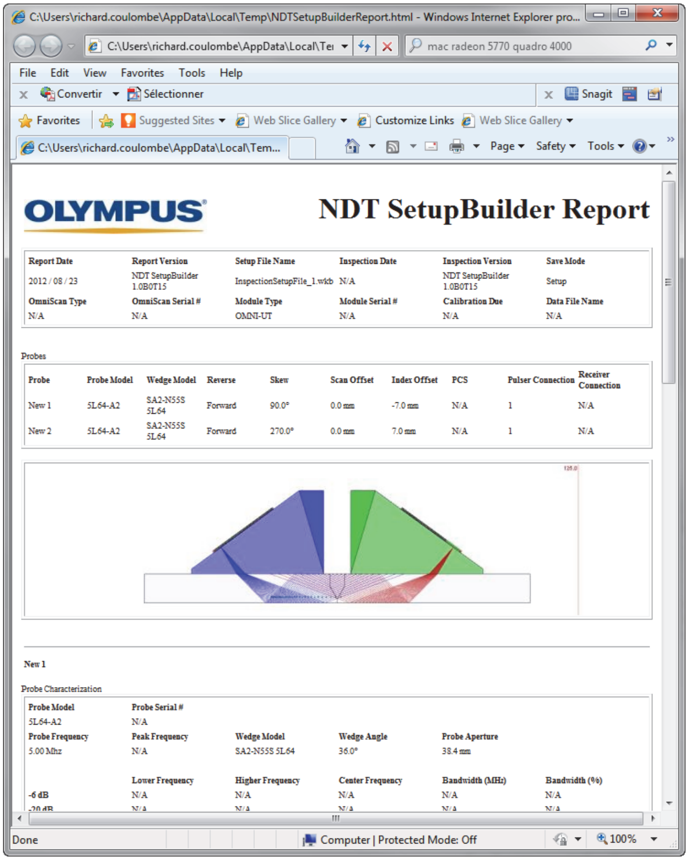


Figure 12-1 NDT SetupBuilder report displayed in Internet Explorer

When saving a report in HTML format in the Microsoft Internet Explorer web browser, in the **Save Webpage** dialog box, make sure that you select the **Webpage HTML only (*.htm, *.html)** option in the **Save as type** list (see Figure 12-2 on page 171).

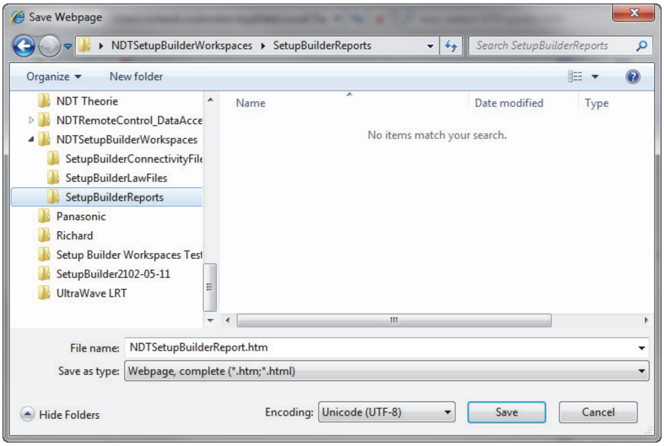


Figure 12-2 The Save Webpage dialog box from MS Internet Explorer

NOTE

The images of your report that are displayed in the web browser are optimized for printing.

Appendix A: Phased Array Technique

This section describes the main concepts relating to phased array inspection and phased array data views.

Phased array technology makes it possible to generate an ultrasonic beam and to modify ultrasonic beam parameters such as angle, focal distance, and focal spot size using software. For example, it is possible to quickly vary the angle of the ultrasonic sectorial beam in order to scan a part or weld without having to move the probe itself. Furthermore, this ultrasonic beam can be multiplexed over a large array, thus creating movement of the ultrasonic beam along the array.

Phased array capabilities also enable the replacement of multiple probes and mechanical scanning devices. Inspecting a part or weld with a variable-angle ultrasonic beam also improves detection, regardless of the defect orientation, and optimizes signal-to-noise ratio.

A.1 Physical Principles

To generate an ultrasonic beam, the various probe elements are pulsed at slightly different times. By precisely controlling the delays between the probe elements, ultrasonic beams of various angles, focal distances, and focal spot sizes can be produced. As shown in Figure A-1 on page 174, the echo from the desired focal point hits the various probe elements with a computable time shift. The echo signals received at each probe element are time-shifted before being summed together. The resulting sum is an A-scan that emphasizes the response from the desired focal point and attenuates various other echoes from other points in the material.

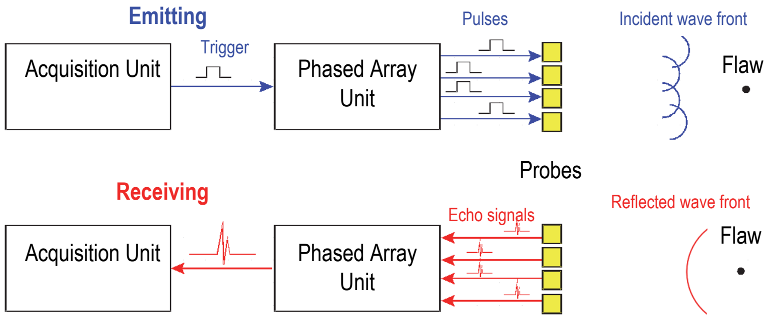


Figure A-1 Emitting and receiving in a phased array system

A phased array probe is typically a one- or two-dimensional array of small probe elements. To control the ultrasonic beam characteristics, the excitation pulse is applied at different times to the various elements of the probe.

The phased array probe is composed of multiple elements that allow for ultrasonic beam angle control (see "Beam Angle Control" on page 174) and ultrasonic beam focus control (see "Beam Focus Control" on page 176).

A.1.1 Beam Angle Control

Beam angle control involves the production of a wave front. As shown in Figure A-2 on page 175, simultaneous firing of all elements of a linear multielement probe produces a series of circular arc waves (one from each probe element). Because all wave fronts are at the same distance from their respective emitter, the resulting wave front, or envelope, is parallel to the probe plane. This is, in fact, very similar to pulsing a single element probe of the same size.

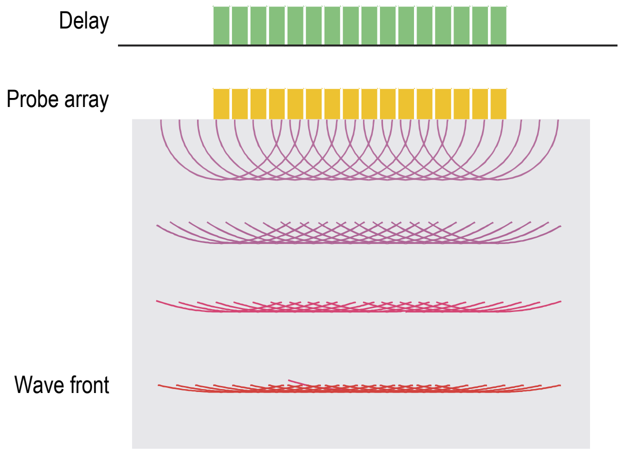


Figure A-2 Ultrasonic wave front of a linear array

The phased array unit enables pulsing of the elements in a sequential manner with a small and precisely controlled time delay between each element. Sequential firing of the various probe elements produces a series of circular arc waves in a wave front, or envelope, that is no longer parallel to the probe surface, but that propagates at an angle (see Figure A-3 on page 175). The pulse delays can be adjusted to produce any desired wave-front angle.

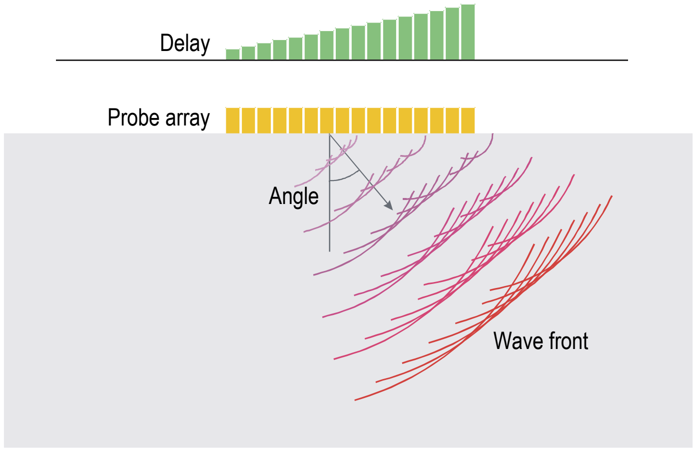


Figure A-3 Ultrasonic beam angle control of a linear array

A.1.2 Beam Focus Control

When generating a focused beam, the delays are adjusted so that all individual wave fronts stay in phase along the path leading to the desired focal point, while canceling each other out at all other points. By accurately controlling the pulse delays, it is possible to focus the beam at a desired point (see Figure A-4 on page 176).

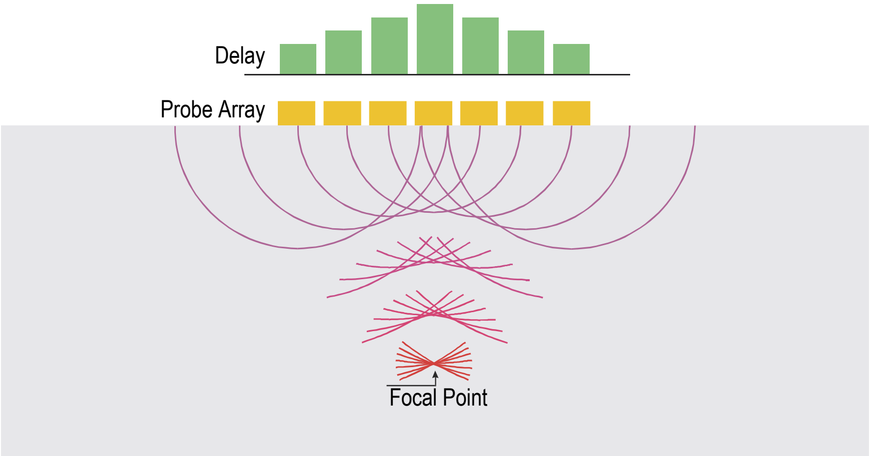


Figure A-4 Ultrasonic beam focusing of a linear array

For beam angle control and beam focus control, signals received by every element are time-synchronized by the phased array system prior to summing the various responses.

A.2 Group Types

Phased array offers the possibility of performing inspections with various angles and focal lengths (focal length is referred to as focusing distance in NDT SetupBuilder).

The two types of groups used for phased array inspection are sectorial groups and linear groups.

A.2.1 Sectorial Groups

For certain applications, conventional UT inspection would require use of a number of different probes, whereas a single phased array probe can be used to sequentially produce the various angles and focal points required by the application (see Figure A-5 on page 177).

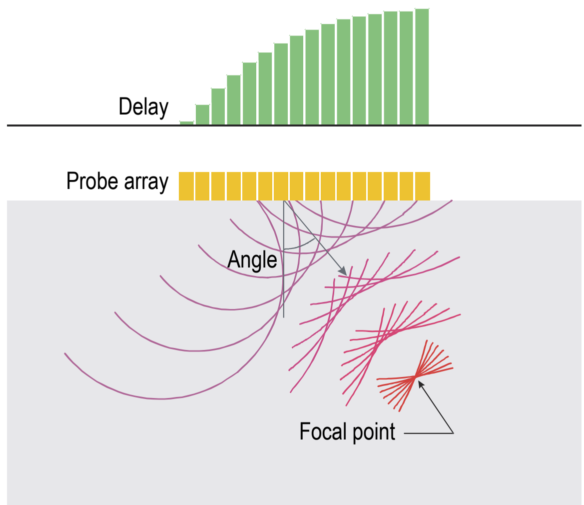


Figure A-5 Ultrasonic beam angle control and focusing of a linear array

Sectorial scanning of the phased array signal is obtained by applying several beams in sequence at each X-Y coordinate of the inspected area.

At each X-Y coordinate of the inspection sequence, an array of elements is used to deflect the ultrasound beam without moving the probe. Scanning can be conducted along a horizontal axis (see Figure A-6 on page 178).

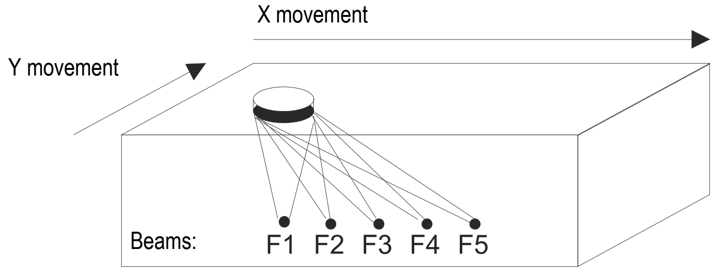


Figure A-6 Sectorial scanning of X-axis using phased array deflection

A.2.2 Linear Groups

For large phased array probes containing a high number of elements, the phased array unit is capable of applying the same beam to different sets of elements. By moving the beam along a probe array, the scanning of an inspection axis is realized electronically without any need for physical displacement of the probe (see Figure A-7 on page 178).

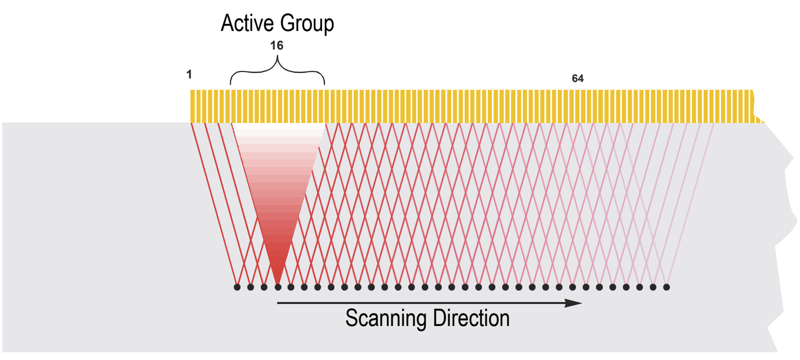


Figure A-7 Electronic scanning along an axis

In Figure A-7 on page 178, a focused beam is created using a few of the many probe elements of a long phased array probe. The beam is then shifted (or multiplexed) to the other elements in order to perform a high-speed scan of the part with no probe movement along the scanning axis. More than one scan can be performed with various inspection angles.

Appendix B: Description of the .law File Format

This appendix describes the .law file format used to create specific beam configurations. The .law file format is contained in text files that can be loaded directly in the OmniScan.

B.1 General Format

This section details the .law file format and provides examples for version 5.0 and version 5.2.

B.1.1 Format

This section describes the .law file format.

Law file version 5.0

The .law file format for version 5.0 is described as follows:

```
<Version> <Number of Laws>

<Number of Active Elements> <Unused1> <Unused2> <Sum Gain> <Unused3> <Unused4>
    <Refracted Angle> <Beam Skew Angle> <First Pulser> <First Receiver> <Scan
    Offset> <Index Offset> <Global Delay> <Focusing Depth> <Material Velocity>

<Element Number> <Beam Gain> <Transmission Delay> <Reception Delay> <Amplitude>
    <Pulse Width>
```

Law file version 5.2

The .law file format for version 5.2 is described as follows:

```
<Version> <Number of Laws>

<Element Number> <Frequency> <Cycles> <Sum Gain> <Mode> <Filter> <Refracted
    Angle> <First Pulser> <First Receiver> <Scan Exit> <Index Exit> <Global
    Delay> <Focusing Depth> <Material Velocity> <Probe Skew Angle> <Beam Skew
    Angle>
```

B.1.2 Examples

Law file version 5.0

The following is an example of a .law file containing two beams of 10 elements each:

```
V5.0 2
16 300 1 -1 1 0 400 900 1 1 0 -46040 13869 50000 3240
1 0 570 570 40 100
2 0 539 539 40 100
3 0 508 508 40 100
4 0 475 475 40 100
5 0 442 442 40 100
6 0 407 407 40 100
7 0 372 372 40 100
8 0 335 335 40 100
9 0 297 297 40 100
10 0 258 258 40 100
11 0 218 218 40 100
12 0 176 176 40 100
13 0 134 134 40 100
14 0 90 90 40 100
15 0 46 46 40 100
16 0 0 0 40 100
16 300 1 -1 1 0 500 900 1 1 0 -44142 14282 50000 3240
1 0 175 175 40 100
2 0 168 168 40 100
3 0 161 161 40 100
4 0 153 153 40 100
5 0 145 145 40 100
6 0 136 136 40 100
7 0 126 126 40 100
8 0 115 115 40 100
9 0 103 103 40 100
10 0 91 91 40 100
11 0 78 78 40 100
12 0 64 64 40 100
13 0 49 49 40 100
14 0 34 34 40 100
15 0 17 17 40 100
16 0 0 0 40 100
```

Law file version 5.2

The following is an example of a .law file for version 5.2:

```
V5.2 2
16 300 1 -1 1 0 400 1 1 -46040 0 13869 50000 3240 900 0
1 0 570 570 40 100
2 0 539 539 40 100
3 0 508 508 40 100
4 0 475 475 40 100
5 0 442 442 40 100
6 0 407 407 40 100
7 0 372 372 40 100
8 0 335 335 40 100
9 0 297 297 40 100
10 0 258 258 40 100
11 0 218 218 40 100
12 0 176 176 40 100
13 0 134 134 40 100
14 0 90 90 40 100
15 0 46 46 40 100
16 0 0 0 40 100
16 300 1 -1 1 0 500 1 1 -44142 0 14282 50000 3240 900 0
1 0 175 175 40 100
2 0 168 168 40 100
3 0 161 161 40 100
4 0 153 153 40 100
5 0 145 145 40 100
6 0 136 136 40 100
7 0 126 126 40 100
8 0 115 115 40 100
9 0 103 103 40 100
10 0 91 91 40 100
11 0 78 78 40 100
12 0 64 64 40 100
13 0 49 49 40 100
14 0 34 34 40 100
15 0 17 17 40 100
16 0 0 0 40 100
```

B.2 Object Description

The .law file object is defined by general parameters related to the file format and by law parameters related to the beams.

B.2.1 General Parameters

The following general parameters are related to the .law file format:

Version

The version of the .law file in the following format: <V> <number> <'.'> <number>.

Number of Laws

The total number of beams defined in the file. This value ranges from 1 to 256.

B.2.2 Law Parameters

This section provides descriptions of the beam-related parameters in the .law file.

Number of Active Elements

The number of active elements used to generate a beam. This value ranges from 1 to 32, and it is determined by the limits of the hardware.

Frequency

Pulse train frequency for the given law, in kilohertz. The default value 300 is not used. The value ranges from 300 to 2000. Used only for EMAT (electromagnetic acoustic transducer).

Cycles

The number of cycles in the pulse train for the given law. The default value is 1. The value ranges from 1 to 15. Used only for EMAT.

Unused1

Unused. Always set to 1000.

Unused2

Unused. Always set to 10.

Sum Gain

The gain working range for a given law, in decibels (dB). Sum Gain is an attenuation value that varies according to the number of active elements. This value is dependent on the instrument that is being used. The value −1 is the Auto Sum gain, which is defined by the hardware. The value ranges from −1 to 30.

Unused3

Unused. Always set to 0.

Unused4

Unused. Always set to 0.

Mode

The inspection mode for the given law:

- 0 = T/R (transmit/receive), which has different pulser and receiver elements
- 1 = Pulse-echo, which has the same pulser and receiver elements

Filter

Specifies the filter applied at reception:

- 0 = no filter (0.5 MHz to 20 MHz)
- 1 = 0.5 MHz to 5 MHz
- 2 = 2.0 MHz to 10 MHz
- 3 = 5.0 MHz to 15 MHz

Refracted Angle

The refracted angle for the given law, expressed in tenths of degrees. The parameter-value range is as follows:

- For .law file version 5.0: 0 to 900.
- For .law file version 5.2: –900 to 900.

Beam Skew Angle

The skew angle for the given law, expressed in tenths of degrees. The parameter-value range is as follows:

- For .law file version 5.0: 0 to 3599.
- For .law file version 5.2: –900 to 900.

NOTE

The Beam Skew Angle is defined only for the beam, and initially assumes that the probe skew angle is 0. The probe skew angle (which determines the scan orientation) must be entered after the .law file is downloaded to the acquisition software.

Probe Skew Angle (version 5.2)

The probe skew angle for the given law, in degrees. The value ranges from 0 to 3599.

First Pulser

Specifies the number of the first pulser hardware connection used for transmission during focal law formation (whether pulsing or not). The value is a positive integer determined by the hardware (instrument and probe).

First Receiver

Specifies the number of the first receiver hardware connection used during focal law formation (whether receiving or not). The value is a positive integer determined by the hardware (instrument and probe).

Scan Offset

The offset of the scan-axis exit point (for the given law) relative to the probe set's origin (the mechanical reference point), expressed in micrometers.

Index Offset

The offset of the index-axis exit point (for the given law) relative to the probe set's origin (the mechanical reference point), expressed in micrometers.

Global Delay

Specifies the global delay (GD) expressed in nanoseconds (ns). The value is a non-negative integer, and is calculated as follows:

GD = ED + WD + LD

ED: electronic delay that is determined by the hardware. The value is 0 for OmniScan instruments.

WD: total wedge delay (transmission and reception).

LD: law delay (global delay introduced by the specified law).

Focusing Depth

Focusing distance, which is a true depth expressed in micrometers. The value is a non-negative integer.

Material Velocity

Specifies the propagation velocity in the material, in meters per second (m/s). The value is a non-negative integer.

Element Parameters

Parameters that are related to the individual elements in a defined beam.

Element Number

The number identifying the individual element of the phased array probe relative to the first pulser and the first receiver (see "First Pulser" on page 184 and "First Receiver" on page 184). The element numbers are consecutive (1, 2, 3, …). Inactive elements are disabled by setting the Transmission Delay and Reception Delay to 65535 (see “Transmission Delay” on page 185 and “Reception Delay” on page 185). The value must be less than or equal to the maximum number of hardware elements. The value is a positive integer.

Beam Gain

The gain applied to the considered beam, in decibels (dB). Admitted range: 0–80. Elements of the same beam must have the same focal law gain.

For .law files generated in offline mode (that is, without phased array equipment connected), this parameter has the default value 0.

Transmission Delay

Specifies the transmission delay for the specified active element. The delay is expressed in nanoseconds, and must be between 0 and 25600. The transmission is deactivated if 65535 is used.

Reception Delay

Specifies the reception delay for the specified active element. The delay is expressed in nanoseconds and must be between 0 and 25600. The reception is deactivated when 65535 is used.

Amplitude

The excitation amplitude for the specified active element, expressed in volts (range: 50–200). The value must the same for all elements of all beams defined in the current .law file.

For .law files generated in offline mode (that is, without phased array equipment connected), this parameter has the default value 180.

Pulse Width

The pulse-width value is applied to the specified active element, expressed in nanoseconds (range: 50–500). The value must be the same for all elements of all beams defined in the current .law file. Generally, the pulse width value can be confirmed as follows:

Pulse Width (ns) = 500 ÷ Probe Frequency (MHz)

List of Figures

List of Tables