RAM Optical Instrumentation

A Quality Vision International Company

eBx™

Reference Guide

This document was produced by the Marketing Communications Department of Quality Vision International, Inc. 850 Hudson Ave., Rochester, New York 14621-4896 USA.
Telephone: 585-544-0450. FAX: 585-544-0131. E-mail: webmaster@qvii.com.

Warranty

RAM Optical Instrumentation, Inc. (RAM™) warrants that the software will operate according to the specifications described in this manual and be free from known defects in materials or workmanship for a period of one year from the date of shipment. During this warranty period, RAM will, at its option, repair, replace or provide a workaround solution to any items that prove to be defective. In order to qualify for such warranty service, a complete description of the problem, with appropriate documentation (such as results, program listing, sample part and program) should be forwarded to RAM for our inspection. Such items will be returned to the customer if requested.

RAM does not warrant that the operation of the software will be uninterrupted or error-free. The information in this manual is subject to change without notice.

This warranty does not apply to defects resulting from customer-supplied or -configured computer equipment, operating systems or software, unauthorized alteration or misuse, or operation outside the environmental specifications for the product.

THESE WARRANTIES AND RAM'S LIABILITY HEREUNDER, ARE THE SOLE AND EXCLUSIVE WARRANTIES AND ARE EXPRESSLY IN LIEU OF ALL OTHER WARRANTIES, EXPRESS OR IMPLIED, INCLUDING THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. IN NO EVENT SHALL RAM BE LIABLE FOR ANY DIRECT, INDIRECT, SPECIAL, CONSEQUENTIAL, INCIDENTAL OR OTHER DAMAGES INCLUDING, AND NOT BY WAY OF LIMITATION, LOSS OF PROFITS, AND WITHOUT REGARD TO THE FORM OF THE ACTION OR THE NATURE OF THE CLAIM WHICH IS MADE.

The information contained herein is based on the experience and knowledge relating to the subject matter gained by RAM prior to publication. No patent license is granted by this information. RAM RESERVES THE RIGHT TO CHANGE THIS INFORMATION WITHOUT NOTICE AND MAKES NO WARRANTY, EXPRESS OR IMPLIED, WITH RESPECT TO THIS INFORMATION. RAM shall not be liable for any loss or damage, including consequential or special damages, resulting from the use of this information, even if loss or damage is caused by negligence or other fault on the part of RAM.

Due to the nature of this material, a number of hardware and software products may be mentioned by name. In most, if not all, cases these product names are claimed as trademarks by the companies that manufacture the products. It is not our intent to claim these names or trademarks as our own.

Third Printing 10-2009 Part Number 795051

Table Of Contents

1. Introduction

2. Getting Started

3. User Interface

4. Setting Up the Part

5. Using the Measurement Tools & Targets

6. Creating, Opening, & Saving a Routine

7. Entering Nominal Values & Tolerances

8. Measure Functions

9. Construct Functions

10. Running the Current Routine

11. Measurement Results & Output

12. Editing a Routine

13. Units & Coordinates

14. Configuration Settings

15. Calibration & Alignment

16. Diagnostics

17. Manual Rotary Motion

18. Using the CAD Navigator

19. MeasureFit Analysis Software

20. MeasureMenu Software

This page was left blank intentionally.

1. Introduction

1.1 Overview

Welcome!

RAM Optical Instrumentation (RAM) has produced this *eBx Reference Guide* as part of its ongoing effort to provide users with useful, comprehensive documentation. We use documentation standards and a design that enhance readability and make information easier to find.

This manual provides the comprehensive information you will need to use the eBx software.

If you have any questions that are beyond the scope of this manual, contact your local authorized RAM Sales or Service Representative.

The software described in this manual is based in part on the work of the Independent JPEG Group.

1.2 Technical Support & Customer Service

RAM Optical Instrumentation, Inc.

A Quality Vision International Company
1175 North Street
Rochester, NY 14621, USA

- Toll Free Support: 877-764-6397
- Email: *sales@ramoptical.com*
- Internet: *www.ramoptical.com*

1.3 Organization

The *eBx Reference Guide* contains the following sections:

Section 1, Introduction (this section),

- includes technical support and customer service information
- explains the warranty RAM provides
- describes the content and organization of this manual
- list related publications
- explains documentation conventions and special symbols used in this manual

Section 2, Getting Started,

- describes how to launch the software
- provides an overview of the tasks involved in creating a routine
- describes how to use the software on-line Help and context sensitive help
- describes how to exit from the software

Section 3, User Interface,

- describes the user interface
- outlines joystick, control panel, and mouse functions

Section 4, Setting Up the Part,

- describes how to stage a part
- describes how to perform the part setup
- describes how to use the datum functions

Section 5, Using the Measurement Tools & Targets,

- describes all of the eBx automatic tools and manual alignment targets

Section 6, Creating, Opening, & Saving a Routine,

- explains how to create a routine
- describes how to open an existing routine
- describes how to save the current routine

Section 7, Entering Nominal Values & Tolerances,

- describes how to enter nominal values and tolerances

Section 8, Measure Functions,

- explains how to use the various eBx measure functions

Section 9, Construct Functions,

- explains how to use the various eBx construct functions

Section 10, Running the Current Routine,

- describes how to run a routine
- covers the various routine run options and run override options
- provides tips for faster and more accurate measurements
- describes how to stop the system

Section 11, Measurement Results & Outputs,

- describes how to output measurement results

Section 12, Editing a Routine,

- explains the edit functions used to edit a routine

Section 13, Units & Coordinates,

- explains coordinate systems, units of measurement, and resolution (number of significant digits)

Section 14, Configuration Settings,

- explains all of the software configuration settings

Section 15, Calibration & Alignment,

- outlines the various options available in the System / Calibration menu

Section 16, Diagnostics,

- describes the eBx diagnostic tools

Section 17, Manual Rotary Motion,

- describes how to use the System / Rotary Move function
- describes how to enable digital I/O

Section 18, Using the CAD Navigator

- describes how to import a CAD file
- describes how to use the CAD Navigator
- describes how to generate measurement steps automatically

Section 19, MeasureFit Analysis Software

- describes the MeasureFit software
- includes guidelines you should observe while creating an eBx rotuine
- describes how to access the MeasureFit software

Section 20, MeasureMenu Software,

- describes the MeasureMenu software
- describes how to optimize the communication between the MeasureMenu software and eBx software

1.4 Related Documentation and Training

- *eBx Fast Start Guide* (PN 795050)
- *Touch Probe User's Guide* (PN 790424) [if the system is equipped with a touch probe]
- *DRS Laser User's Guide* (P/N 795054) [if the system is equipped with a DRS laser]
- *MeasureMenu Fast Start Guide* (P/N 790337) [if the system is equipped with the MeasureMenu software]
- Standard training part (PN 790012) for testing and self-training. It is used in all the examples in the Fast Start Guide.
- Application training to use, run, and program eBx

For information about additional documentation or training, contact your local RAM Sales or Service Representative.

1.5 Documentation Conventions

To help you locate, interpret, enter, or select information easily, this manual uses consistent visual cues and standard text formats. For example, capital letters (or upper-case letters) are used to call attention to certain words and to help eliminate ambiguity. These documentation conventions are explained in the table below.

Type Style; Symbol	Used for	Examples and Explanations
slash: /	• Selections from a main menu and submenu(s)	• Select **System / Exit**
italic or **bold**	• Emphasized words	• Select the *highest* magnification • **Do not** repeat this step
bold, sans-serif typeface	• Commands to be typed • Keys to be pressed • Buttons to be pressed or selected • Menu items to be selected	• Type **b:install**; type **Exit** • Press the **Enter** key • Select the **Done** button or press the **Start / Stop** button • Select **System / Calibration**
ALL CAPS	• Acronyms • File names • Path statements	• ASCII; RAM • Edit INPUT.INI • View C:\\W3_1
Initial Caps	• Proper nouns • Product names • Sections; figures	• System menu • Sprint CNC 300 • See Section 3

2. Getting Started

2.1 Overview

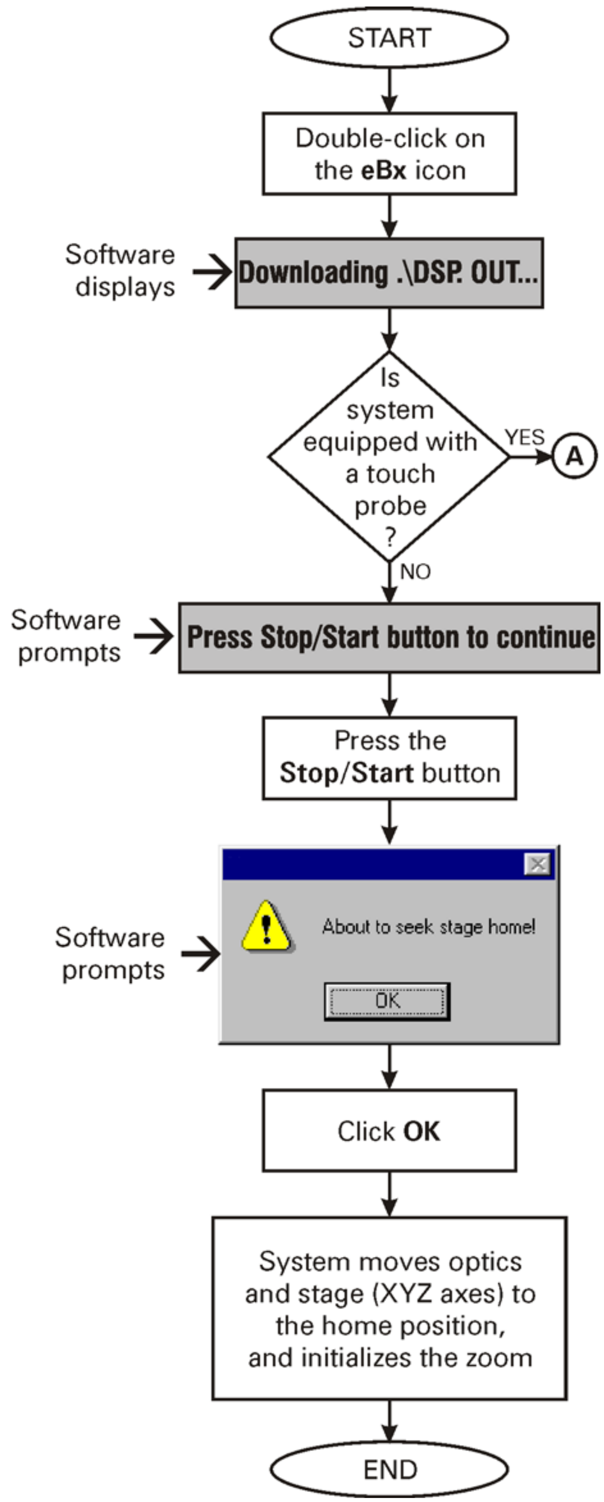
eBx is a powerful dimensional measurement software package. With it, you can use the latest optical, video and image processing technologies to measure parts quickly and accurately.

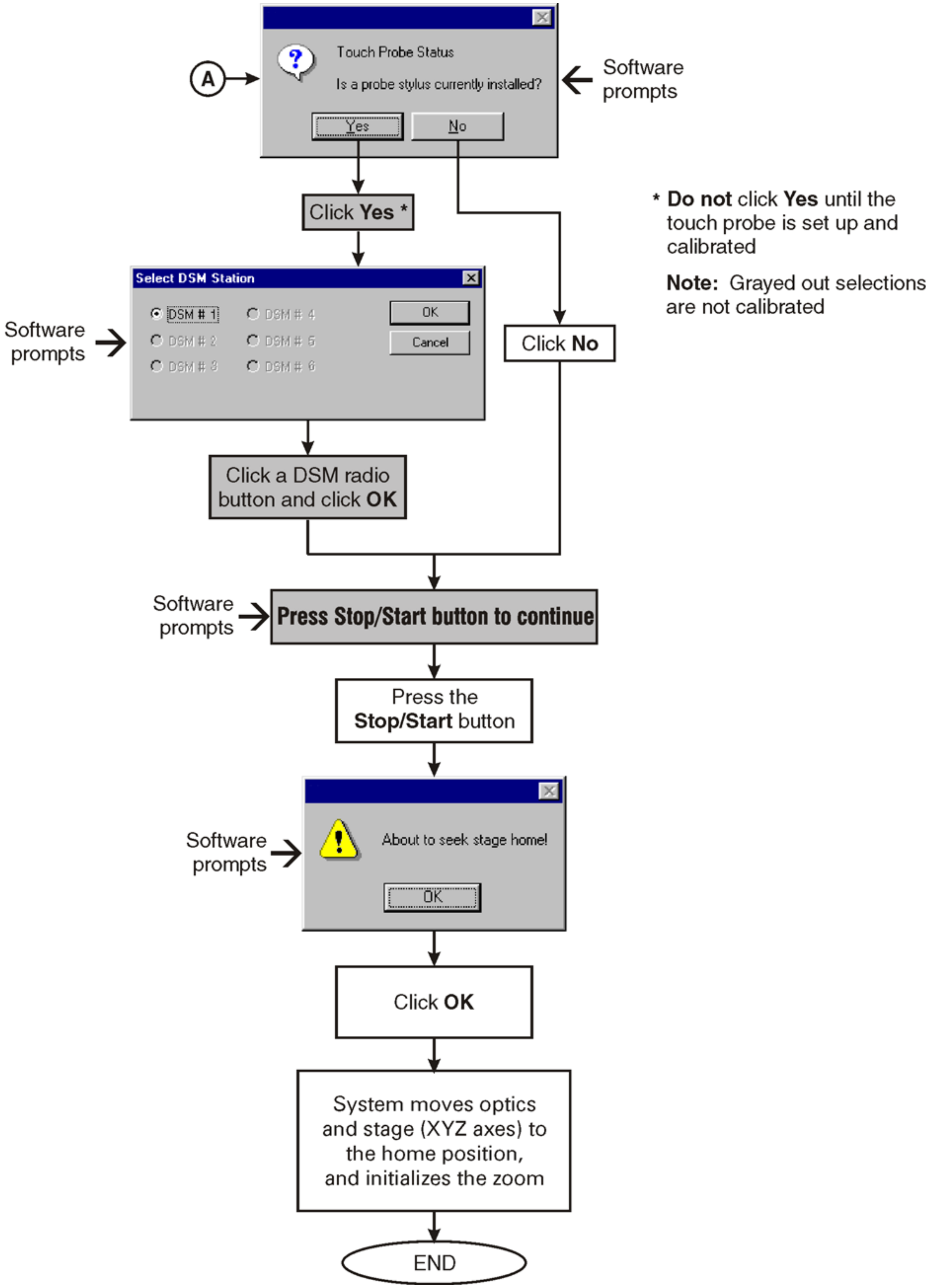
eBx features a full set of image processing tools, geometric functions, ANSI/ISO tolerancing, and real-time data transfer to SPC and custom reporting software. Support for multiple sensors such as touch probes, is integrated into the software.

To view the revision of the eBx software, select **Help** / **About** from the main menu.

This manual is current with version 4.2.x of the eBx software.

2.2 How to Launch the Software




A
Touch Probe Status
Is a probe stylus currently installed?
Yes
No
Software prompts
Click Yes *
* Do not click Yes until the touch probe is set up and calibrated
Note: Grayed out selections are not calibrated
Select DSM Station
DSM # 1
DSM # 2
DSM # 3
DSM # 4
DSM # 5
DSM # 6
OK
Cancel
Software prompts
Click No
Click a DSM radio button and click OK
Software prompts
Press Stop/Start button to continue
Press the Stop/Start button
Software prompts
About to seek stage home!
OK
Click OK
System moves optics and stage (XYZ axes) to the home position, and initializes the zoom
END

2.3 Overview of Tasks to Measure Features & Create a Routine

	Task	Described in…
1	Set up the part and clear previous measurements	Section 4
2	Add part setup instructions	Section 4
3	Become familiar with the proper tools	Section 5
4	Measure the datum features and define the datums	Section 4
5	Measure and construct features on the part	Sections 8 and 9
	Enter nominal values and tolerances	Section 7
	Save your work and run the routine	Section 10
6	Edit any desired measurement steps	Section 12
7	Output the measurement results	Section 11

If the part you want to measure has a corresponding CAD file, you can use the optional CAD Navigator to import the CAD file, and then use the CAD file to generate measurement steps automatically. Refer to Section 18 for more information.

2.4 Software Security

Since you have a license to operate the system software (you do not own it), system operation requires a software security dongle and associated license file. The software will operate only if these two security components are in place.

The software security dongle is a hardware security device that prevents unauthorized use of protected software and its options. The dongle is shipped with the manufactured system or with separately purchased software updates and options. The dongle has a corresponding license file for each protected application.

The license file contains information pertinent to the options that are enabled on your system, and is created for use with the dongle that has been programmed specifically for your system.

The dongle must be securely plugged into an addressable port on the system computer, and its accompanying license file(s) must be present in a specific directory.

You cannot launch and use the software when either the dongle or license file is not present. In the absence of a properly programmed dongle, or its accompanying license file, the system displays appropriate error messages that inform you that the application or option is not enabled, that the dongle is not addressable, or that the dongle or license file could not be found (note the error message that appears and then refer to *Troubleshooting Software Security*).

A new license file (not dongle) is required for each major release of the applications and options that it is supplied for (for example, eBx Rev. 3, eBx Rev. 4, etc.).

- The dongle will enable only the associated maintenance and minor releases. This means that a minor release (for example, revision 3.01) will run with the dongle issued for its major release (for example, revision 3).

- To ensure the software has full access to the required license file, do not deny read or write permission to the C:\Program Files\QVI\QVI License Manager\License Files directory or any files in this directory.

2.5 Troubleshooting Software Security

If you have trouble accessing software that is protected with the software security dongle and license file, the system displays the messages listed below.

- If you attempt to initialize protected software that is not enabled by the connected dongle, the following message appears:

  THE APLICATION YOU ARE TRYING TO ACCESS IS NOT ENABLED FOR USE, CONTACT QVI CUSTOMER SERVICE.

- If you attempt to initialize protected software when the dongle is disconnected, the following message appears:

  THE APPLICATION YOU ARE TRYING TO ACCESS REQUIRES THE PRESENCE OF A SECURITY DONGLE THAT CANNOT BE FOUND. MAKE SURE THE DONGLE SUPPLIED WITH THE SOFTWARE IS SECURELY SEATED IN AN ADDRESSABLE USB PORT AND TRY AGAIN.

- If you attempt to initialize protected software when the license file is absent, the following message appears:

  THE APPLICATION YOU ARE TRYING TO ACCESS REQURIES THE PRESENCE OF A LICENSE FILE THAT CANNOT BE FOUND. MAKE SURE THE LICENSE FILE SUPPLIED WITH THE SOFTWARE IS INSTALLED ON THE SYSTEM CONTROLLER AND TRY AGAIN.

- If the dongle is disconnected while the application is running, the application will pause and display the following message:

  THIS SOFTWARE APPLICATION HAS LOST CONTACT WITH THE SECURITY DONGLE THAT IT REQUIRES TO OPERATE. MAKE SURE THE DONGLE IS SECURELY SEATED IN AN ADRESSABLE USB PORT AND CLICK OK.

Certain applications and options may continue to operate even if the software security dongle is disconnected. In these cases, you are allowed to save any open part routines. However, the **File / New**, **File / Open**, and **File / Run** functions are disabled and you will receive an error message if you attempt to use any of these functions. To continue normal operation, reinsert or reseat the dongle and click **Retry**.

2.6 How to Use the On-Line Help

You can access and view the Help topics in the following ways:

- Click  (Help icon)
- Select **Help** / **Help Topics** from the main menu

The software displays the main Help viewer. It works much the same way as the Internet Explorer browser.

- The top pane displays the Help toolbar. It provides buttons for showing and hiding the left-hand pane, printing topics, going back to previous topics, and selecting other options.
- The navigation pane on the left side displays tabs for the Contents, Index, Search, Favorites, and Glossary functions.
- The topic pane on the right side displays topic content. If it does not fit into one window, use the scroll bar on the right to view the rest of the text.
    - You can jump to any topic that is shown in blue simply by clicking it. The color changes to indicate that the topic has already been viewed.
    - The selected book or page at the left is synchronized with the active topic on the right.
- The browse sequence bar is displayed above the topic pane on the right.
- The default setting is to keep the Help viewer open when you go back to using the software. This enables you to view a Help topic and use the software at the same time, for example, when you wish to follow a step-by-step procedure.
    - You can move the Help window by clicking its title bar and dragging it to the desired location.
    - You can change the size of the Help window by dragging its corners to the desired size.
    - The navigation pane is displayed as soon as you select the Help window again.

You may also view the context-sensitive Help within the measurement software.

2.6.1 Help Toolbar

The top pane of the Help viewer displays the Help toolbar. The buttons work much the same way as the Internet Explorer browser.

- Click the **Hide** button to hide the left-hand pane. If it is hidden, click this button to show the pane and tabs.
- Click the **Back** button to go back to the previous topic.
- Click the **Print** button to display the Print dialog for printing topics.
- Click the **Options** button to display a drop-down menu of options. For example, you can select whether to highlight text that you are searching for.

For additional information, also see *How to Use This Help* and *Context-Sensitive Help*.

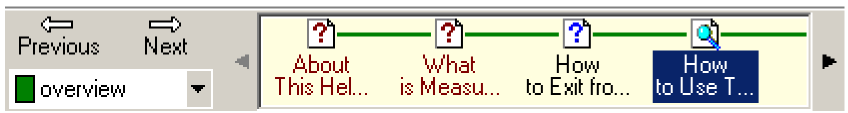
2.6.2 Navigation Pane

The Help navigation pane displays tabs for the Contents, Index, Search, and Favorites functions.

- Click the **Contents** tab to view the list of main Help topics, represented by icons of closed books. Double-click the desired topic to "open" the book and view a list of subtopics. Then double-click the desired topic to view it in the topic pane.
- Click the **Index** tab to view the list of all the index keyword entries. You can use the scroll bar to search for the desired topic or type the word you are looking for in the field at the top. Then double-click the desired topic to view it in the topic pane.
- Click the **Search** tab and type the word that you are looking for. Then click the **List Topics** button to display the topics that contain the word. To view a specific topic, click the title and then click the **Display** button.

- Click the **Favorites** button to display the topics added to the Favorites list. To add the displayed topic to this list, right-click in the topic list and select **Add** from the pop-up menu.
- Click the **Glossary** button to display a list of glossary terms. The definitions are at the bottom of the pane.

2.6.3 Browse Sequence Bar



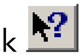
The browse sequence bar is displayed above the topic pane on the right. Each browse sequence contains topics related to a specific function.

- To display the topics in a browse sequence, click the arrow next to the drop-down list and select the desired browse sequence. The topics are displayed on the right side of the bar.
- To view a specific topic in the selected browse sequence, click the topic.
- To see all the topics in the browse sequence, click the Previous or Next arrows.

2.7 Context-Sensitive Help

Context-sensitive Help refers to information that you can display from a particular location in the software. There are two types of context-sensitive Help:

- Dialog Box Help, which displays the specific Help topic associated with the displayed dialog box or measurement window.
  - To view this help, click the **Help** button in the dialog box (if it has this button) or press the **F1** key while the dialog box or measurement window is displayed.
  - This also allows you to navigate to other topics within the Help file.

- **What's This? Help**, which displays a popup Help topic for a field, tab, button, etc. The popup topics provide definitions and usage tips for the fields in the dialog box.
  - To view this help, click  in the toolbar of the displayed window, or click  in the title bar of the dialog box or window, and then click a field to pop up a definition of that field.
  - Click anywhere on the screen to close the popup window.

2.8 How to Exit the Software

Before exiting the eBx software, be sure to complete the current measurement and save the routine.

To exit from the software:

1. Click the **X** in the upper right corner of the title bar or select **File / Exit**.
   - If a measurement has not been completed, the software displays a prompt to complete the measurement.
   - If the routine has not been saved, the software displays a prompt to save the changes to the routine.
   - Then the software displays a confirmation prompt to end the eBx session.
2. Click **OK**.

> If a control window is displayed over the toolbox or over a measurement results window, you cannot exit from the measurement software until you close the control window and complete the measurement, if applicable.

3. User Interface

3.1 Overview

This section describes the eBx user interface and explains how to use the following:

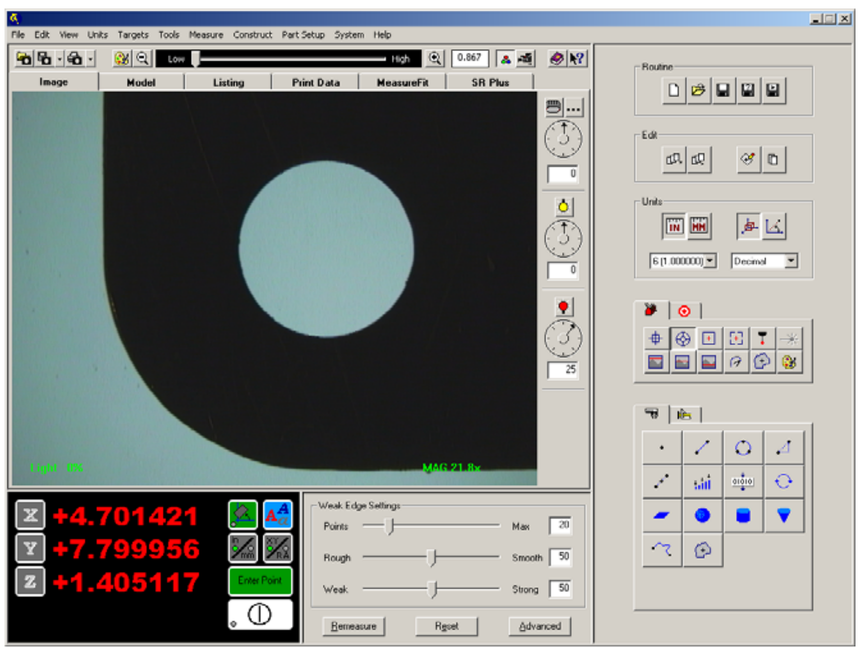
- Joystick
- Control panel
- Mouse

3.2 Screen Layouts

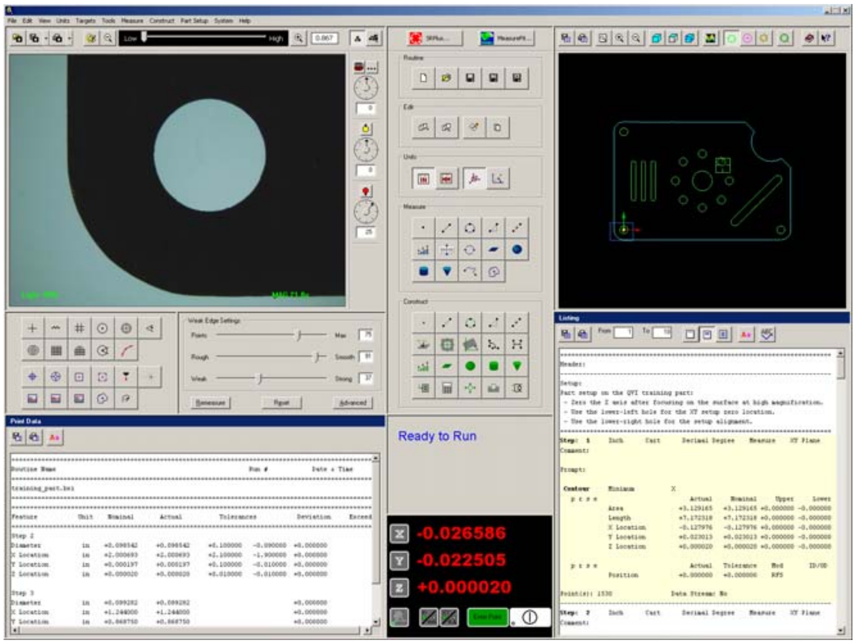
eBx supports three user interface types depending on how the software is configured and the type and resolution of the system monitor(s). The screen layout varies depending on how the user interface is configured.

> 🗎 Widescreen monitor support for the Single Monitor User Interface and Dual Monitor User Interface is only available in eBx version 3.05 or higher.

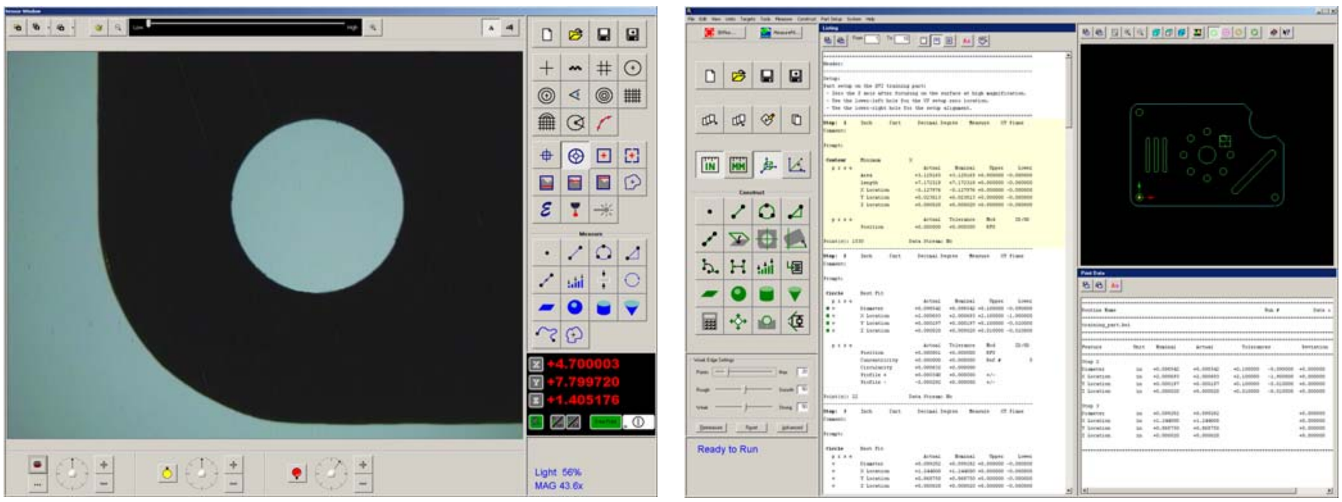
- **Standard User Interface**—traditional tabbed interface, already familiar to users who currently have version 1 or version 2 of the software. It is identical to the older versions, but has the new Enhanced Editor feature. This interface will only display on the screen at 1024 x 768 pixel resolution, regardless of the current screen resolution.



- Single Monitor User Interface (optional)— displays all the tools and windows at the same time on a single screen. This user interface requires a monitor capable of:
  - 1600 x 1200 pixel resolution [4:3 aspect ratio monitors]
  - 1680 x 1050 pixel resolution [16:10 aspect ratio monitors]
  - 1920 x 1080 pixel resolution [16:9 aspect ratio monitors]



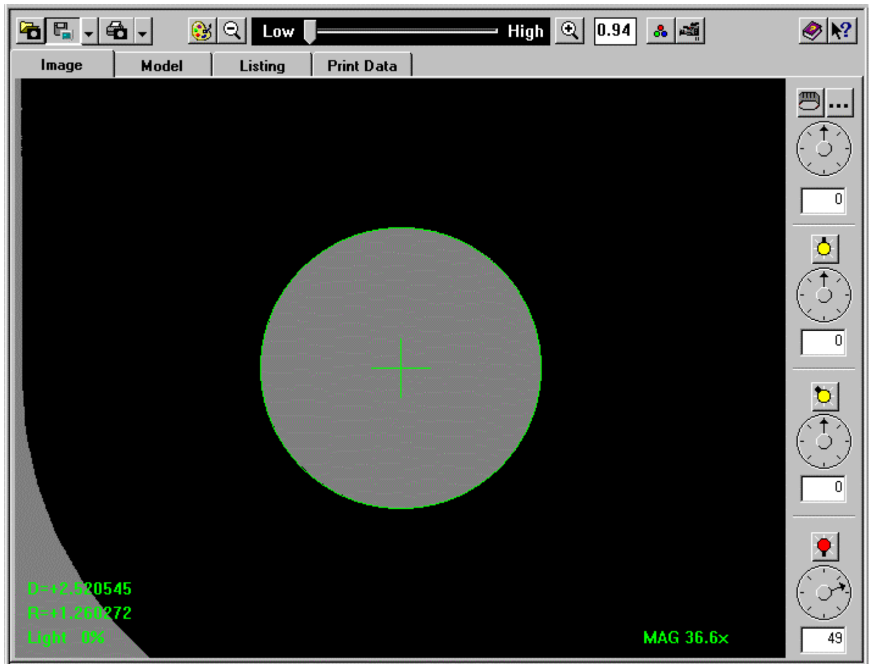
- **Dual Monitor User Interface** (optional)— separates the tools, controls, and windows onto two video monitors. One monitor displays the Image window, illumination controls, File icons, Manual Target icons, Automatic Tool icons, Measure icons, and DRO window. The second monitor displays the Model window, main menu, File icons, Edit icons, Unit icons, Construct icons, Tool & Target Settings window, Print Data window, and Listing window. Some windows can be resized and moved. This user interface requires two monitors capable of:
  - 1600 x 1200 pixel resolution [4:3 aspect ratio monitors]
  - 1680 x 1050 pixel resolution [16:10 aspect ratio monitors]
  - 1920 x 1080 pixel resolution [16:9 aspect ratio monitors]



> 🗎 The screen layout for the Dual Monitor User Interface varies slightly depending on the aspect ratio of the monitors used. For example, when using 4:3 aspect ratio monitors, the illumination controls appear below the Image window. However, when using 16:10 or 16:9 aspect ratio monitors the illumination controls appear on the right-hand side of the Image window.

3.3 Image Window

The Image window displays a live video image of the part on the worktable, under the optics.



Use the icons in the Image Window toolbar to open, save, and print the image, or create masks and overlays.

The image window also shows other information when appropriate:

- Pop-up dialog boxes, prompts and messages
- Light level (measured at the cursor position) in the lower left part of the window when the video image is displayed

Approximate magnification of the video image in the lower right part of the window when the video image is displayed

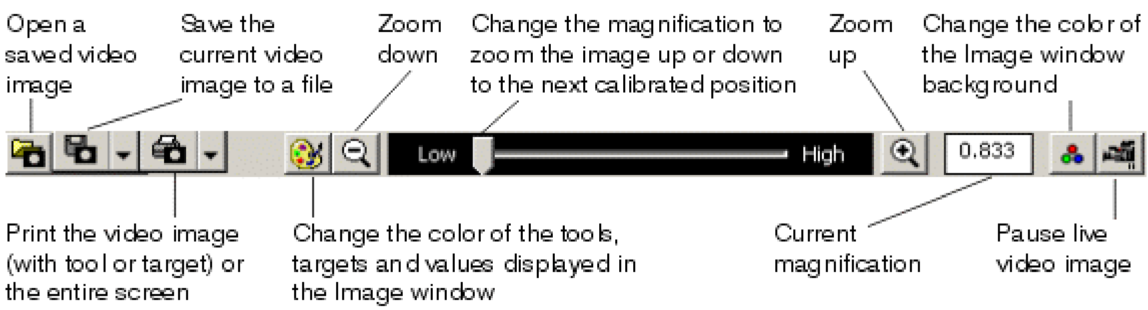
[Classic User Interface] The Image window has different functions, depending on which tabs you click or which menu selections you make:

- Displays the video image of the part being measured when you use the Measure function, when you click the **Image** tab or select **View** / **Image** / **Camera** from the main menu

- Displays a model of the part measurement when you use the Construct function, when you click the **Model** tab, or when you select **View** / **Model** / **Display** from the main menu
- Displays the current routine listing when you click the **Listing** tab or when you select **View** / **Listing** from the main menu
- Displays the print data output when you click the **Print Data** tab or when you select **View** / **Print Data** from the main menu

3.3.1 Image Window Toolbar

The Image window toolbar appears above the Image window. The toolbar contains icons to control Image window functions. If you do not know the function of an icon function, position the mouse cursor over the icon to display a tooltip for the icon.



You can control Image window functions in the following ways:

- Click the **Open Image File** icon to open a saved image of a part.
- Click the **Save Image File** icon to save the current video image to a file.
- Click the **Print Image** icon to print the video image or the entire screen.
- Click the **Color Palette** icon to change the color of the tools, targets and values displayed in the Image window.
- Change the magnification to zoom the image up to the next calibrated position.
- Click the **Color** icon (to the right of the magnification controls) to change the color of the Image window background (available only on certain systems).
- Click the **Video** icon to:

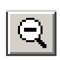
- Pause the live video image (the image from the camera does not go to the video buffer). For example, this occurs automatically when you create an overlay or mask of the video image or when you open a saved video image.
- Display the live video image (the image from the camera goes to the video buffer). This is the default setting.

3.3.2 Magnification Slider

The magnification slider appears in the Image window toolbar when an image is displayed.

The value displayed next to the magnification slider bar is an approximate magnification of the video image. It is based on the System / Configuration / Optics settings and the position of the zoom lens.

You can change the magnification in the following ways:

- Move the slider bar right to increase the magnification and move it left to decrease the magnification.
- Click  to zoom the image up to the next calibrated position.
- Click  to zoom the image down to the next calibrated position.
- Twist the joystick knob while pressing and holding the button on top of the joystick.

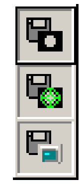
3.3.3 Mouse Functions in the Image Window

The functions for a 3-button mouse in the Image window are listed below.

Button	Function
Left	• Take points when using the Point Entry Target, Strong Edge and Weak Edge tools • Indicate the Start and End points and measurement direction when using the Edge Trace tool
Middle	• Hold the button down and drag the mouse to move the stage in X and Y • Hold the Ctrl key and button down and drag the mouse to move the stage in Z
Wheel	• Rotate the wheel to move the Z axis up and down (if enabled)
Right	• Hold the button down and drag the mouse to fine adjust the stage in X and Y

See the *Mouse* topic later in this section for information about controlling XYZ stage motion with the mouse.

3.3.4 Save Image Options

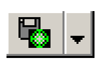


An image is typically saved as a reference for comparisons with other identical parts. For example, you can save an image of the master part or of a part with a known defect. You may also want to archive images of measured parts for future use.

You can save the video image displayed in the Image window in the ways described below. The image is saved in the Routines folder or another folder that you may have specified.

- Live video image. It can be saved in a bitmap (BMP), Tagged Image File (TIF), JPG, or 24-bit Targa (TGA) format. This is the default.
- Live video image and any displayed tool or target. It can be saved only in a bitmap (BMP) format.
- Full screen, including the image and any displayed tool or target. It can be saved only in a bitmap (BMP) format.

3.3.5 How to Save the Video Image



This function allows you to save the video image that is displayed in the Image window in a file.

To save an image:

1. Click next to the Save Image icon in the Image window toolbar and select the desired icon from the drop-down list. Each icon represents an option to save the video image.
2. Click the selected **Save Image** icon to save the image in the selected format. A window pops up displaying the image files in the Routines folder or in the folder that was used last.
3. Type a unique name for the image file. You can also change the folder if you want. Then click **OK** or press **Enter** on the keyboard.

3.3.6 How to Open a Saved Video Image

This function loads any video image that you saved earlier.

To load a saved video image:

1. Click  in the Image window toolbar. A window pops up displaying the saved image files in the folder that was used last.
2. Select the desired format (BMP, TIF, JPG, or TGA).
3. Specify the desired image file in one of the following ways:
   - Type the name of the file in the File Name field and click **Open** (or press Enter on the keyboard).
   - Scroll through the list until you find the file. Then highlight the file name and click **Open** (or press Enter on the keyboard), or double-click the file name.

The software loads the image into the video buffer. If the image is larger than the video buffer (i.e., it is a full screen), it is cropped to 640 x 480 pixels.

To clear the image from the video buffer and enable the live video image, click the **Video** icon in the Image window toolbar.

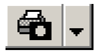
3.3.7 Print Image Options

This function allows you to print the image displayed in the Image window.

You can print the video image in the ways described below.

- Live video image (this is the default)
- Live video image and any displayed tool or target
- Full screen, including the image and any displayed tool or target

3.3.8 How to Print a Video Image



This function allows you to print the image displayed in the Image window to the printer that may be configured to the computer. The Windows print driver for the selected printer must be loaded before you can print the image.

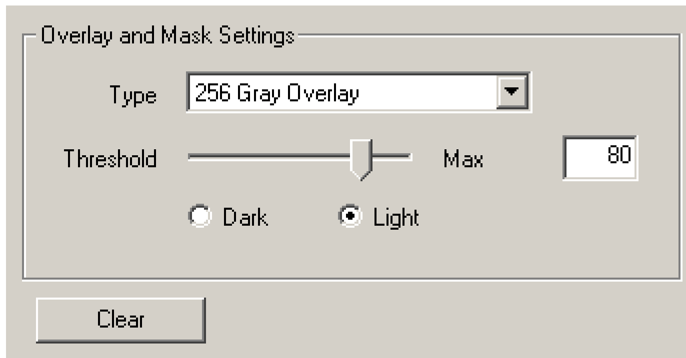
To print a video image:

1. Click next to the Print Image icon in the Image window toolbar and select the desired icon from the drop-down list. Each icon represents a different print image option.
2. Click the selected **Print Image** icon. The image is immediately sent to the printer.

- Images take time to print. You can continue working with the measurement software while the image is being printed but you cannot exit from the measurement software until the printing is completed.
- If you want to change the page size, orientation or destination, click Print Setup and make the desired changes before printing the image.
- Images print only in the number of colors in the video mode of the computer.
- If your system is equipped with a color camera and a video printer, you can print the video image by pressing the Print button on the printer.

3.3.9 How to Create an Overlay or Mask of the Video Image

1. Display the Overlay and Mask Settings window at the bottom of the screen in one of the following ways:
   - Right-click to display a list of windows that can be selected and use the left mouse button to select the window.
   - Select **View** / **Overlays and Masks** from the main menu.



2. Select the type of overlay or mask that you want to use from the drop-down list.
3. Select the desired radio button. These buttons are inactive for an edge mask.
   - If you select **Dark**, the overlay appears in the dark area.
   - If you select **Light**, the overlay appears in the light area.
4. Use the **Threshold slider** to set the light intensity threshold. The range is 0 to 100%. This slider is inactive for an edge mask.
   - If you select the Dark radio button, any area that is darker than the threshold is digitized.
   - If you select the Light radio button, any area that is brighter than the threshold is digitized.
   - The box next to the Threshold slider displays the percentage of the image that is dark or light. These percentages can also be used in the comparisons.
5. Click in the Image window to create the overlay or mask. An hourglass cursor appears until it is completed.

6. Drive the stage (e.g., using the fine-adjust knob (if equipped) or right mouse button) to view the live image behind the overlay or mask and to make the desired comparisons.

If you do not see the live video image, adjust the Threshold slider, clear the overlay or edge mask, and repeat Steps 5 and 6.

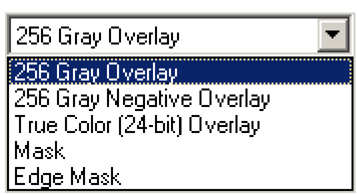
7. Click **Clear** to remove the current overlay or edge mask.

3.3.10 Overlays and Masks of the Video Image

An overlay or mask is a graphic representation of the video image. When you create an overlay or mask, the video image is digitized and "frozen" and becomes a snapshot.

You can select the type of overlay or mask that you want to create in the Overlay and Mask Settings window:

- **256 gray-scale overlay**. This creates an overlay using 256 shades of gray.
- **256 gray-scale negative overlay**. This inverts the gray-scale intensity of the overlay, e.g., it turns dark into light.
- **True color (24-bit) overlay**. This creates an overlay in 24-bit true color and displays it at the current screen display settings.
- **Mask**. This creates an overlay using the current tool or target color.
- **Edge mask**. This displays the mask as an outline anywhere that a sharp contrast exists between light and dark. The edge mask uses the currently selected tool or target color.



After creating an overlay or mask, you can view the live video image behind the overlay or mask for comparison purposes.

For the procedure to create an overlay or mask, see *How to Create an Overlay or Mask of the Video Image*.

3.3.11 Auxiliary Input

When you view the live video image in the Image window, the default setting is for the image to come through the camera. This is indicated by a check mark next to *Camera* in the *View / Image* menu.

If the video image comes from another source, e.g., video printer, you need to select **View** / **Image** / **Auxiliary** from the main menu.

For example, you must use the Auxiliary option if you have a video printer, because you cannot control the operation of the video printer using Print Image (see *Print Image Options*). In this case, select **Auxiliary** to view the on-screen menu of the printer. To print the live or stored image, press the Print button on the front panel of the printer.

If you selected Auxiliary in the View / Image menu, the software automatically selects Camera when you:

- Create or edit a step, or run the routine
- Click another tab in the Image window area
- Start a calibration procedure
- Select another tool or target

3.4 Illumination (Light) Sources

QVI video measuring systems may have the following illumination (light) sources:

 The **Backlight** provides light from below the part, through the stage glass, to create a contour or profile shadow image of the part. This is also known as profile lighting. Typical applications include profile images, edges and through-feature measurements.

 The **Ring Light** provides an oblique top light. A cone of light projected onto the surface creates a three-dimensional image that highlights heights, depths and surface imperfections. Typical applications include countersinks/bores, chamfers and rounded edges.

 The **Surface Light** provides direct "square-on" illumination onto the surface of the part where it is reflected back into the system optics. This is also known as coaxial or auxiliary illumination. Typical uses include surfaces, blind holes and Z axis autofocus measurements.

 The **VectorLight** enables you to control surface illumination more precisely by varying the intensity and the angle of incidence at which the light is projected.

Both the Ring Light and Surface Light are considered surface lighting. They are sometimes used together for optimum surface illumination.

You can control the three primary light sources and the light levels in the following ways:

- Dials in the Light Control Window
- Illumination knobs (if equipped) on the joystick
- Illumination knobs on the control panel (if equipped)

> If your system is equipped with a grid projector, you can also perform a Grid Focus using the illuminator.

3.4.1 Light Level

The light level displayed in the lower-left corner of the Image window shows the amount of light reaching the camera. The light level is measured around the mouse cursor location when the cursor is in the Image window.

The range is from 0% (dark - no light) to 100% (maximum brightness). At 100% the camera is fully saturated and cannot report any further increase in illumination.

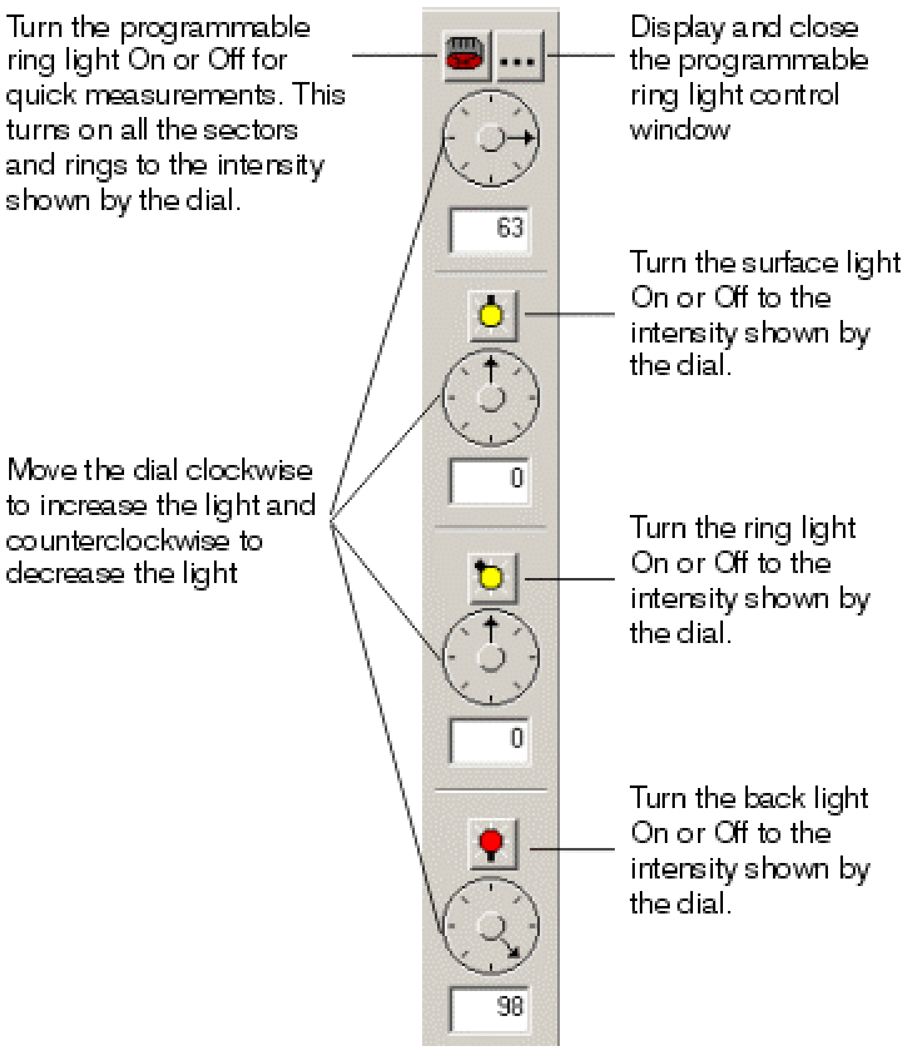
To adjust the light sources, turn the illumination knobs on the joystick (if equipped) or control panel (if equipped), or use the Light control window.

Tip When using any of the edge analysis tools, try to keep the light level at approximately 40 to 55%.

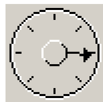
3.4.2 Light Control Window

The Light Control window allows you to:

- Turn the different light sources on to the intensity shown by the dial (see *How to Turn the Lights On*).
- Turn the different light sources off (see *How to Turn the Lights Off*).
- Access the programmable ring light illumination controls, which are displayed over the Measurement Results window or toolbox.



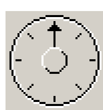
3.4.3 How to Turn the Lights On



- Position the mouse cursor on the dial arrow, press the left mouse button, and move the dial clockwise to increase the light and counterclockwise to decrease the light.
- Highlight the value in the light display box, type the desired value, and press **Enter** on the keyboard.
- Position the cursor at the desired location on the dial, press the left mouse button, and move the mouse cursor at least 1 pixel.
- Turn the illumination knob on the joystick (if equipped) or control panel (if equipped). This automatically moves the slider bar. However, moving the slider does not turn the illumination knob.

If a light is off (light icon is grayed out), click the desired light icon to turn it on to the current dial setting.

3.4.4 How to Turn the Lights Off



- Click the illumination button. This turns the light off but leaves the current dial setting.
- Position the cursor over the dial arrow, press the left mouse button, and move the dial all the way to the off position at the top.
- Position the cursor at the top of the dial, press the left mouse button, and move the mouse cursor at least 1 pixel.
- Turn the illumination knob on the joystick (if equipped) or control panel (if equipped) to Off. This automatically moves the dial to the off position.
- Highlight the value in the light display box, type **0**, and press **Enter** on the keyboard.

3.4.5 Lights Shutdown Mode

The lights shutdown mode turns off all the lights when the machine is not being used. This extends the life of the lights and prevents them from burning out prematurely.

The lights are shut off automatically after the machine has been idle for a specified amount of time.

- A fast blinking of the green status LED indicates that the lights are in shutdown mode.
- The automatic shutoff is controlled by the WATT_WATCHER_TIMEOUT parameter (see the Lights section in the Configuration Parameters Editor). The time is specified in seconds. The default value is 0, which means that this mode is disabled.

To get the machine out of the lights shutdown mode, you may:

- Press **Enter** on the joystick (recommended). However, this does not change the state of the machine.
- Change the zoom position with the zoom slider or joystick knob, or move the stage with the joystick.
- Change any of the light settings with the dials in the Light Control window.
- Start to run a routine.
- Change and save the value of the WATT_WATCHER_TIMEOUT parameter.
- Press **Start** / **Stop** on the joystick to put the machine in stop mode and then again to come back out of stop mode.

Moving the mouse does not get the machine out of the lights shutdown mode.

3.4.6 VectorLight Illumination

The VectorLight enables you to control surface illumination more precisely by varying the intensity and the angle of incidence at which the light is projected. It consists of six concentric rings of LED lights that provide different angles of incidence relative to the Z axis.

- The inner rings illuminate the part from above, almost "square-on."
- The outer rings provide more oblique lighting.

For example, the inner rings are used for illuminating surfaces that are flat or perpendicular to the optical axis. The outer rings are ideal for illuminating features like chamfers, countersinks, and blended radii.

The VectorLight is also divided into eight pie-shaped sectors that control the rotational direction of the light relative to the XY plane. Each sector spans 45 degrees. The sectors provide directional lighting so you can highlight just the area you need.

To display the VectorLight control window, click the VectorLight icons at the top of the Light Control window.

> The Ring Light knob on the control panel and the VectorLight illumination sliders share control of the light intensity of the two inner rings of the VectorLight. In this case, the VectorLight illumination sliders control the light intensity of the outer six rings and the Ring Light knob can be used to control the light intensity of the two inner rings only. For example, if the two inner rings are turned on to 50% with the Ring Light knob, and then you turn all the rings on to 50% with the VectorLight illumination sliders, the intensity of the two inner rings will be at the maximum of 100% (50% from the VectorLight slider and 50% from the Ring Light knob). If you then turn the Ring Light knob off, the VectorLight will be fully under the control of the VectorLight slider at 50%.

VectorLight Illumination Controls

In the Light Control window:

- Click to turn the VectorLight on or off for quick measurements. This turns on all the sectors and rings to the intensity shown by the dial. If the dial is at zero, you need to adjust the dial to turn the lights on.
- Click to display the VectorLight control window over the Measurement window or toolbox and to close the control window.

Tip If the VectorLight control window is displayed over a measurement that has not been completed, you cannot complete the measurement. Remember to close the VectorLight control window first and then complete the measurement.

VectorLight Control Window

To view the VectorLight control window, click in the Light Control window.

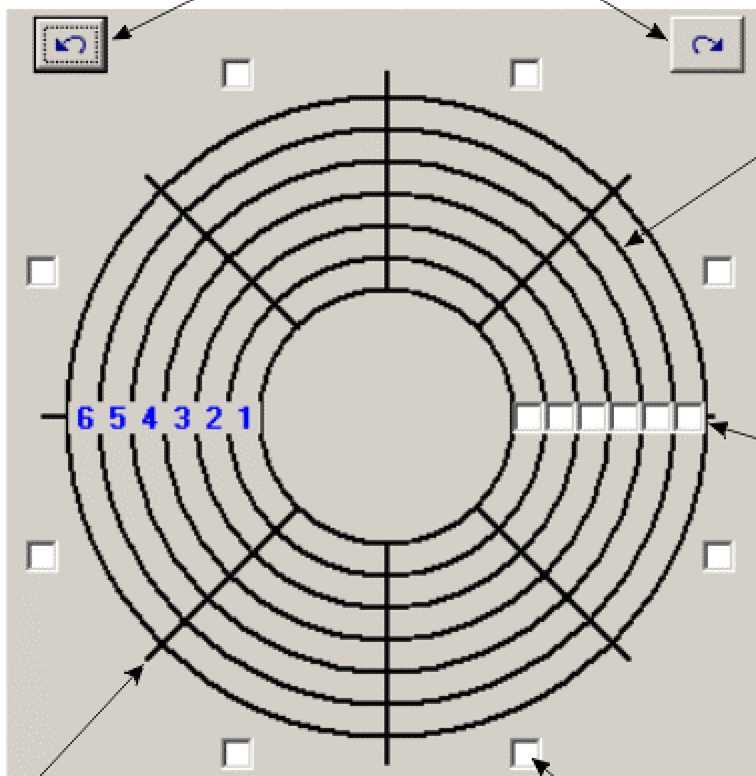
The control window consists of 6 concentric rings divided into 8 sectors. This enables you to control the illumination more closely by turning the individual ring segments on and off and by rotating the lights one sector at a time.

- The concentric rings control the angles of incidence at which the lights are projected relative to the Z axis. The number in each ring circle indicates the corresponding slider that controls the light intensity for that ring.
- The sectors identify the direction of the illumination relative to the XY axes. They are laid out counterclockwise starting with Sector 1, which is the 0 to 45 degree sector. Each sector contains six ring segments.
- The arrows in the two boxes at the top rotate the light settings in a sector clockwise or counterclockwise one sector at a time. For example, this is useful to keep the same illumination for round, chamfered parts.

To use the controls, see *How to Use the VectorLight Controls*.

To use the sliders and check boxes, see *VectorLight Controls*.

How to Use the VectorLight Controls

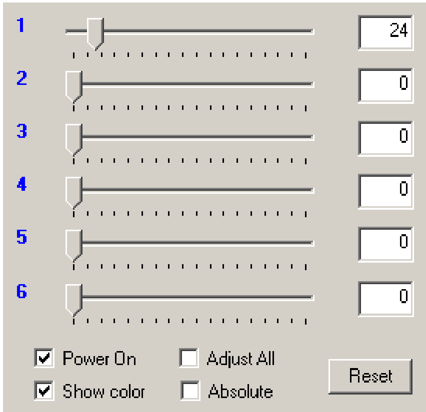


VectorLight Controls

The sliders in the VectorLight control window control the light intensity for each of the six rings.

Each slider controls the light intensity for any or all the segments in that ring. The same light intensity applies to any segments that are currently turned on, or that will be turned on during the creation or run of the current routine.

Light intensity values range from 0 (no light) to 255 (maximum light). When you first activate the software, the default light intensity values for all the lights are taken from the values specified in the HARDWARE.INI file.



You can also control the sliders and illumination in the following ways:

- Select the **Adjust All** check box to move all the sliders together. If any slider reaches the end, it stops while the others continue to move.
- Select the **Absolute** check box to reset the values of all the individual sliders to the value of the slider that you click and move. When you move that slider, all the other sliders move by the same amount.
- Select the **Show Color** check box to turn the camera color on or off.
- Select the **Power On** check box to turn the lights on or off.

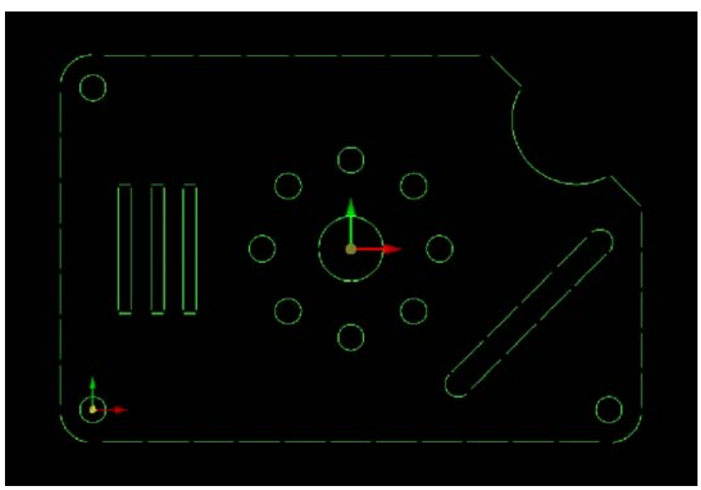
If you are measuring a feature, the default values are the values that were used the last time the graphic was up during the creation or current run of the routine. If you are editing a step or you are in step edit mode, the light intensity initializes to the values stored for the current point.

Tip When you finish using the VectorLight, remember to clear the Power On check box to turn the lights off. If you click the icon to close the VectorLight control window, this will remove the graphic but the lights will still stay on (assuming you selected the Power On check box).

3.5 Model Window

The Model window displays a CAD-like sketch of the measured and constructed features. A blue rectangle indicates the field of view. The model is always drawn to reflect the current datum system. The current datum is shown by the intersection of three lines indicating the XYZ axes.

[Classic User Interface] To display the Model window, click the **Model** tab or select **View** / **Model** / **Standard** from the main menu, or click  if the Surface window is displayed. The Model window is also displayed any time you use any of the Construct or Edit functions.



The Model window has two modes.

- Standard
- Show CAD (only available with the optional CAD Import functionality)

In Show CAD mode, the CAD model appears in the Model window and is overlaid by features that have been measured or constructed in the current program. When you Import a CAD file, the Model automatically switches to Show CAD mode.

To switch between the two modes, click . When the button is depressed, Show CAD mode is active. When the button is not depressed, Standard mode is active.

If you select the Display Windows check box in the routine run options, the Model window displays all the features when you run a routine. Features that have not been measured yet are displayed with a dark gray color.

If the SHOW_STAGE_LIMITS parameter in the Input.INI file has a value of 1 (On), the Model window displays a gray rectangle, which represents the stage travel. The rectangle is always in the same orientation as the features, i.e., the rectangle rotates along with the features.

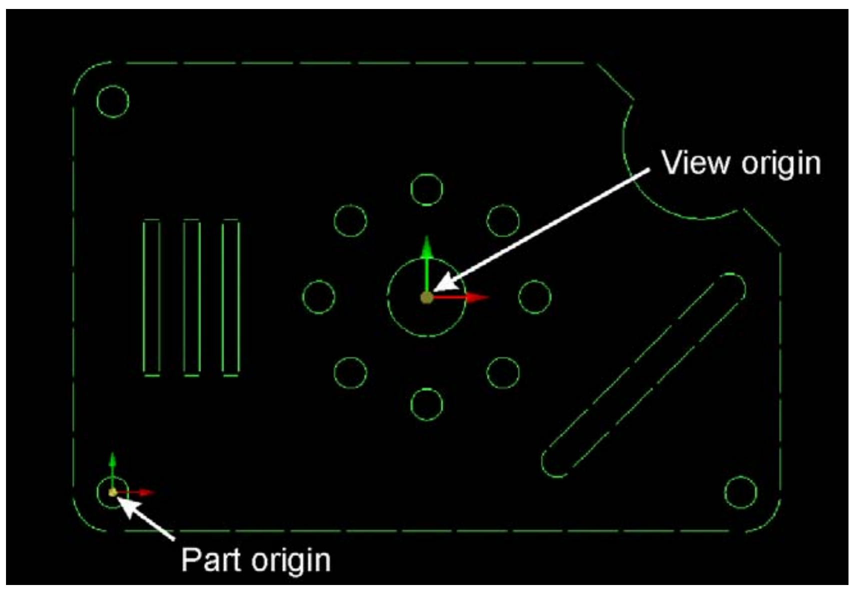
The system dynamically redraws the Model as the cursor is moved. For example, the system updates the Model in real time as you zoom, pan, or rotate the Model.

When constructing a feature, you can use a Box Capture to select reference features in the model window. During Construct, by holding down the Alt key on the keyboard, then pressing the left mouse button and dragging, a box will appear in the Model window. All features that fall within the box will be used as reference features for the constructed feature.

3.5.1 Coordinate System Axes

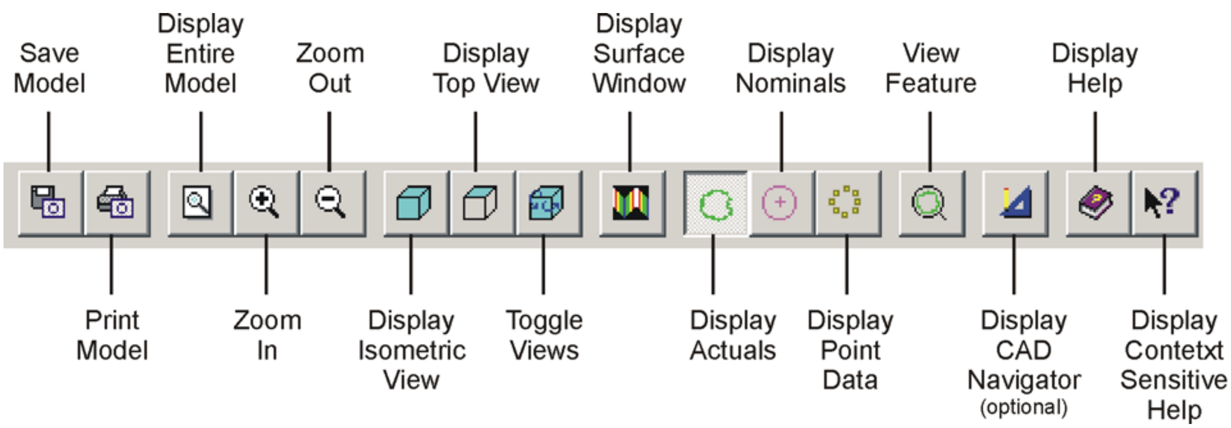
The Model window displays two sets of XYZ coordinate system axes.

- **Part origin**, shown as smaller, thinner lines. This indicates the XYZ origin of the part being measured and is always displayed with the part image. If the image is moved or rotated, the part origin moves and rotates with the image.
- **View origin**, shown as larger, thicker lines. This is a set of XYZ reference axes that always remain in the center of the Model window.
  - The view origin provides a constant visual reference for the location and orientation of the part in the Model window.
  - Whenever you select Zoom In or Zoom Out, the zoom is done about the view origin.
  - The view origin is displayed only when you click and hold one of the mouse buttons.



3.5.2 Model Window Toolbar

The Model window toolbar appears above the Model window. It includes icons that represent the most common functions in the Model menu. If you do not know the function of an icon, position the mouse cursor over the icon to display a tooltip for the icon.



You can control the model that is currently displayed, in the following ways:

- Save the model in an image file.
- Print the model.
- Use the three zoom control icons to control the size of the model.
- Use the **Isometric View** and **Top View** icons to display the appropriate views.
- Use the **Next View** icon to toggle through six views in the following order: Top, Front, Right, Back, Left, and Bottom. When you click the button, the tooltip indicates which view will be displayed next.
- Display the Surface window for the selected feature.
- Use the **Actuals**, **Nominals** and **Points** icons to control the display of the applicable data.

- Use the View Feature icon to maximize the feature currently being edited in the Model window. To view a feature, select the feature in the Model window and click the View Feature icon.
  - The top view of the *feature* (not the entire part) appears in the Model window. The feature step is displayed in the Measurement window.
  - This function is useful when you want to view the actual data and compare them to the nominal data.
- Use the Show CAD icon to change the mode of the Model window. This allows you to display the CAD model under measured and constructed features when using the CAD Navigator. This icon only appears when the system is configured to display Single Monitor User Interface or Dual Monitor User Interface, if the CAD Import option is enabled.

Refer to Section 18 for more information about using the optional CAD Navigator.

3.5.3 How to Save the Model

This function allows you to save the displayed model as an image in bitmap (BMP) or Jpeg (JPG) format. It can be saved in the Part Routines folder or another folder that you may have specified.

To save the model in an image file:

1. Click  in the Model window toolbar. A window pops up displaying the image files in the folder that was used last.
2. Type a unique name for the file. You can also navigate to a different folder if you want. Then click **OK** or press **Enter** on the keyboard.

3.5.4 How to Print the Model

This function allows you to print the displayed model to the data printer that may be configured to the computer. The Windows print driver for the selected printer must be loaded before you can print the image.

The printed images are typically based on an 8 1/2 x 11 format.

To print the model, click  in the Model window toolbar.

The software immediately sends the model to the currently selected printer, which can be local or on a network. If you have not selected a printer, the software displays the standard Print Setup window to select the printer.

- You can continue working with the software while the model is being printed but you cannot exit from the software until the printing is completed.

- If you want to change the page size, orientation or destination, click Print Setup and make the desired changes before printing the image.

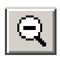
3.5.5 Zoom Control Modes

When you open a routine, the model is dynamically reduced or enlarged, in scale, to fit entirely within the Model window.

You can change the size of the entire model (change the scale) in these ways:

 Click the **Zoom All** (fit to window) icon in the Model window toolbar or select **All** in the Model menu to display the entire model in the Model window.

 Click the **Zoom In** icon in the Model window toolbar or select **Zoom In** in the Model menu to increase the size of the model.

 Click the **Zoom Out** icon in the Model window toolbar or select **Zoom Out** in the Model menu to reduce the size of the model.

 Select a feature in the Model window and click the **View Feature** icon to maximize the *feature* in the Model window.

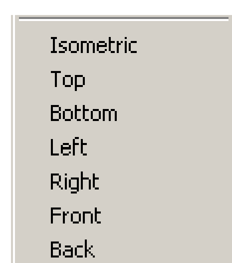
To zoom in on a specific area, do the following:

1. Press and hold the Shift key on the keyboard.
2. Press and hold the left mouse button.
3. Drag the mouse diagonally to draw a box around the desired area in the Model window.

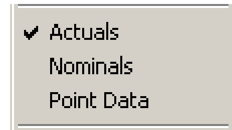
> Changing the size of the model changes its scale for easier viewing. This does not change the actual dimensions of the part and features.

3.5.6 Views in the Model Window

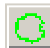
You can display different views in the Model window. To display a specific view, click the appropriate view icon in the Model window toolbar or select the desired view in the Model menu. The default is Top View.

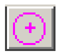


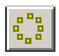
3.5.7 Graphical Representation of Displayed Data



The following data can be displayed in the Model window by selecting the data type in the Model menu or by clicking the applicable icon in the Model window toolbar.

 Actual features. This displays the actual features (default).

 Nominal features. This displays the nominal features. These can be displayed only if nominal and tolerance values were specified in the feature measurement.

 Point data. These are all the raw points in multiple-point features.

You can only select actual features in the Model window. You cannot select nominal features or data points.

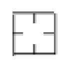
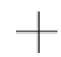
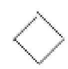
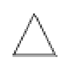
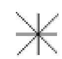
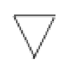
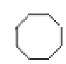
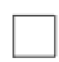
See *Colors of Model Window Attributes* for a description of the colors used for the data displayed in the Model window.

3.5.8 Features in the Model Window

Features are displayed in the following ways in the Model window:

- Measured features are displayed as solid lines.
- Constructed features are displayed as dashed lines.
- Lines are drawn along the entered points.
- Circles are displayed in two ways:
  - As a full circle, if the diameter is output.
  - As an arc along the entered points, if the radius is output.
- An intersection between two lines displays a cross at the intersection location and joins the lines to create the corner formed by the intersection. When circles and lines are intersected, they are joined in a similar way.

- Contour features are displayed as a continuous line or curve with all the data points connected. The nominal contour location is displayed as an octagon.
- Spheres are displayed as five circles. One circle is near the top and one near the bottom, and the other three circles connect the top and bottom circles.
- Planes are displayed as a rectangle enclosing all the points of the plane.
- If the touch probe is enabled, the current location of the probe is shown with a tip (solid red circle) and stylus. The tip and stylus move as you move the probe.
- All measured points are displayed differently based on the tool or target used to measure them.

Tool	Symbol	Tool/Target	Symbol
Strong edge		All manual targets	
Average weak edge		Constructed point	
Maximum weak edge		Touch probe point	
Minimum weak edge		Centroid	
Focus			

3.5.9 How to Select Features in the Model Window

You can view and select any feature that appears in the Model window. For example, you can select features for the following kinds of functions:

- Include it in the current construction step
- Automatically enter its step number into an edit step field
- Automatically edit the feature if no other feature is currently being measured, constructed, or edited
- Include one of the feature's dimensions in a math or branch step expression window

To select a specific feature for editing, position the cursor over the feature and press the left mouse button. The color of the feature changes to indicate that it was selected.

- When you select a constructed feature, the Construct window appears on the right showing the choices and calculations used to define the feature.
- When you select a measured feature, the Measurement window appears on the right showing the measurement results and the Image window reappears showing the actual feature. The stage moves to the feature location if the MODEL_MOVE_STAGE parameter in the HARDWARE.INI file is turned on.
- When editing a feature, you must click **Cancel** or **OK** in the Construct or Measurement window before you can view another feature. If the Image window is displayed, click the Model tab to display the model.
- When you select a feature, the software displays the feature type and its step number in a pop-up menu if the MODEL_1_FEATURE_DROP_DOWN parameter is set to 1.

You can select more than one feature when you use an appropriate Construct or Edit function. For example, to construct a width, select the first feature and then select the next feature. The results then appear in the Construct window.

If any features are close to each other, you can select the desired feature as follows:

1. Place the cursor on the desired feature and click the left mouse button once. The software displays the feature types and their step numbers in a dynamic list.
2. Place the cursor on the step number of the desired feature and click the left mouse button to select it.

If the current step is a math or branch step, the system displays dimensions for the selected feature in a dynamic menu. Click the desired dimension to insert it into the expression window.

> To select a constructed contour, click the features rather than the contour itself.

If the touch probe is deployed when you select a feature that was measured optically, the software displays a prompt to dock the probe. Click **Yes** to dock the probe and then edit the feature. Click **No** if you do not want to dock the probe. For example, this allows you to change nominal and tolerance values while the probe is deployed. In this case, be very careful about moving the stage or optics.

3.5.10 Colors of Model Window Attributes

The Model window has a black background. This enables the other colors to be displayed more vividly. However, if you print the model (see *How to Print the Model*), the system reverses the black and white colors to save printer ink.

Coordinate system axes

X: (red) Y: (green) Z: (yellow)

(green) Actual feature - in tolerance or not toleranced

(yellow) Actual feature - within the tolerance band specified in the YELLOW_PERCENT parameter in the INPUT.INI file

(red) Actual feature - out of tolerance:

(light cyan) Current feature

(cyan) Features not measured yet

(light magenta) Selected features

(magenta) Reference features of selected features

(light blue) Nominal dimensions for all features (only if you entered nominal values for all feature and size dimensions)

Data points of a measured feature

(gray) The rectangle representing the stage travel

Touch probe attributes

(green) Location of tip

(yellow) Point not entered yet

(red) Missed point

The system displays the actual features by default. You need to click the Nominals or Data Points icons in the Model window toolbar to display these features.

3.5.11 Mouse Functions in the Model Window

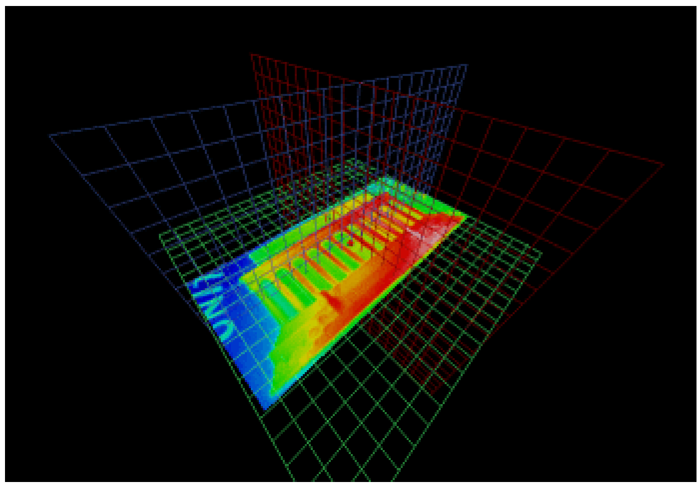
The functions for a 3-button mouse in the Model window are listed below.

Button	Function
Left	• Click to select features or icons • Hold the button down and drag the mouse up to zoom the entire model up • Hold the button down and drag the mouse down to zoom the entire model down • Zoom in on a specific area by holding the button and Shift key down and dragging the cursor diagonally to draw a box around the desired area
Middle	• Pan (move) the entire model in any direction by holding the button down and moving the cursor in the desired direction
Right	• Rotate the entire model around the view origin by holding the button down and moving the cursor: – Left/right to rotate the model around the Z axis – Up or down to rotate the model about the horizontal screen axis

3.6 Surface Window

To view the Surface window, click  in the Model window toolbar or select **View** / **Model** / **Surface View** from the main menu.

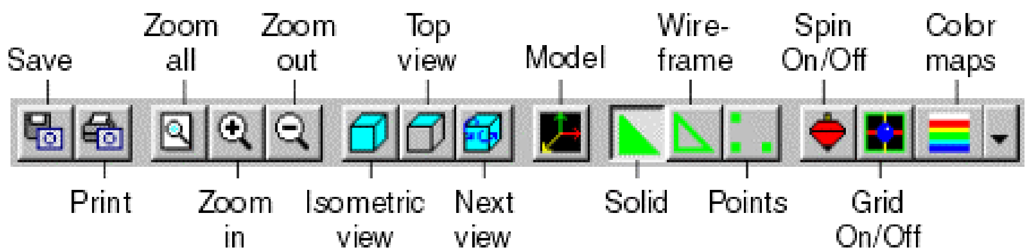
The Surface window displays a topographical view of the surface of a feature measured with a laser area scan. After measuring the feature, you must select it in the Model window by double-clicking it before it can be displayed in the Surface window.



See the *Surface Window Toolbar* topic for descriptions on how to display different views and rotate the surface in the Surface window.

3.6.1 Surface Window Toolbar

The Surface window toolbar appears above the Surface window. It includes icons to help you manipulate the surface in the window. If you do not know the function of an icon, position the mouse cursor over the icon to display a tooltip for the icon.



You can control the surface that is currently displayed, in the following ways:

- Save the image in an image file.
- Print the image.
- Use the three zoom control icons to control the Zoom All (fit to window), Zoom In, and Zoom Out functions.
- Use the **Isometric View, Top View** (default), **Front View**, and **Next View** icons to display the appropriate views. If the Grid is turned on, the grid colors indicate the orientation of the grid.
- Click the **Model** icon to close the Surface window and display the Model window.
- Three icons display different aspects of the surface:
  - Solid, shaded view
  - Wireframe view
  - All the point data
- Toggle the **Auto Spin On/Off**. This controls the continuous "spin" of the surface.
- Toggle the **Grid Axes On/Off**. This is useful in viewing the orientation as the surface is rotated.
- Display and change the colors of the surface window attributes.

Tip When looking at the surface in Wireframe or Point mode, turn the grid projector off to view the surface more easily.

3.6.2 Colors of Surface Window Attributes

The Surface window has a black background. This enables the other colors to be displayed more vividly. However, if you print the surface (see *How to Print the Model*), the system reverses the black and white colors to save printer ink.

You can select different color maps for displaying the surface of the measured feature:

- Rainbow (default)
- Grayscale
- Blue/Yellow/Red

To select a color map, you can either:

- Select the desired color map in the **View** / **Model** / **Surface View** / **Color Map** submenu.
- Click the **arrow** next to the color map icon in the Surface window toolbar. This displays a drop-down list with color map icons. Then click an icon to select the desired color map for the surface.

If you want to display a legend of the selected color map, click the color map icon again.

When the grid is turned on, its colors are the same as the coordinate system axes:

- XZ Plane: (red)
- XY Plane: (green)
- YZ Plane: (yellow)

3.6.3 Surface Color Map Legend

The color map legend contains a range of colors, which are used to produce a smooth gradient from the lowest points to the highest points on the surface.

For example, the legend of the Rainbow color map contains the following range of colors:

- White: highest points
- Red
- Yellow
- Green
- Blue-green
- Dark Blue: lowest points

To display a legend of the selected color map in the Surface window, you can do either of the following:

- Select **View** / **Model** / **Legend** from the main menu.
- Click the displayed color map icon in the Surface window toolbar. You can click the icon to turn the legend off (default).

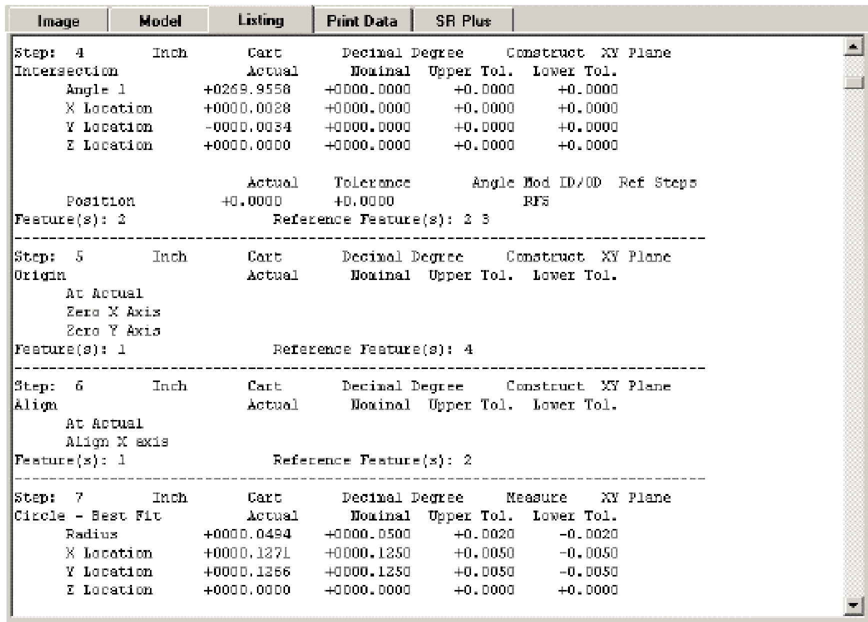
3.6.4 Mouse Functions in the Surface Window

The functions for a 3-button mouse in the Surface window are listed below.

Button	Function
Left	• Hold the button down and drag the mouse up to zoom the entire model up • Hold the button down and drag the mouse down to zoom the entire model down • Hold the button down and drag the mouse left for a Counter-Clockwise motion (roll) • Hold the button down and drag the mouse right for a Clockwise motion (roll)
Middle	• Hold the button down and drag the mouse to move the stage in X and Y • Hold the Ctrl key and button down and drag the mouse to move the stage in Z
Wheel	• Rotate the wheel to move the Z axis up and down (if enabled)
Right	• Rotate the entire surface around the view origin by holding the button down and moving the cursor: – Left/right to rotate the surface left or right (yaw) – Up or down to rotate the surface up or down (pitch)

3.7 Routine Listing

Click the **Listing** tab or select **View** / **Listing** from the main menu to display the current listing of the routine. For example, the listing shown here is in a standard format, which lists the details of the measurements in each step.




```
Step:  4       Inch       Cart       Decimal Degree     Construct  XY Plane
Intersection              Actual      Nominal  Upper Tol.  Lower Tol.
      Angle 1        +0269.9558    +0000.0000     +0.0000     +0.0000
      X Location     +0000.0028    +0000.0000     +0.0000     +0.0000
      Y Location     -0000.0034    +0000.0000     +0.0000     +0.0000
      Z Location     +0000.0000    +0000.0000     +0.0000     +0.0000

                          Actual    Tolerance       Angle Mod ID/OD  Ref Steps
      Position         +0.0000      +0.0000                  RFS
Feature(s): 2                Reference Feature(s): 2 3
--------------------------------------------------------------------------------
Step:  5       Inch       Cart       Decimal Degree     Construct  XY Plane
Origin                    Actual      Nominal  Upper Tol.  Lower Tol.
      At Actual
      Zero X Axis
      Zero Y Axis
Feature(s): 1                Reference Feature(s): 4
--------------------------------------------------------------------------------
Step:  6       Inch       Cart       Decimal Degree     Construct  XY Plane
Align                     Actual      Nominal  Upper Tol.  Lower Tol.
      At Actual
      Align X axis
Feature(s): 1                Reference Feature(s): 2
--------------------------------------------------------------------------------
Step:  7       Inch       Cart       Decimal Degree     Measure    XY Plane
Circle - Best Fit         Actual      Nominal  Upper Tol.  Lower Tol.
      Radius         +0000.0494    +0000.0500     +0.0020     -0.0020
      X Location     +0000.1271    +0000.1250     +0.0050     -0.0050
      Y Location     +0000.1266    +0000.1250     +0.0050     -0.0050
      Z Location     +0000.0000    +0000.0000     +0.0000     +0.0000
```

3.7.1 Routine Listing Toolbar

When the routine listing and its toolbar are displayed, you can control the listing in the following ways:

- Select the steps to be displayed and output in the **From/To** boxes.
- Output the listing to a text file or the printer by clicking the appropriate icon.
- Select the format to be displayed and output by clicking the appropriate icon: 
  - **Condensed**. For each step, this lists the feature, XYZ (or RAZ) location and size. If the measurements in a step are constructed from previous measurements, the step number(s) are listed in the Reference column.
  - **Standard**. This lists the details of the measurements in each step. It includes the units of measurement, features measured, actual measurements, and nominal values and tolerances.
  - **Expanded**. This lists the standard information with details about each point in the measurement. The details include location, zoom, lighting, and tool or target used. For manual targets and point tools (strong edge point, focus, touch probe, laser), this also lists the programmed locations of the points that were used to measure the feature.
- Display a dialog box where you can select the desired font, font style, color, etc., by clicking the **Font** icon.

3.7.2 How to Output a Routine Listing

To output the routine listing and the name of the routine to a printer or file:

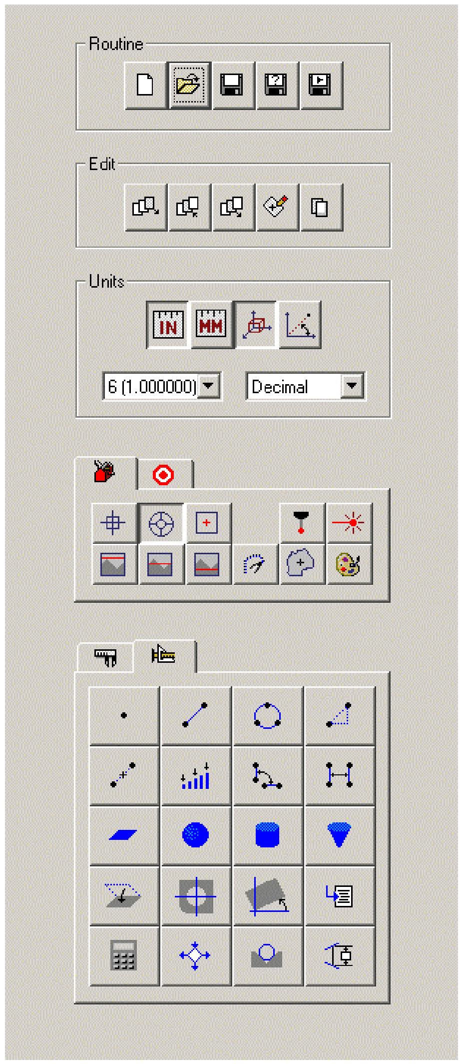
1. Save the routine if you want to include the name of the routine at the top of the printed report.
2. Click the appropriate icon to select the desired format (default is Standard).
3. Specify the steps that you want to print in the **From**/**To** boxes (default is all steps).
4. Click the **Print Listing**  icon or the **Save Listing As**  icon.
   - If the listing is printed, the software immediately sends the selected steps to the currently selected printer, which can be local or on a network. If you have not selected a printer, the software displays the standard Print Setup window to select the printer.
   - If the listing is output to a file, a window pops up where you can specify the name of the file and where it will be located. The default file extension is TXT. The software overwrites the data in this file each time you output data to this file.

The actual values in a routine listing are current only as of the last run of the routine. When you load a routine, the listing contains the recalculated dimensions of each feature.

If the listing in one of these windows contains too many lines, the system displays a message indicating that the routine is too long. The entire listing cannot be displayed on-line; only a partial listing can be displayed. To see the entire listing you need to print it or send it to a file and view the file.

3.8 Toolbox

The toolbox appears on the right side of the screen. It shows icons in related functional groups that correspond to the functions in the File, Edit, Units, Tools, Targets, Measure, and Construct menus. You can use the icons instead of selecting them from the menus.



If you are not sure what a particular icon represents, position the mouse cursor over the desired tab or icon (do not click it). This displays the name of the function below the icon.

To select a function, position the mouse cursor over the desired icon and press the left mouse button. This activates the selected function.

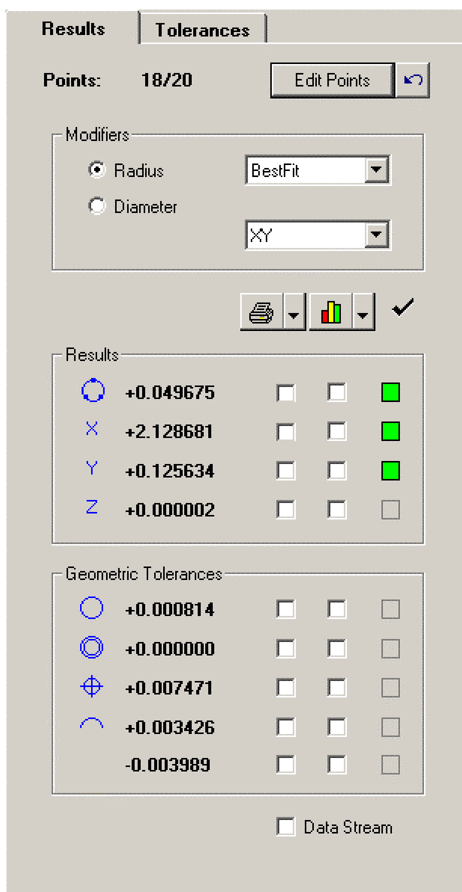
To select an edge analysis or focus tool or a manual alignment target, click the tab first to display the applicable icons.

A "pushed in" look indicates the default and selected (active) functions. For example, the default units of measurement are inches and Cartesian coordinates.

For Measure (the Measure tab is “pushed in”), only the Measure functions are enabled; the invalid functions are not displayed. For example, Width and Intersection are construct functions and they are displayed only when the Construct tab is pushed in.

3.9 Measurement Window

The Measurement window appears on the right side of the screen. Initially this window displays the toolbox.



This window becomes active with different options when you make selections in the Edit, Measure, Construct, and System menus or when you click the equivalent icons in the toolbox.

For example, this window displays measurement results after you measure or construct a feature.

It also displays the VectorLight controls when you click the VectorLight icon in the Light Control window.

Measurement Window Buttons

Use the buttons at the bottom of the Measurement window to perform an action in the window.

- Click OK to accept the measurement results and any entered data such as nominal and tolerance values. This saves the measurement step in the current routine and closes the Measurement window.
- Click Cancel to close the Measurement window without saving the measured feature.
- Click Again to measure or construct the same type of feature again. This saves the current measurement and clears all the displayed values from the Measurement window.
- Click Global Edit... to use the Global Editor to change multiple steps at the same time. This button appears in the same location as the Again button when you are editing/changing a step.
- Click Search to use the Search function, which is available only in Measure steps.
- If you click Clear:
    - When the Results window is displayed, this removes all the check marks for all of the output destinations.

- When the Tolerances window is displayed, this sets all the nominal and tolerance values to zero. If you display the Results window and then display the Tolerances window again before closing the step, the software includes the actual values in the nominal fields and any tolerances that have been carried forward from a previous similar step.
- When the Branch window is displayed, this sets both Go To Step values to 0.
- When the datum windows are displayed, this resets the controls to their default settings.

• Click Prompt / Text... to display a dialog box to enter text and a prompt for the feature.

3.10 DRO Window

This window, in the lower part of the screen, shows the following information:

- Current location of the centerline intersection and stage in the Image window. The location is reported in the X, Y and Z axes (or R, A and Z axes if polar coordinates are selected).
- Whether the axis alignment is activated (indicated by a green light)
- Current units of measurement and coordinate system (indicated by a green light)
- Whether the system is in Stop mode (indicated by a yellow light). The stage, Z-axis slide, focus, and zoom lens cannot be moved as long as the light is on.

You can do the following in the DRO window by positioning the cursor over the desired button and pressing the left mouse button:

- Click the **X**, **Y**, and **Z** buttons to zero out the X, Y, and Z axes. You can do this only when starting a new routine or before running a routine, when the run confirmation window is displayed.
- Click the **Axis Alignment** button to set the horizontal alignment of the X axis.
- Click the **in/mm** button to change the linear units of measurement.
- Click the **XY/RA** button to change the coordinate system.
- Click the **Stop** / **Start** button to cut power to all motors, stop the stage, and cancel the current measurement. If the system is in Stop mode, press this button to resume operation again.
- Click the **Enter** button to take a measurement point. The Enter button in the DRO window may be enabled or disabled in the System Configuration Editor, EMX General section.
- If the system is configured to display the Classic User Interface, you can click the blue **A** button to change the DRO display font and color. In all modes, you can change the DRO display font and color by selecting **View** / **DRO Font** from the main menu.

You can use the Units and Coordinates function to change the significant digits that are displayed.

Enter Point Button

To enter manual points on systems without an Enter button, or as an alternative on those systems that do have an Enter button, the Enter Point button is available as an option. It appears in the DRO window.

To enable the Enter Point button, select **System** / **Configuration** / **Editor** from the main menu and go to the **EMX General** section. Change the EMX_DROEnterPoint parameter to **1**. After you save the parameter change and exit the Configuration Editor, the Enter Point button appears in the DRO window.

3.11 Status Window

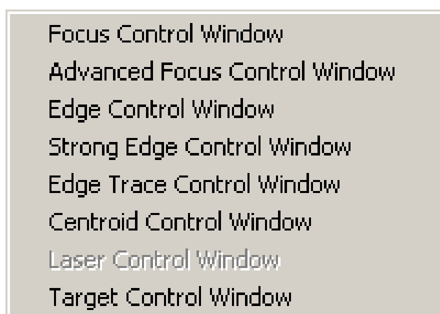
The Status window shows you the current operational status of the eBx software.

3.12 Tool Setting Window Display Options

When you select a tool or target, the software displays its settings window next to the DRO window.

To display a different tool or target control window:

1. Right-click in the Tool and Target Settings window to display a list of windows that can be selected.



2. Click the desired settings window to display it.

You can also display different settings windows by selecting the desired window in the View menu. A check mark indicates the currently displayed settings window.

3.13 Title Bar

The title bar displays the name of the current routine on the left, and minimize, maximize, and close buttons on the far right.

Click the Minimize button to minimize the eBx window when it is not in use. It is then displayed in the Windows task bar. This does not exit you from the eBx software. Click the button in the task bar to redisplay the full eBx window.

Click the **X** in the upper right corner to exit from the eBx software.

It is strongly recommended that you do not click the Minimize button while the routine is running. This may cause unpredictable results.

3.14 Menus

eBx has pull-down menus for its primary functions. Each function contains additional choices that appear as menu items under or next to the function. The most common menu items are displayed as icons in the toolbox.

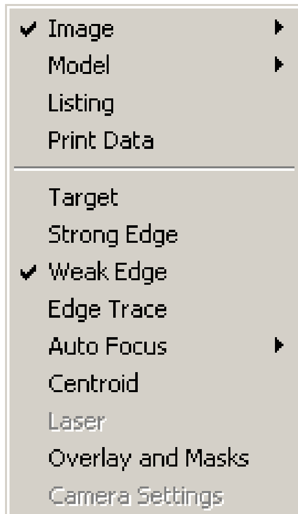
Use the mouse to select a function. When you select a function it is highlighted and becomes active. A second level of menu items also appears. For example, Weak Edge is selected in the Tools menu.

If the menu item contains additional choices, an arrow follows the function title. When you select the menu item, a submenu appears. For example, when you select Weak Edge, three weak edge point choices and FeatureFinder appear in a submenu.

If you select a menu item showing three dots, the software displays a dialog box or a measurement window. In a dialog box you typically need to make appropriate choices and then confirm them.

3.14.1 View Menu

Select **View** from the main menu to display the View menu. You can then select what is displayed in the Image window area and in the tool/target control window.



To select what is displayed in the Image window area:

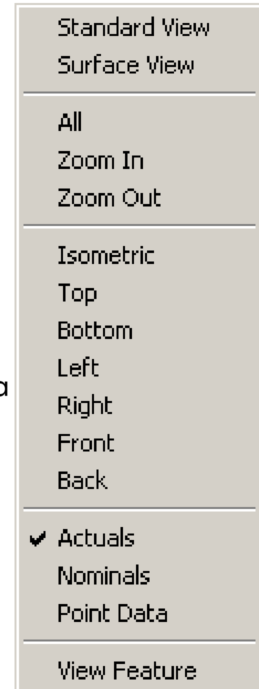
- Select **Image** or click the **Image** tab to display the actual video image in the Image window. This is the default when you start the system.
- Select **Model** / **Display** or click the **Model** tab (Classic User Interface) to display the Model window, which displays a CAD-like sketch of the part.
- Select **Listing** or click the **Listing** tab (Classic User Interface) to display the current listing of the routine.
- Select **Print Data** or click the **Print Data** tab (Classic User Interface) to display the dimensions that were output to a printer in a measurement step.

To select the settings to be displayed in the tool/target settings window:

- Select **Target** to display the target settings.
- Select **Strong Edge** to display the strong edge settings.
- Select **Weak Edge** to display the weak edge settings.
- Select **Edge Trace** to display the edge trace settings.
- Select **Focus** to display the autofocus settings.
- Select **Laser** to display the laser settings. This is enabled only if your system is equipped with a laser.
- Select **Overlay and Masks** to display the overlay and mask settings.

3.14.2 Model Menu

Select **View** / **Model** from the main menu to display the Model menu. This also displays the Model window.



The Model menu is grouped into the following functional categories:

- Zoom control modes
- Views in the Model window
- Graphical representation of displayed data
- View the selected feature

3.15 Joystick

Use the joystick (if equipped) to control:

- Stage motion, i.e., move the stage along the X and Y axes. The image on the screen moves in the direction in which you move the joystick lever.
  - Move the lever to the left or right for the X axis.
  - Move the lever up or down for the Y axis.
- Speed of the stage motion. The stage moves faster as you move the lever farther from the center.

Use the knobs to control:

- Z-axis slide and manual focus. Twist the knob at the top of the joystick to move the Z-axis slide and manually change the focus and adjust the sharpness of the image.
    - Twist the knob clockwise to move the Z axis up.
    - Twist the knob counterclockwise to move the Z axis down.
- Magnification. If the system is equipped with a zoom lens, you can control the magnification by twisting the knob while pressing and holding the button on top of the joystick. The magnification zooms through its full range.
    - Twist the knob clockwise to increase the magnification.
    - Twist the knob counterclockwise to decrease the magnification.
    - You can also do this using the magnification slider bar.
- Illumination. If your joystick is equipped with illumination knobs, turn the desired illumination knobs to control three light sources and the light level. You can also control the illumination in the Light Control window.

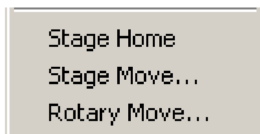
Use the buttons for the following controls:

- Press the **Enter** and **Cancel** (if equipped) buttons to accept or delete measured points, for example, when measuring features with the Focus tool, Strong Edge tool, or the alignment targets. Pressing the Enter button is equivalent to clicking the OK button in the software.
- If your joystick is equipped with a **Stop** / **Start** button, press it to:
    - Stop the system and cut power to all motors (Stop mode)
    - Continue the system power-up procedure
    - Take the system out of Stop mode and resume operation
    - Delete a measured point

3.15.1 Stage Move Function

This function enables you to specify the X, Y, and/or Z coordinates of a destination and move the stage to the specified location. You can use this method to move the stage whenever the machine is idle. For example, if you have a floor-model machine with a large stage, you can move to a specified location more quickly by using this function instead of the joystick.

- You may move the stage both when the Image and Model windows are displayed.
- You may move the stage when a step is open or when there is no step in the Measurement window.



To access the Stage Move function, select **System** / **Stage Move** from the main menu. You may also click the right mouse button with the cursor in the DRO window, or press the F2 key. The software displays the Stage Move Settings window where you can specify the destination and initiate the move.

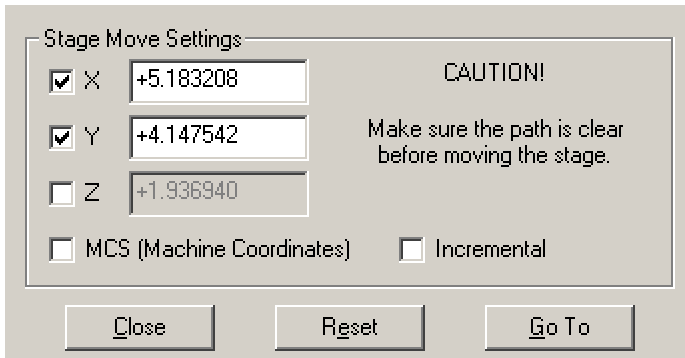
If a safe zone is enabled and the travel to the specified destination intersects with a safe zone, the software will cancel the move and display a message indicating that it cannot move the stage to the specified destination.

If you specify an axis or location that is beyond the range of travel, the software will move the stage to the physical stage limit of the selected axis (axes) and stop at the limit. The software will not display any error message or prompt if the WARN_TRAVEL_EXCEEDED parameter is set to 0.

Setting all the fields to zero will move the stage to the origin of the current coordinate system.

3.15.2 Stage Move Settings

When you select **System** / **Stage Move** from the main menu, press the **F2** key on the keyboard, or right-click in the DRO window, the software displays the Stage Move Settings window and populates the XYZ fields with the current XYZ location, i.e., the coordinates of the center of the field of view. This is the default setting.



When using the Stage Move function, you can do the following:

- Type the **X**, **Y** and/or **Z coordinates** to specify the desired location that the stage will move to. The coordinates are in the current units and are based on the current coordinate system.
  - Select the check box next to a coordinate axis to activate the field and specify the location to which the stage (or optical assembly in the case of Z) will move.
  - If a coordinate check box is not selected (default), the software grays out the displayed value and the stage does not move in that coordinate axis even when a value is specified. For example, if X and Y are checked but Z is not, the stage will move to the specified XY location at the current Z.
- Select the **MCS (Machine Coordinates)** check box to move the stage to the specified location relative to the specified Setup Zero location (or MCS, if a Setup Zero location is not defined). If the check box is not checked (default), the stage moves relative to the current Part Coordinate System (PCS).

The Machine Coordinate System (MCS) is the base coordinate system that is set up by a stage home when the machine starts up.

🗉 The Part Coordinate System (PCS) is the coordinate system that is in effect during a part run. This coordinate system is based on part routine steps that set datums. This coordinate system does not exist outside the scope of a part routine run.

- Select the **Incremental** check box to move the stage by the specified increment, e.g., 2 inches in the X direction. In this case you must specify the desired axes and increments. If the check box is not checked (default), the moves will be Absolute, i.e., to the specified coordinates.
- Click **Reset** to repopulate the coordinate fields with the current XYZ location.
- Make sure the path is clear before clicking the **Go To** button to move the stage.
- Click the **Go To** button to move the stage to the specified location in the selected axes.
  - If the MODEL_MOVE_STAGE parameter is set to 0, the software will move the stage without displaying a prompt.
  - If the MODEL_MOVE_STAGE parameter is set to 1 or 2, the software will display a prompt before moving the stage.
- Click the **Close** button to close the Stage Move window.

3.15.3 Z Axis Slide

To move the Z-axis slide and manually change the focus, twist the knob at the top of the joystick.

- Twist the knob clockwise to move the Z axis up.
- Twist the knob counterclockwise to move the Z axis down.

3.16 Control Panel

If your system is equipped with a control panel, you can control the following functions:

- Use the illumination knobs to control the sources and levels of light.
- Press the Zero Set and Axis Alignment buttons to zero out the XYZ axes and align the X axis. These buttons are used most often for the part setup.
  - The three **Zero Set** buttons on the control panel zero out the XYZ axes. You can also zero a location value by pressing the X, Y, or Z button in the DRO window.
  - Press the **Axis Align** button to set the horizontal alignment of the X axis. The red status light is lit when the axis is aligned. You can also set the alignment from the screen by pressing the Axis Alignment button in the DRO window.
- Select the units of measurement and coordinate system.
- Press the **Stop** / **Start** button to cut power to all motors and cancel the current measurement, and to resume operation again. This button is on the joystick on certain systems.

3.17 Mouse

Use the mouse to move the cursor to the desired location. Press the left button once and release it to select an item. For example, when you select Targets in the menu bar, the Targets pull-down menu appears. Then you can select a menu item.

You can also use the mouse to:

- Change the spacing of a selected tool or target. To do so, point the cursor on the tool or target, and press and hold the left button while dragging the border. See the Circle target for an example.
- Control XYZ stage motion, as outlined in the table on the next page.

> Do not configure the mouse buttons to do other functions because this may interrupt the operation of the measurement software.

To…	Do this…
Drive the XY axes to the point clicked in the Image window	1. Display the Image window. 2. Select the Crosshair target. 3. Click the desired point in the Image window.
Fine adjust the XY axes	1. Display the Image window. 2. Move the mouse cursor into the Image window. 3. Press and hold the right mouse button while moving the mouse in the desired direction of XY motion. 4. Release the right mouse button when finished.

To…	Do this…
Drive along the XY axes (requires 3-button mouse)	1. Display the Image window. 2. Move the mouse cursor into the Image window. 3. Press and hold the middle mouse button while dragging the mouse in the desired direction. The speed of XY motion increases as you move the cursor farther from the center of the Image window. 4. Release the middle mouse button when finished.
Drive along the Z axis (requires 3-button mouse)	1. Display the Image window. 2. Move the mouse cursor into the Image window. 3. Press and hold the **Ctrl** key on the keyboard. 4. Press and hold the middle mouse button while you drag the mouse in the desired Y direction – away from you moves the Z axis up (+) and toward you moves the Z axis down (-). The Z axis moves faster as you move the cursor farther from the center of the Image window. 5. Release the middle mouse button and **Ctrl** key when finished.

3.18 Function Keys

3.18.1 Alt Key

You can select a top-level menu item by pressing **Alt** and the underlined letter simultaneously. For example, to select System / Calibrate, press **[Alt] [S]** and then **[A]**.

You do not need to press the Alt key when you select an item from a submenu.

You can also use the Alt key with one of the function keys to automatically load and run a frequently used routine.

3.18.2 Alt Key with Routines

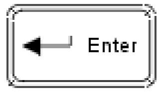
You can use the Alt key with one of the function keys to load and run a frequently used routine automatically.

To do this, save the frequently used routine with the name Fn.BXI, where "n" is the number of the function key. You can do this for up to 10 routines. Although the standard PC keyboard has 12 function keys (F1 - F12), you cannot use the Alt key with the F4 and F10 function keys.

When you press the **Alt** key with one of the function keys, the software automatically loads and runs the eBx routine.

> Make sure the routine is saved to the default routine folder, typically C:\QVSI\BASICX\ROUTINES. You can change the default routine folder by changing the PATHS ROUTINE_PATH parameter in the INPUT.INI file. See *How to Edit Configuration Parameters* for more information.

3.18.3 Enter Key



You can use the **Enter** key on the keyboard to simulate clicking the OK or Continue buttons that appear in prompts or the Measurement window.

Do not use the Enter key on the keyboard to enter points, however. If the cursor is in any other window, pressing Enter on the keyboard may activate other features unintentionally and cause unpredictable results.

If you want to enter points, press the Enter button on the joystick or click the Enter button in the DRO window (if enabled).

3.18.4 Keyboard Functions

The table below summarizes the keyboard functions used with the software.

Key	Function
[Alt] + [underlined letter]	• Select top-level menu items
[Alt] + [Tab]	• Pan (move) the entire model in any direction by holding the button down and moving the cursor in the desired direction
[Shift] + [Tab]	• Move the cursor back to the previous cell in the Tolerances window
Tab	• Move the cursor to the next cell in the Tolerances window
Enter	• Simulate clicking the **OK** or **Continue** buttons appearing in prompts or windows • Do not use this key to enter points; use the Enter button on the joystick or control panel

4. Setting Up the Part

4.1 Overview

This section describes how to:

- Stage a part
- Perform a part setup
- Use the datum functions

4.2 Staging a Part

Before you can set up a part and create or run a routine, you must make sure the part is staged properly.

Staging a part consists of the following procedures:

- Position the part anywhere on the stage or in any type of fixture.
- Make sure the part can be seen in the Image window.
- Clear any routine that may be in memory, if applicable.

4.2.1 Stage Home

Each time you start the measurement software the system displays the prompt "Press OK to initialize Home." When you click OK, the system performs the stage home process by driving the optics all the way up and the stage to the lower-left (front-left) corner.

This sets the machine origin (MCS) to XYZ zero. This is known as stage home.

- The machine's Z origin is approximately at the bottom of Z travel.
- The machine's XY origin is at the lower-left corner of the stage.

> The Machine Coordinate System (MCS) is the base coordinate system that is set up by a stage home when the machine starts up.

All measurements are based on stage home until you set up the part or define a datum in the routine.

You can also repeat the stage home procedure anytime you are in the measurement software by selecting **System** / **Stage Home** from the main menu. This also resets the machine origin to XYZ zero.

4.2.2 Staging a Part

Before you can set up a part and create or run a routine, you must make sure the part is staged properly.

Staging a part consists of the following procedures:

- Position the part anywhere on the stage or in any type of fixture.
- Make sure that the part can be seen in the Image window.
- Clear any routine that may be in memory, if applicable.

4.2.3 Staging a Part on the Stage

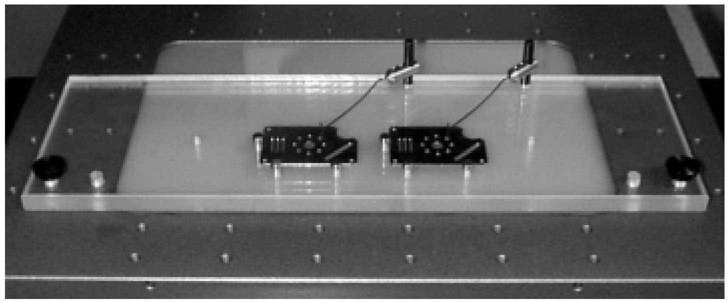
Staging a part on the stage enables you to set up and measure a single part quickly and easily.

To place and secure a part on the stage:

1. Clear the stage of any obstructions.
2. Place the part on the stage so it is level.
3. Secure the part to the stage so it will not move when the stage moves.

4.2.4 Staging a Part in a Fixture

Staging a part in a fixture enables you to place multiple parts in a fixture (if it is designed for that purpose) and to create one routine to measure all the parts in the fixture. Also, you can do the manual setup on the fixture instead of on each individual part.



To place and secure a part in a fixture:

1. Clear the stage of any obstructions.
2. Secure the part in the fixture.
3. Secure the fixture to the stage.

> It is strongly recommended that you enter the staging information in the setup instructions of the routine. For example, it is important to indicate where the part was secured in the fixture and how the fixture was secured to the stage.

4.2.5 Viewing the Part in the Image Window

After the part is secured on the stage or in a fixture, you must make sure that the part and the features to be used for the part setup can be seen clearly in the Image window.

To display the part or fixture in the Image window:

1. Turn on the desired light slightly and locate an edge or surface on the part. If it is not visible, move the stage so that the edge or surface appears at the center of the screen.

2. Place the cursor next to the edge or on the surface and turn on the desired light until there is sufficient contrast. For example, turn on the backlight to approximately 35% to 50% for the QVI training part.
3. If the edge or surface is not in focus, turn up the lights (if necessary) and twist the focus knob on the joystick until the image appears to be in focus. Then do an autofocus using the **Basic** Focus tool.

4.2.6 Clearing a Routine from Memory

It is recommended that you clear any routine that may be in memory whenever you want to start creating a new routine This is especially useful when you are measuring different parts or more than one person is using the software and someone else's routine may be in memory.

You do not need to clear the previous measurements if you are creating a new routine immediately after powering up the machine or if you are using the current routine to measure another identical part.

You can clear a routine in two ways:

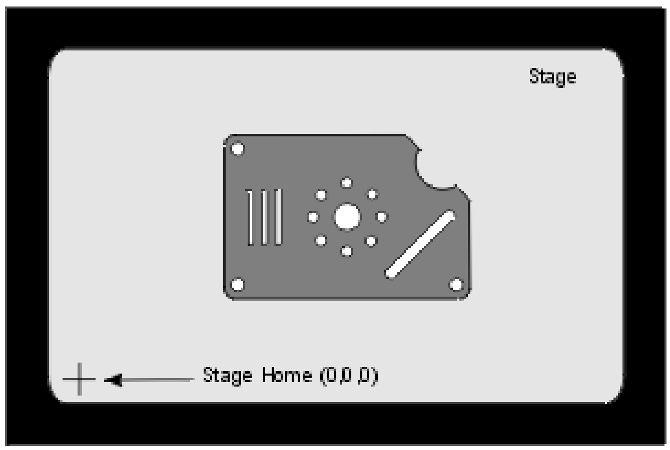
- Select **System** / **Reset** from the main menu to clear the routine in memory and to clear any previous XYZ Zero and Axis Align settings and reset the machine origin to stage home. For example, use this method if you are staging a different part. Also use this method if the part is placed in a fixture that is permanently attached to the stage and the routine uses stage home as the part location.
- Select **File** / **New** from the main menu to clear the routine in memory but use the same XYZ Zero and Axis Align settings. For example, you can use this method if you are staging another part in a fixture and the part location is based on the fixture features instead of part features.

If a routine is in memory, eBx displays a prompt to save or clear the routine.

4.3 Part Setup

When the system is turned on and the stage is initialized, it displays the XYZ coordinates based on the stage home position as shown here. After you stage the part, the coordinates are still based on stage home. Te software does not know where the part is located or how it is aligned.

Therefore, it is necessary to set up the part to define the part location and align the part.



Part setup is a manual and visual procedure that:

- Defines the setup zero location to tell the system where the part or fixture is located on the stage. This sets the Z axis to 0 and the XY axes to (0,0) at the respective locations of the selected features. The XY location also becomes the pivot point for the setup alignment.
- Aligns the part to tell the system how the part is rotated relative to the stage travel. For example, this rotates the coordinate system about the setup zero location such that the X axis passes through the location of the selected feature. The setup alignment also compensates for any misalignment between the part and the stage.
- Establishes the Setup-Zero Coordinate System (SCS).

The Setup-Zero Coordinate System (SCS) is set up with the manual part setup. The setup zero location and axis alignment tell the system where the part or fixture is located on the stage.

Part setup is done after staging the part and before you begin to create a routine. When you complete the part setup, the setup zero location (Zero X, Y and Z) and alignment are based on features of the part or fixture rather than on stage home.

Part setup enables you to quickly run a routine to measure the same or identical part at another location on the stage. The routine will run no matter where the part is placed, as long as the part setup is done in the same way using the same setup zero location and setup alignment.

If you are doing manual measurements or you are interested in a quick measurement of a single feature, it is not necessary to perform a part setup.

4.3.1 When to Set Up a Part

You must set up the part in the following situations:

- The first time you stage a part for measurement or start to create a routine
- Every time if additional samples of the same part are not located in the same place as the first part (for example, fixtures are not used or the fixtures are movable)

4.3.2 Useful Features for Setting Up a Part

When setting up a part, use features that are easily seen.

Useful features include focus points for the Z axis, and holes and corners for defining the part setup zero location and part setup alignment point.

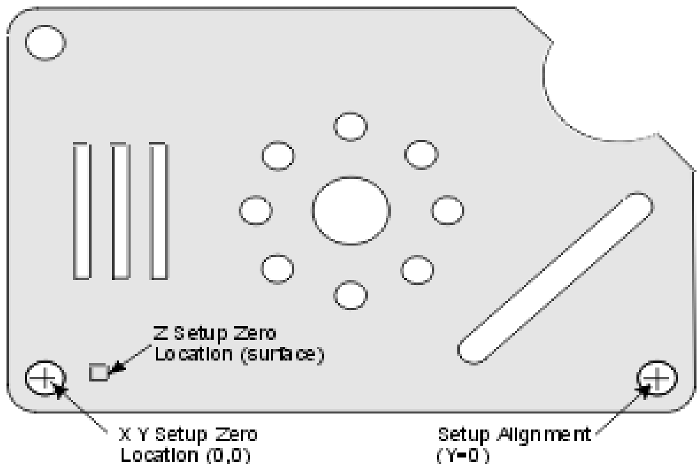
4.3.3 Ways to Set Up a Part

You can set up a part in the following ways to create and run a part routine.

- Manual (visual) setup on part features. With this method you can use features on the part itself for the setup zero location and setup alignment. For example, you can use the left and bottom edges, or the lower-left and lower-right holes on the QVI training part. This method is useful for quick part setups and when the part is not secured in a fixture (for example, it can be placed in different locations on the stage). If you use this method, you must do a manual setup each time you stage a part and load a measurement routine.
- Manual (visual) setup on the fixture. With this method you can use features on the fixture. A manual setup is required when the fixture is secured or moved.
- Using stage home. It is not necessary to set up a part if it is always located in the same place on a fixture that is permanently secured to the stage. In this case, just load the routine (File/Open) and run it (select File/Run and then click OK). However, it is strongly recommended that you define a datum within the routine (see *Datum Functions* for more information).

4.3.4 Manual Setup Using a Part Feature

You can do the manual part setup by lining up a feature of the part to an alignment target. For example, you can define the setup zero location and setup alignment using the lower-left and lower-right holes on the QVI FastStart training part.



To define the setup zero location and setup alignment using a part feature:

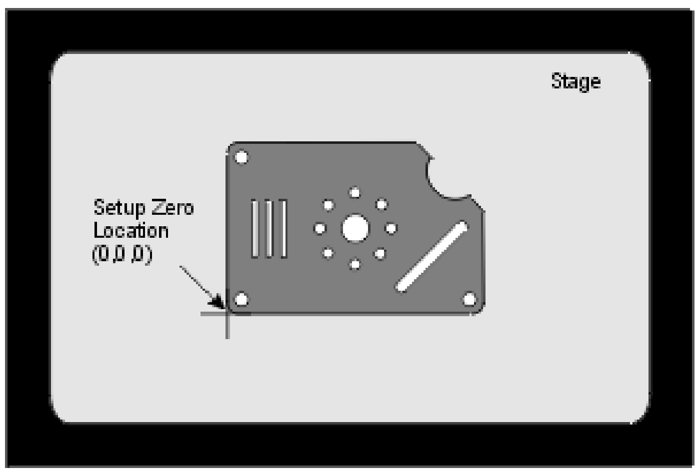
1. Select the Focus tool and adjust the box size as needed.
2. Make sure that the edge of the lower left hole is within the focus box and click in the Image window to do an autofocus on the edge.
3. Click the **Zero Set Z** button in the DRO window to zero the Z axis at the focus point.
4. Select an appropriate target, for example, Circle.
5. Move the stage so that the center of the hole lines up with the center of the target's crosshairs.
6. Size the target until the target and hole are the same size.
7. Click the **Zero Set X** and **Zero Set Y** buttons in the DRO window. The software displays all zeros in the X and Y fields.
8. Move the stage along the X axis in a positive direction until the lower-right hole lines up with the center of the target's crosshairs.

9. Click the **Axis Align** button in the DRO window. This "draws" the positive (+) X axis between the setup zero location and the setup alignment point. The X field displays the value along the new X axis. The Y field is reset to 0, reflecting the rotation of the coordinate system to align with the part.

It is strongly recommended that you enter the part setup information in the setup instructions of the routine.

4.3.5 Manual Setup Using an Edge or Surface

Another way to do the manual part setup is to "line up" a feature of the part to an alignment target. For example, you can define the setup zero location and setup alignment using the lower-left and lower-right holes on the QVI training part as shown here.

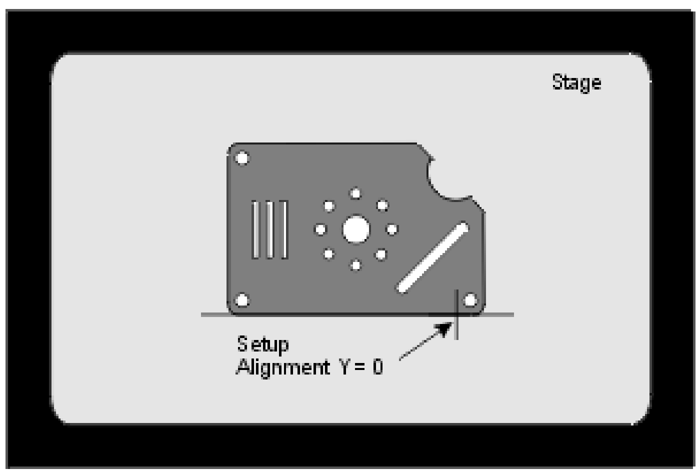


To define the setup zero location and setup alignment using an edge or surface:

1. Click the Basic Focus icon and adjust the box size as needed.
2. Make sure that the edge or surface is within the focus box and click in the Image window to do an autofocus on the edge or surface.
3. Click the **Zero Set Z** button in the DRO window to zero the Z axis at the focus point.
4. Pick an easy feature such as the lower left corner of the QVI training part.
5. Select an alignment target, e.g., Crosshair.

6. Move the stage so that the edge or surface is aligned with the crosshairs.
7. Click the **Zero Set X** and **Zero Set Y** buttons. This defines the setup zero location and the software displays all zeros in the X and Y fields.
8. Move the stage along the X axis in a positive direction with the same edge (or surface) appearing along the horizontal crosshair.
9. Stop at another location on the same edge (or surface) as far away as possible from the setup zero location. This becomes the setup alignment point.

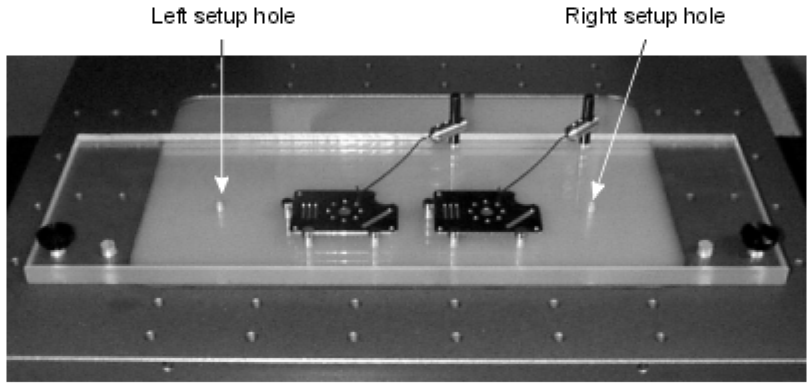
   The X and Y fields display the coordinates of the setup alignment point based on the motion of the stage's X and Y axes.
10. Click the **Axis Align** button in the DRO window. This "draws" the positive (+) X axis between the setup zero location and the setup alignment point. The X field displays the value along the new X axis. The Y field is reset to 0, reflecting the rotation of the coordinate system to align with the part.



It is strongly recommended that you enter the part setup information in the setup instructions of the routine.

4.3.6 Manual (Visual) Setup Using a Fixture

When you measure parts in fixtures, you can perform the manual part setup on fixture features rather than part features. For example, if you measure multiple parts, this enables you do the manual setup on the fixture only once (if you do not move the fixture) rather than on each separate part.



To define the setup zero location and setup alignment using features on a fixture:

1. Secure the part(s) in the fixture and secure the fixture to the stage.
2. Locate features on the fixture that can be used for the setup zero location and the setup alignment.
3. Select the appropriate illumination and adjust the focus to view the features.
4. Select an appropriate target for the setup zero feature.
5. Move the stage so that the target and setup zero feature are aligned.
6. Click the **Zero Set X**, **Zero Set Y,** and **Zero Set Z** buttons in the DRO window. The software displays all zeros in the X, Y and Z fields.
7. Select an appropriate target for the setup alignment feature.
8. Move the stage along the X axis in a positive direction until the setup alignment feature lines up with its target.
9. Click the **Axis Align** button in the DRO window.

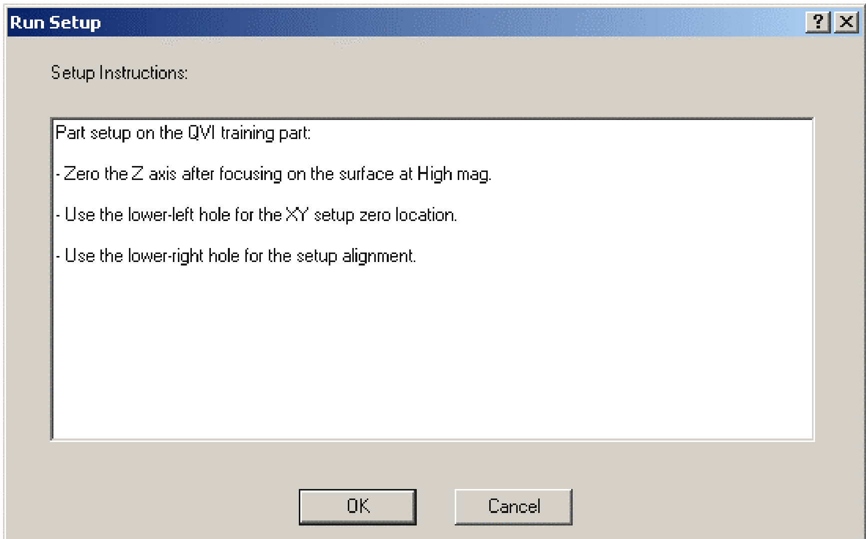
It is strongly recommended that you enter the part setup information in the setup instructions of the routine.

4.3.7 Part Setup Notes

- The Zero Set XYZ and Axis Align buttons are typically used only when you do the part setup.
- The Zero Set XYZ and Axis Align buttons are locked out when you are creating or editing a routine. However, you can use them when Run is selected and the run setup screen is displayed.
- If the setup zero location is at the upper or lower right and you need to move the part along the X axis for setup alignment, the stage motion will be in a negative direction. In this case the software also displays the axis alignment as a negative number. However, the model will display the part opposite to the way it is on the stage.
- Any previous Axis Align setting is no longer valid if you change the setup zero by pressing the Zero Set X and Y buttons again.
- If your system is equipped with the touch probe, you can also perform the part setup using the probe contact points.

4.3.8 Setup Instructions

When you select **Part Setup** / **Setup Instructions** from the main menu, a dialog box pops up where you can enter or view specific instructions for setting up the part. For example, you can specify the location, orientation and alignment of the part, or explain how to begin measuring the part. Also, if you are using a fixed lens, it is important to specify the magnification when you create a routine.



To enter each line of text:

1. Place the cursor in the dialog box and click the left mouse button once.
2. Type the desired text. If you reach the end of the line, characters will wrap to the next line.
3. If you want to go to another line, press **Enter** and type the desired text.
4. Click **OK** or Cancel to close the dialog box.
    - OK saves the instructions and any changes/additions you made.
    - Cancel saves only your original instructions; any additions or changes are not saved.

4.4 Datum Functions

A datum is a feature whose location is considered exact for the purpose of determining relationships to other features. Defining the datum typically aids with measuring features according to blueprint specifications.

It is strongly recommended that you do both the part setup and define a datum when you begin to create a routine. All routines should have the datum features and the datum steps at the beginning of the routine, regardless of how you set up the part.

Datums are constructed within a routine from previously measured or constructed features. These features must be measured or constructed prior to defining the datum. For example, if you want to include the part setup features in the routine, you need to measure the features first and then use the datum functions to define the datum origin and datum alignment.

> If you do not define datums in the routine, the measurements are based on the part or fixture location (PCS) established during the manual part setup.
>
> The Part Coordinate System (PCS) is the coordinate system that is in effect during a part run. This coordinate system is based on part routine steps that set datums. This coordinate system does not exist outside the scope of a part routine run.

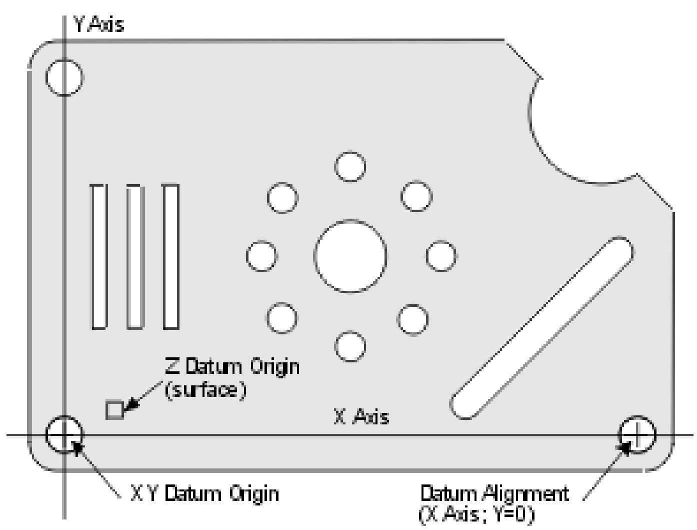
4.4.1 Defining a Part Datum

Datums define the location and orientation of a part within a routine and establish the Part Coordinate System (PCS).

Defining the datum origin and datum alignment in a routine consists of the following functions:

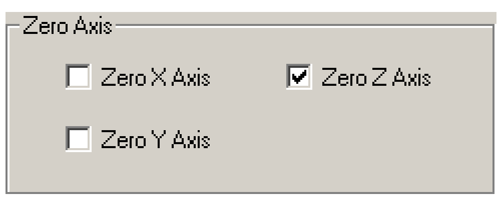
- Measure a point on the surface and then set the Z axis to zero (for 2D parts).
- Measure a plane and then use the datum level function (for parts where the plane is not level).
- Measure or construct a feature, e.g., lower-left hole, and then define the XY datum origin (0,0).
- Measure or construct a feature, e.g., lower-right hole, and then define the datum alignment point.

The instructions in these topics use features on the QVI training part. In these instructions, the datums are defined by measuring one feature at a time followed by a datum construction.



4.4.2 How to Set the Z Axis to Zero

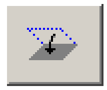
1. Turn up the surface light and move the stage so that a surface appears in the Image window.
2. Zoom to the highest magnification and adjust the surface light as needed.
3. Twist the focus knob on the joystick until the image is in best focus.
4. Click the **Basic Focus** icon in the toolbox. Change its size if needed, and make sure that it is completely over the surface.
5. Click in the Image window to perform an autofocus on the displayed surface.
6. Press **Enter** on the joystick to accept the focus point.
7. Click **OK** in the Measurement window to complete the measurement.
8. Click the **Construct** tab (Classic User Interface only) and then click the Datum **Origin** icon. The system displays the Model window and the Datum Origin settings in the Measurement window.
9. Click the measured focus point in the Model window.
10. Select the **Zero Z Axis** check box and then click **OK**. This sets the Z axis to zero at the measured location of the focus point.



If you have a machine with a self-calibrating zoom lens, it is strongly recommended that you reestablish the Z axis datum whenever you change the magnification in your routine and you plan to do Z axis measurements at the new magnification. To do so, repeat the initial measurement used for the focus and redo the Z axis datum step. Be sure to do the autofocus on the same edge or surface that you used for the current Z datum.

If you want to set the Z axis to zero on a reference plane, see the *Datum Level* topic.

4.4.3 Datum Level



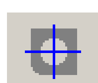
The datum level function establishes a Z axis datum using a reference plane with multiple measured focus points. This aligns the part coordinate system to the measured plane. After this leveling, all points on this plane have the same Z value.

To use the datum level function:

1. Measure a plane or construct a plane.
2. Click the **Construct** tab (Classic User Interface only) and then click the **Datum Level** icon.
3. Type the step number of the reference plane in the Reference field in the Measurement results window.
4. Click **OK**.

4.4.4 Datum Origin

The datum origin is set to zero by measuring a feature, followed by a datum origin construction using the selected feature.

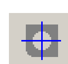


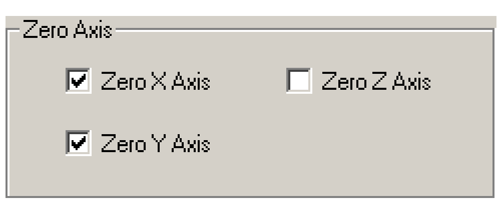
When you construct a datum origin, the Measurement window displays:

- Modifier radio buttons:
  - At Actual indicates that the datum origin is set at the location of the actual measured feature, e.g., the center of a circle. This is the default setting.
  - At Nominal indicates that the datum origin is based on the nominal location of a feature rather than the actual location.

- Check boxes to zero the selected axes. Select the appropriate check boxes to zero the selected axis or axes. For example, to set the XY origin to zero, select the Zero X Axis and Zero Y Axis check boxes. This sets the datum origin at the XY location of the selected feature.
- Boxes to set the X, Y, or Z axis to a non-zero value. This is typically used when you want to specify a precise location for the origin. For example, this may be an offset from an actual measured location.

4.4.5 How to Define a Datum Origin

1. [Classic User Interface] Click the **Image** tab to display the video image if it is not already displayed.
2. Zoom back to the lowest magnification, turn off the surface light, and turn on the backlight.
3. Move the stage until the desired feature, e.g., lower-left hole is in the field of view.
4. If needed, twist the joystick knob to focus the hole.
5. [Classic User Interface] Click the **Tools** tab.
6. Click the **FeatureFinder** icon and measure the desired feature, e.g., lower-left hole.
7. Click **OK** to complete the measurement.
8. [Classic User Interface] Click the **Construct** tab.
9. Click the **Datum Origin**  icon. The system displays the Model window and the Datum Origin settings in the Measurement window.
10. Click the feature, e.g., lower-left circle, to select it.
11. Select the **Zero X Axis** and **Zero Y Axis** check boxes.



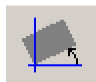
12. Click **OK** to complete the construction.

This sets the datum origin to (0,0) at the actual measured center of the lower-left hole. The Model window remains displayed. The intersection of the X and Y axes moves to the datum origin.

> 🗎 If you are using a line to define a datum, the datum operation uses the midpoint of the line (not the angle) in its calculations.
>
> 🗎 At Nominal is not used very often. If used, it is typically for datum constructions from dummy points. It is also used when you want to measure features from theoretical datums rather than actual measured datums.

4.5 Datum Alignment

Datum alignment is used to align the specified axis and set the orientation of a part.



You can construct the datum alignment in the following ways:

- Along the X or Y axis
- At a specified angle relative to the X axis
- At a specified value (X basic or Y basic)

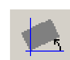
When you construct a datum alignment, the Measurement window displays:

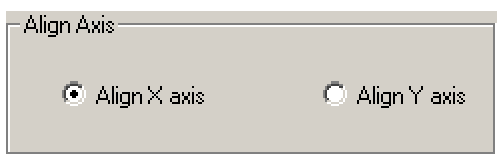
- Modifier radio buttons:
  - At Actual Location indicates that the datum alignment is set at the location of the actual measured feature, e.g., the center of a circle. This is the default setting.
  - At Nominal Location indicates that the datum alignment is based on the nominal location of a feature rather than the actual location.
  - At Feature Orientation indicates that the datum alignment is set at the axis of a feature (cone or cylinder).
- Radio buttons to construct the datum alignment along the X axis (default) or the Y axis.
- Field to specify a datum alignment angle relative to the X axis. If you do not enter a value, the angle is zero. The range of valid values is ± 180°.
- Radio buttons to set the datum alignment to X basic or Y basic along with a field to specify the basic value in the current units. If you do not enter a value, the default is zero.

4.5.1 How to Define the Datum Alignment

The datum alignment is done after the datum origin is defined.

To define the datum alignment:

1. [Classic User Interface] Click the **Image** icon to display the video image if it is not already displayed.
2. Move the stage until the desired feature, e.g., lower-right hole is in the field of view.
3. If needed, twist on the focus knob on the joystick until the hole is in best focus.
4. Click the **FeatureFinder** icon.
5. Measure the feature, e.g., lower right hole.
6. Click **OK** to complete the measurement.
7. Click the **Axis Align**  icon. The system displays the Model window and the Datum Alignment settings in the Measurement window.
8. Click the feature, e.g., lower-right circle, to select it. If the feature is not displayed, click the **Fit to Window** button to display all the features in the Model window.
9. Select the **Align X Axis** check box next.



10. Click **OK** to complete the construction.

This rotates the coordinate system about the XY datum origin such that the X axis passes through the location of the selected feature, e.g., the actual measured center of the lower-right hole.

> If you are using a line to define a datum, the datum operation uses the midpoint of the line (not the angle) in its calculations.

At Nominal is not used very often. If used, it is typically for datum constructions from dummy points. It is also used when you want to measure features from theoretical datums rather than actual measured datums.

4.5.2 Datum Alignment to X Basic or Y Basic

The datum alignment is typically defined at an angle of 0°. For example, on the QVI training part, the lower-right hole used for the datum axis is on the same axis as the datum origin. Thus, the angle is 0°.

In some cases the X or Y value (location) of the feature that is to be used for datum axis is not at an angle. Instead, it is specified as a fixed value. The basic value must be less than the straight-line distance from the origin to the feature. The X Basic or Y Basic value is shown on the dimension drawing (blueprint).

4.5.3 Other Methods for Defining Datums

How to Set the X Axis and Y Axis for the Datum Origin

In most cases the datum origin is set to zeros (Zero XYZ Axis). In some instances the dimension drawing may indicate a precise location for the datum origin. This may be an offset from an actual measured location.

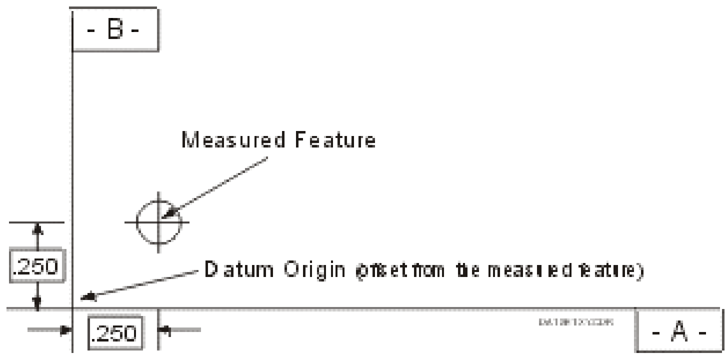
The Set X and Y Axis functions allow you to designate a specific location (offset from a datum feature) as the datum origin. The following example shows a datum feature set at (.250, .250) on the QVI training part.

To designate a specific location as the datum origin, follow the steps below.

1. Measure the datum feature (e.g., lower-left hole).
2. [Classic User Interface] Click the **Construct** tab.
3. Click the **Datum Origin** icon in the toolbox. The model is displayed over the Image window. The Measurement window displays the Origin entry window.
4. Click the measured feature (e.g., lower-left circle) to select it.

5. Type **.25** for Set X Axis and Set Y Axis, be sure to select the **At Actual Location** radio button is selected, and click **OK**.

   The intersection of the X and Y axes moves to the datum origin at the specified offset as shown here.



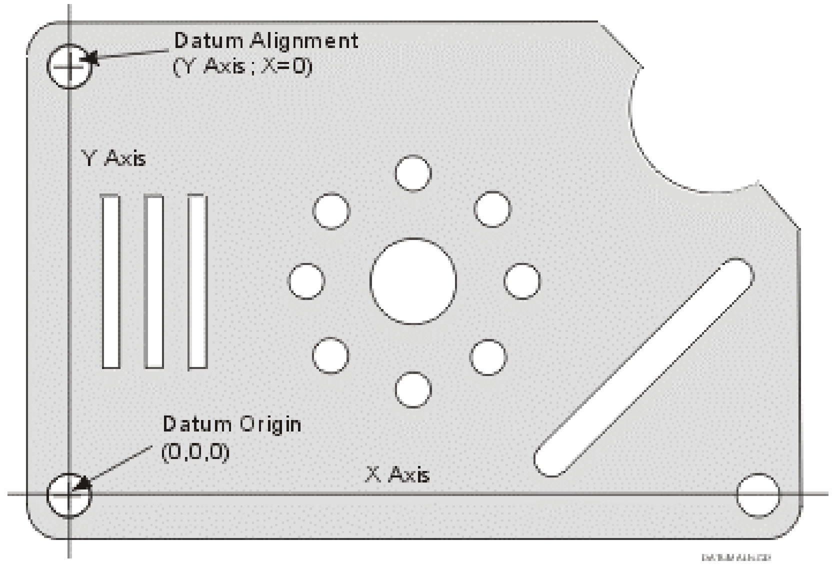
How to Set the Datum Alignment along the Y Axis

Datum Alignment is typically done along the X axis. In this case, the center of the lower-right hole of the QVI training part is used as the alignment point and the value of Y is 0.

If the datum alignment is set along the Y axis, the value of X is 0. To set the datum alignment along the Y axis, follow the steps below.

1. Measure the lower-left hole and the upper-left hole.
2. [Classic User Interface] Click the **Construct** tab.
3. Click the **Datum Origin** icon in the toolbox. The model is displayed over the Image window. The Measurement window displays the Origin entry window.
4. Click the lower-left circle to select it. Then select the **Zero X Axis** and **Zero Y Axis** check boxes to set the datum origin.
5. Click **OK** to complete the construction.
6. Click the **Datum Alignment** icon in the toolbox.
7. Click the upper-left circle to select it.
8. Select **Align Y Axis** radio button and also select the **At Actual Location** radio button. This aligns the Y axis at the actual measured center of the upper-left hole.
9. Click **OK** to complete the construction.

The Model displays the datum axis. It is similar to the training part shown here with the datum axis going along the Y axis through the center of the upper-left hole.



How to Set the Datum Alignment at an Angle

In some cases the datum alignment is at a specified angle relative to the selected axis.

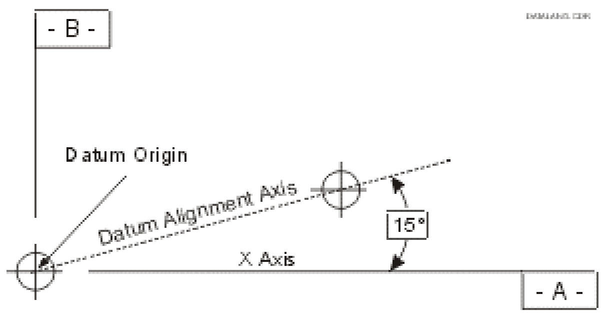
For example, to set the datum axis at a specified angle (e.g., 15°) relative to the X axis of the QVI training part, follow the steps below.

1. Measure the lower-left hole and the bottom hole of the bolt circle.
2. [Classic User Interface] Click the **Construct** tab.
3. Click the **Datum Origin** icon in the toolbox. The model is displayed over the Image window. The Measurement window displays the Origin entry window.
4. Click the lower-left circle to select it. Then select the **Zero X Axis** and **Zero Y Axis** check boxes to set the datum origin.
5. Click **OK** to complete the construction.
6. [Classic User Interface] Click the **Construct** tab.
7. Click the **Datum Alignment** icon in the toolbox.
8. Click the bottom hole of the bolt circle to select it.

9. Type **15** in the **Align Axis to Angle** box and select the **At Actual Location** radio button. This creates an alignment axis through the actual measured center of the bottom hole of the bolt circle.

10. Click **OK** to complete the construction.

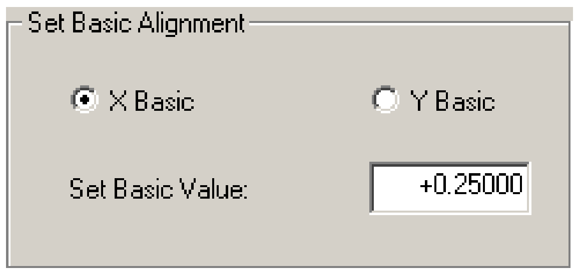
The Model displays the X axis at an angle 15° clockwise from the datum alignment point as shown here.



How to Set the Datum Alignment to X Basic

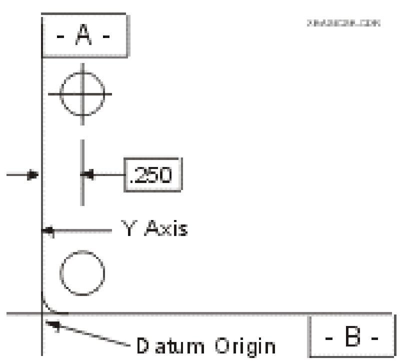
To set the datum alignment to X basic, follow the steps below. It is assumed that you have set the datum origin at the lower-left corner of the QVI training part.

1. Measure the upper-left hole.
2. [Classic User Interface] Click the **Construct** tab.
3. Click the **Datum Alignment** icon in the toolbox.
4. Click the upper-left hole to select it.
5. Select the **X Basic** radio button and type **.25** in the Set Basic Value box. This creates a Y axis .250 inches to the left of the actual measured center of the upper-left hole.



6. Click **OK** to complete the construction.

The figure here shows a simple dimension drawing with the part features that are used for the datum origin and datum alignment. The X value of the circle used for the datum alignment is .250 inches to the right of the Y axis.

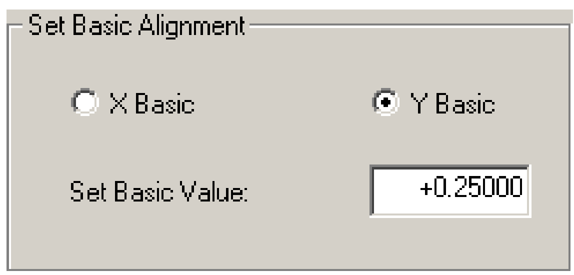


How to Set the Datum Alignment to Y Basic

The datum alignment is typically set along the X axis. In this case, the center of the lower-right hole is used as the alignment point and the value of Y is 0.

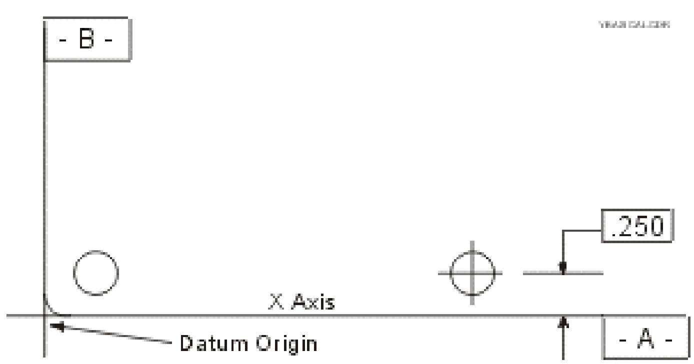
To set the datum axis to Y basic, follow the steps below. It is assumed that you have set the datum origin at the lower-left corner of the QVI training part.

1. Measure the lower-right hole.
2. [Classic User Interface] Click the **Construct** tab.
3. Click the **Datum Alignment** icon in the toolbox.
4. Click the lower-right hole to select it.
5. Select **Y Basic** radio button and type **.25** in the Set Basic Value box. This creates an X axis .250 inches below the actual measured center of the lower-right hole.



6. Click **OK** to complete the construction.

The figure here shows a simple dimension drawing with the part features that are used for the datum origin and datum alignment. The Y value of the circle used for the datum alignment is .250 inches above the X axis.



How to Set the Datum Alignment to a Feature's Axis

When At Feature Orientation is selected in the datum alignment step, you can set the datum alignment to the actual axis of a feature, i.e., cone or cylinder. This also enables you to construct the datum alignment along the Z axis.

For example, to align the X axis to the axis of a cylinder, follow the steps below.

1. Measure a feature, e.g., circle, to be used for the datum origin.
2. Measure a cylinder to be used for the datum alignment.
3. [Classic User Interface] Click the **Construct** tab.
4. Click the **Datum Origin** icon in the toolbox. The model is displayed over the Image window. The Measurement window displays the Origin entry window.
5. Click the circle to select it. Then select **Zero X Axis** and **Zero Y Axis** check boxes to set the datum origin.
6. Click **OK** to complete the construction.
7. Click the **Datum Alignment** icon in the toolbox.
8. Select the **At Feature Orientation** radio button.
9. Select the **Align X Axis** radio button.
10. Click the cylinder to select it.
11. Click **OK** to complete the construction. This aligns the X axis to the actual measured axis of the cylinder.

4.5.4 Guidelines for Defining Datums

Use the following guidelines to determine which features can be used in a datum step.

- Circle
  - Can be used for an XY origin
  - Can be used for axis alignment (angle, Y basic, X basic)
- Intersection between 2 measured lines
  - Can be used for the XY origin or axis alignment, but the same point cannot be used for both the origin and alignment
- Point
  - Can be used for the origin or XY axis alignment
- Plane
  - Can be used to set an origin in a datum level step
- Line
  - Can be used to set an X or Y origin or an XY axis alignment. However, each line must be in a separate datum step. If used for axis alignment, the alignment is through the XY zero point and the selected line's midpoint.

4.5.5 Guidelines for Changing Datums

When you create a routine you can change the part datum or define other datums as needed. Measurements that follow are based on the new datum.

Use the following guidelines when you change a datum.

- Setting the datum origin and the datum alignment is strongly recommended at the beginning of the routine.
- Once you change the datum origin, a new coordinate system is in effect. If you construct a measurement using steps that were measured prior to the datum change, their coordinates are converted into the new, current coordinate system.
- It is not recommended to insert a datum origin or datum alignment step in the middle of an existing routine.

If you copy steps that contain datums, make sure that you copy the datum steps (including the desired offset) separately. Then copy the remaining features with a zero offset.

This page was left blank intentionally.

5. Using the Measurement Tools & Targets

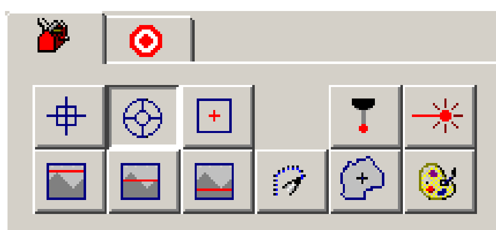
5.1 Overview

You can use the following kinds of tools to measure part features in the Image window:

- Edge analysis tools help you automatically measure points accurately and quickly.
- The Focus tool is used with different focus methods to focus an edge or surface and measure the Z axis position.

You can also use the alignment targets help you manually align geometric shapes and objects for accurate and precise location measurements.

To select a measurement tool, click the Tools tab in the toolbox and click the desired icon, or select it from the Tools menu.

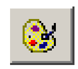


The selected tool or target remains on the screen until you select another tool or target. When you turn the machine on, the last tool or target that was used appears on the screen.

5.2 Tool and Target Color

You can change the color of an edge analysis tool, focus tool, or a manual alignment target. This allows you to choose the optimum contrast between the part image and the tool or target.

To change the color to any of the 16 basic colors:

1. Select the desired tool or target.
2. Click the color palette icon  in the Image window toolbar.
3. Click the desired color in the displayed palette.

The selected color becomes the default color until you change it again.

5.3 Edge Analysis Tools

The following edge analysis tools are available to measure edges:

- Use FeatureFinder when you want to find several points that define a feature in the shape of a line, arc or circle. The feature must be in the field of view. When you run the routine, the stage is positioned at the same location as it was when the routine was created. The FeatureFinder tool is superimposed over the feature, and all data points are measured without any additional stage movement.
- Use Weak Edge Point when you want to find one point on low-contrast or ragged edges.
- Use Strong Edge when you want to find one point on a strong edge. When you run the routine, the tool and edge are positioned at the center of the screen.
- Use Edge Trace when you want to measure many points on a line, arc, circle, Min/Max/Avg feature, or an irregular contour. The feature does not have to be in the field of view.

5.4 Field of View Measurements

The field of view is the maximum area that can be seen by the camera at one time. The field of view changes when the magnification is changed.

The edge analysis tools can be used to measure edges that are within the field of view as well as edges that are not in the field of view. However, the feature must be in the field of view when you measure it with the FeatureFinder tool.

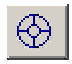
Full field-of-view processing allows you to measure any edge, visible anywhere within the field, without moving the stage. Integrating stage motion with field-of-view processing minimizes the number of stage moves. This significantly reduces the overall inspection time.

If a Feature Is Larger Than the Field of View

If a feature does not fit entirely within the field of view, you can:

- Lower the magnification until it does.
- Measure single points along the edge while moving the feature from one point to the next. Use one of the following tools:
  - Using the Strong Edge tool, select the desired measurement function and measure the individual points of the feature. When you run the routine, each point is measured at the center of the screen.
  - Using Weak Edge Point, measure each point in a separate step, and then construct the feature from the measured points. When you run the routine, the software displays the measured points in the Image window and the constructed feature in the Model window.
- Measure successive portions of the feature (in separate steps) using the FeatureFinder tool. Then use the Construct Line or Construct Circle functions, select the reference features, and select the Composite check box to construct the feature.

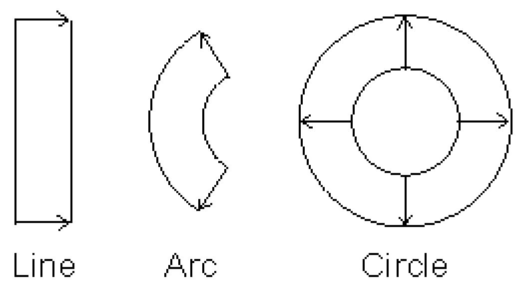
5.5 FeatureFinder



FeatureFinder enables you to measure practically any kind of edge quickly and easily. When you measure an edge, it automatically finds the points along the selected geometric shape (line, arc or circle), performs all the edge analyses, and displays the measurement results.

When you use FeatureFinder, it is very important that you carefully place the points **on** the edge and be sure that the points are spaced evenly from one other. This is critical for accurate measurements and part repeatability. Also, be sure to enter the nominal values and tolerances for the features that are measured.

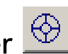
- Use FeatureFinder **Arc** to measure a feature such as a curve, or FeatureFinder **Circle** to measure a hole. You must specify three points on the edge.
- Use FeatureFinder **Line** to measure a feature such as a straight edge. You must specify a start and end point on the edge.



The feature must fit in the field of view and you cannot move the stage in between the points. If the feature does not fit in the field of view, use Weak Edge Point to measure individual points, for example, and then construct a composite circle or line.

5.5.1 How to Use FeatureFinder

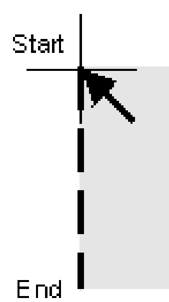
When you use FeatureFinder, you must be very precise in placing each point **on** the desired edge and be sure that the points are spaced evenly from one other. To measure an arc, circle or line with FeatureFinder, follow the steps below.

1. Click the **FeatureFinder**  icon or select **Tools / Weak Edge / FeatureFinder** from the main menu.

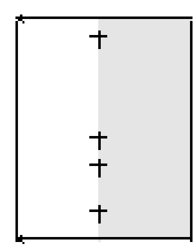
   The software displays the FeatureFinder tool.

2. Position FeatureFinder on the first point on the edge where you want to start the measurement.

   - To specify the starting point and measure the feature automatically, double-click the left mouse button. The system automatically determines whether the feature is a line or a circle, and measures it. This works best when the edge is of high contrast.
   - To measure a straight line, press and hold the left mouse button to indicate the location of the first point. While holding down the left mouse button, drag the line along the edge and release the button when you are at the second point. This completes the line measurement and defines the search area. It is recommended that the light be on the right as you drag the cursor from the start point to the end point.

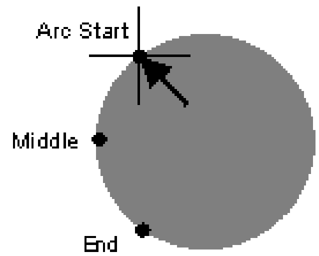


Defining a Line by Dragging a Line from the Start Location to the End Location

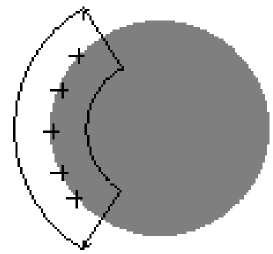


Edge Analysis Results

- To measure an arc, press the left mouse button. The software displays the first point. The indicated point will say either Arc or Circle. If it is Circle, click the same spot again to display Arc. Do Steps 3 and 4.

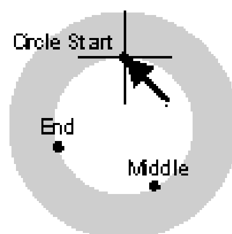


Defining 3 Points of an Arc

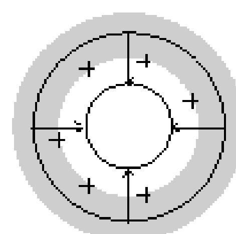


Edge Analysis Results

- To measure a circle, press the left mouse button. The software displays the first point. The indicated point will say either Arc or Circle. If it is Arc, click the same spot again to display Circle. Do Steps 3 and 4.



Defining 3 Points of a Circle



Edge Analysis Results

3. Place the FeatureFinder tool on the second point on the edge. Press the left mouse button. The software displays the second point.
4. Place the FeatureFinder tool on the third point on the edge. Press the left mouse button. The software displays the last point.

The software performs the edge analysis and displays the search area and the points that it found within that area. It displays at least three points for a circle or arc and at least two points for a line.

If you want, you can now change the boundaries of the search area and any of the parameters and weighting factors.

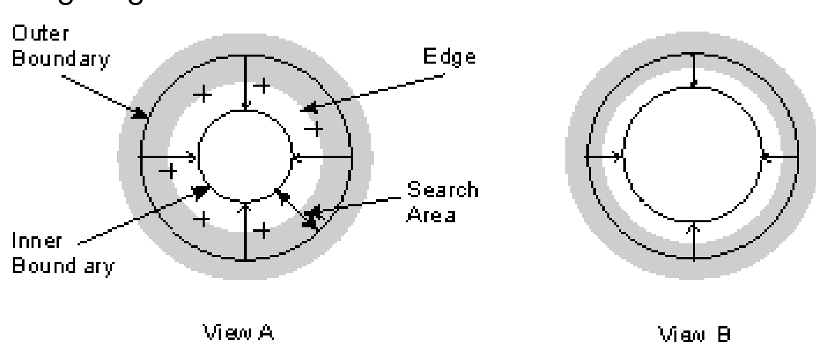
> If you are remeasuring an arc or circle, it is strongly recommended that you do **not** decrease the number of points; doing so may decrease the accuracy.

5.5.2 FeatureFinder Search Area

When you measure an edge with FeatureFinder, you define the search area when you select the points along the edge. The software searches this area and displays it after performing the edge analysis.

The software displays the following elements of the search area:

- Edge - The line or curve in the middle of the search area with points along the edge. The found edge is the nominal until you enter X, Y and Z nominal values. Then the entered nominal values are used in the edge analysis and they override the line or curve that was found originally.
- Inner boundary - The smaller of the two boundaries (for a circle) or the line between the two arrowheads (for a line).
- Outer boundary - The larger of the two boundaries (for a circle) or the line between the two arrow tails (for a line).
- Search area - The area between the inner and outer boundaries.
- Direction of search - The arrow indicates the direction of the search, going from dark to light. For example, if there are several edges, the first point that the software finds going in the direction of the arrow, is the first edge. However, it may not necessarily be used based on the other parameters and weighting factors.



To change the size (width) of the search area:

1. Press and hold the left mouse button to select either the inner or outer boundary.
2. Drag the boundary in or out with the cursor.

   Both the inner and outer boundaries move together in opposite directions. For example, the search area increases as you drag the inner boundary away from the edge. The search area decreases as you drag the inner boundary toward the edge.

The crosses, indicating the points along the found edge, are displayed when you create a routine and when you run a routine. The boundaries are displayed only when you are creating a routine or if the Display Windows check box is ON when you run a routine.

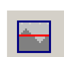
5.6 Weak Edge Point

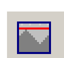
Weak edge point is typically used with low contrast or ragged edges or to measure points on a feature that does not fit in the field of view.

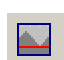
When you use this tool to automatically measure an edge, the software evaluates and processes a number of point samples within the search area, but it returns (displays) only one point.

When you use a weak edge point tool, you must be very precise in selecting the point. This is critical for accurate measurements and part repeatability. Also, be sure to enter the nominal values and tolerances for the features that are measured.

You can indicate the kind of weak edge point that the software should look for, in the search area that you define:

 **Average Weak Point**. This is the average point found within the search area. It is typically displayed in the middle of the search area.

 **Maximum Weak Point**. This is the point where the edge is closest to the arrowhead, based on the direction of the arrows in the search area.

 **Minimum Weak Point**. This is the point that is closest to the tail of the arrow, based on the direction of the arrows in the search area.

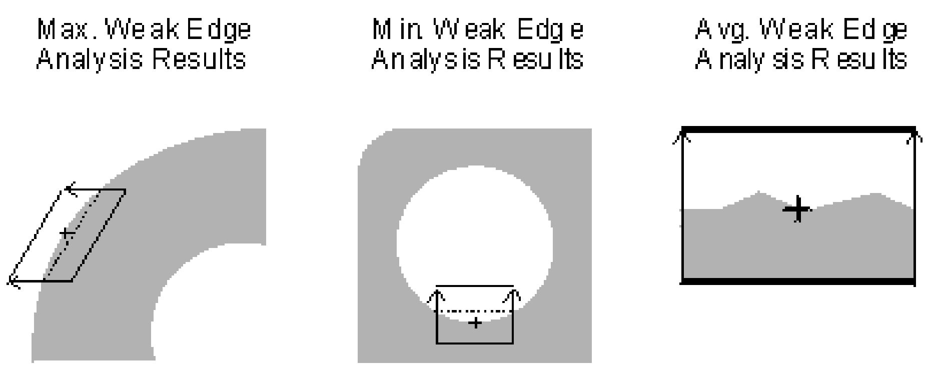
Tip If a feature such as a hole cannot fit in the field of view, use Minimum or Maximum Weak Point to measure at least three points. Then use the points to construct the circle and calculate the diameter or radius.

5.6.1 How to Find a Weak Edge Point

To find a point with one of the weak edge point tools:

1. Select the desired weak edge point tool in one of the following ways.
   - Click the desired weak edge point icon in the toolbox.
   - Select **Tools** / **Weak Edge** from the main menu and Average Weak Point, Maximum Weak Point, or Minimum Weak Point from the submenu.
2. Position the mouse cursor at the first point on the edge where you want to start the measurement.
3. Press the left mouse button to indicate the beginning of the search area. While holding down the mouse button, drag the cursor along or across the edge and release the button at the end of the search area. This defines the length of the search area.

   The software performs the edge analysis and displays the search area and the point that it found within that area. The measurement results appear in the Point Measurement Results window.
4. If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances. You can also change the weak edge settings and re-measure the point.
5. Click **OK** to complete the measurement.



5.6.2 Weak Edge Point Search Area

When you use a Weak Edge Point tool to measure a point on an edge, you define the search area when you drag the cursor along or across the edge. The software searches this area and displays it after performing the edge analysis.

The software displays the following elements of the search area:

- Edge - The line or curve in the middle of the search area with the found point. This point is the nominal until you enter X, Y and Z nominal values. Then the entered nominal values are used in the edge analysis and they override the point that was found originally.
- Inner boundary - The line between the two arrowheads.
- Outer boundary - The line between the two arrow tails.
- Search area - The area between the inner and outer boundaries.
- Direction of search - The arrow indicates the direction of the search, going from dark to light. For example, if there are several edges, the first point that the software finds going in the direction of the arrow, is the first edge. However, it may not necessarily be used based on the other parameters and weighting factors.

To change the size (width) of the search area:

1. Press and hold the left mouse button to select either the inner or outer boundary.
2. Drag the boundary in or out with the cursor.

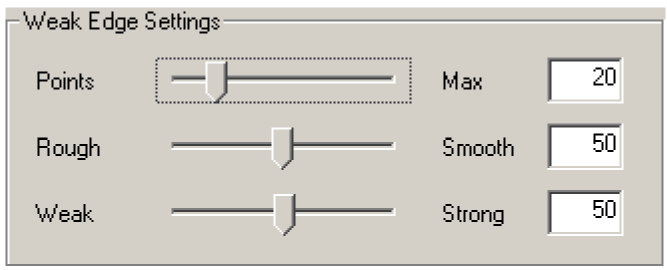
   Both the inner and outer boundaries move together in opposite directions. For example, the search area increases as you drag the inner boundary away from the edge. The search area decreases as you drag the inner boundary toward the edge.

> The cross, indicating the measured point, is displayed when you create a routine and when you run a routine. The boundaries are displayed only when you are creating a routine or if the Display Windows check box is ON when you run a routine.

5.6.3 Weak Edge Settings

When you measure a feature with a weak edge tool such as FeatureFinder or Weak Edge Point, the software evaluates the parameters listed below when it performs the edge analysis. It automatically adjusts itself for optimum performance and determines what it "feels" to be the best values based on the illumination, the edge that you have selected, and the size of the search area. You generally do not need to alter any of these decisions.

The weak edge parameters are displayed in the Weak Edge Settings window.



You can change any or all of the parameters using the slider bars and weak edge analysis buttons in the tool setting window:

- Points - This controls the number of patches that the software evaluates internally to determine the measured points.
  - Move the slider toward Max to increase the number of points for a better statistical sample.
  - Move the slider left to decrease the number of points to be evaluated. This may be useful if the edge is very ragged and this also reduces the system cycle time.
  - This parameter does not apply to Average Weak Edge Point.
- Roughness of the edge (for FeatureFinder measurements only) - For edges that are very ragged due to dirt, burrs, flash, etc., this indicates the extent to which evaluated points are included in the edge analysis.
  - Move the slider toward Smooth to statistically evaluate and remove points that do not belong in the analysis.
  - Move the slider toward Rough to ensure that more points are included in the analysis.

- **Strength of the edge** (contrast threshold) - This indicates the contrast level at which the software will accept an edge as a real or valid edge. The contrast is high when it is very dark (black) on one side of the edge and very light (white) on the other side. The contrast is low when there is a certain shade of gray on one side of the edge and a different shade of gray on the other side.
  - Move the slider bar toward Strong if the edge shows high contrast.
  - Move the slider bar toward Weak if the edge shows low contrast.
  - Normally you do not need to change this parameter unless the contrast of the edge varies significantly from one part to the next.

5.6.4 Weak Edge Analysis Buttons

Three buttons in the Weak Edge Settings window provide you with additional control of the weak edge analysis parameters and advanced weak edge settings. The buttons have the following functions:

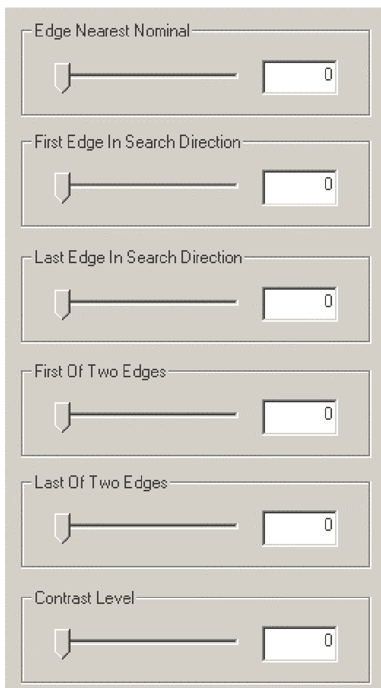
- **Remeasure** - If you change any of the parameters and/or advanced weak edge settings, click this button to test your changes. The software incorporates the changes into the edge analysis, re-measures the feature and redisplays the feature, allowing you to see the effect of the changes.
- **Reset** - If you do not like the effects of your changes, click this button to:
  - Reset all the advanced settings to the original default calculations.
  - Reset the weak/strong threshold to the default calculation.
  - Reset the number of points (min/max) for arcs and circles

> This button does **not** reset the number of points (min/max) for lines, the roughness of the edge (rough/smooth), or the boundaries.

- [Shift] + Reset - Resets the original boundaries and all the parameters and settings to the original default calculations done by the software.
- Advanced - Displays the advanced weak edge settings in the Measurement Results window.

5.6.5 Advanced Weak Edge Settings

The advanced weak edge settings indicate the relationship of an edge to neighboring edges. They are calculated when you first measure a point with FeatureFinder or Weak Edge Point. They are displayed in the Measurement window when you click the **Advanced** weak edge analysis button in the Weak Edge Settings window.



You can control the use and importance of each setting (that is, "fine-tune" which settings are used in the edge analysis) in the following ways:

- Move the slider right to increase the importance of a setting.
- Move the slider left to decrease the importance of a setting.
- Type the desired value in the field of the setting you want to change.

The weak edge analysis uses the advanced settings listed below.

- Edge Nearest Nominal - Highly weights the edge that is closest to where you clicked and dragged the cursor or to the center of the search area. The measured edge is considered the nominal until you enter X, Y and Z nominal values.
- First Edge in Search Direction - Highly weights the edge that is found first in the search area, going in the direction of the arrow.
- Last Edge in Search Direction - Highly weights the edge that is found last in the search area, going in the direction of the arrow.
- First of Two Edges - Highly weights the first of two edges that are very close to each other.
- Last of Two Edges - Highly weights the last of two edges that are very close to each other.
- Contrast Level - Highly weights the edge that has a contrast value closest to the value that was calculated during the initial measurement.

Press OK after making the desired changes to save the changed settings.

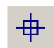
The most commonly used advanced setting is First Edge or Last Edge depending on the direction of the dark-to-light search.

Changing the advanced settings overrides the optimized calculations. For example, you may want to change them if the software is not picking the points at the desired location or when you expect part variations. Otherwise, it is recommended that you do not change them.

5.7 Strong Edge Tool

The strong edge tool is typically used with high-contrast, smooth edges. When you use this tool to automatically measure an edge, the software evaluates and processes a number of point samples within the search area, but it returns (displays) only one point.

To use the Strong Edge tool:

1. Click  in the toolbox or select **Tools** / **Strong Edge** from the main menu.
2. A strong edge tool appears at the center of the screen. The tool consists of crosshairs with a box around the center of the crosshairs. The area within the box is the search area.
3. Measure the edge in one of the following ways:
   - Non-directional search - This is the default search mode. It is typically used when the edge is a curve or circle, or if the edge being measured is the only edge in the field of view.
   - Directional search - This is typically used when several valid edges are very close to one another in the field of view. For example, a corner has two valid edges. A directional search may also be used when you want to pick the same edge consistently, such as the same side of a line.

The point selected will be automatically entered. Once entered, no additional points will be entered if changes are made within the target box. Therefore, direction of the target can be changed and the location of the point can be changed within the box even if the table is moved. The last point taken is the only one used. The point counter will not change as long as you are within the target box. If a point is taken outside of the target box, this point will be added at which time the point counter will update.

5.7.1 Strong Edge Search Area

You can change the size and shape of the Strong Edge search area both with the non-directional and directional searches. When many edges are close together, you may want to make the box smaller to make sure you measure the desired edge. On the other hand, if there are fewer edges or you are measuring the same feature on multiple parts, you may want to enlarge the box to make sure Edge Finder finds the desired edge.

To change the size of the search area:

1. Position the mouse cursor on one of the target box edges or corners and press the left mouse button.
2. Hold down the button and move the displayed hand in a left-right direction and/or an up-down direction.
3. Release the button when the box is the desired size.

If you pick one of the lines on the side of the box, only the horizontal width will change; if you pick either the line on top or the line on the bottom, only the vertical height will change. To change both the width and height at the same time, pick one of the four corners of the box.

When you enter each point in a routine, the size of the Strong Edge box becomes part of the routine. You can change the size and shape of the box as often as you like.

Tip To find an edge more easily:

- Use a horizontal box for a vertical edge.
- Use a vertical box for a horizontal edge.

Changing the box size in a directional search has an effect only in the length of the search.

When you change the box size, make sure it is not too large. If it is too large and more than one edge appears in the field of view, Edge Finder may find a different edge from the one you selected.

5.7.2 Directional Search

In a directional search, the Strong Edge tool looks for edges within the search area, in a direction specified by the user. To use this method you first need to define the search direction and then measure the desired edge.

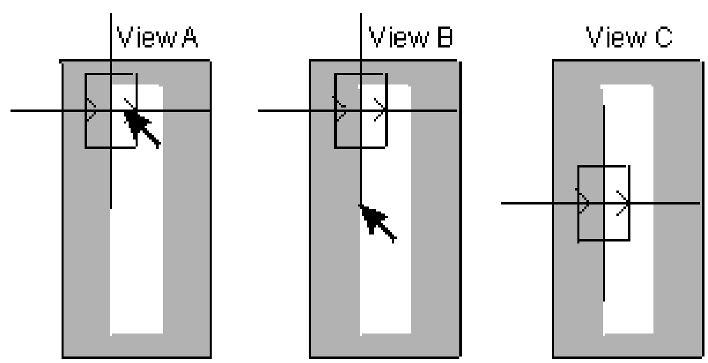
To define the search direction:

1. Position the mouse cursor **inside** the box (see View A).
2. Move the cursor around until you see arrows aimed in the direction you want to search (Left, Right, Up, Down).
3. Press the left mouse button once on an arrowhead while the arrows are on (see View A).

   The arrows will remain on when you move the cursor outside of the box, and all edge scans will be in the direction indicated. Within the box, Strong Edge will find the first edge in the direction indicated that is above the current contrast threshold.

To measure the edge:

1. Position the mouse cursor on the desired edge, **outside** the box (see View B).
2. Press the left mouse button once.



The software searches for a point in the direction indicated by the arrows. It moves the stage until the found edge appears at the intersection of the crosshairs in the center of the box (see View C).

The software will play back the measurement using the directions that you defined when you created the routine.

To return to the default non-directional mode, move the cursor to the middle of the box (so that no arrows are on) and click once with the left mouse button. No arrows should show when the cursor is outside of the box.

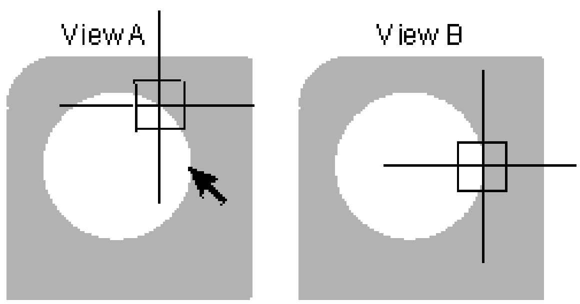
5.7.3 Non-Directional Search

In a non-directional search, the Strong Edge tool scans for the highest-contrast edge along the crosshair lines in both the X and Y directions.

To measure an edge:

1. Position the cursor on the desired edge, **outside** the box (see View A).
2. Press the left mouse button once.

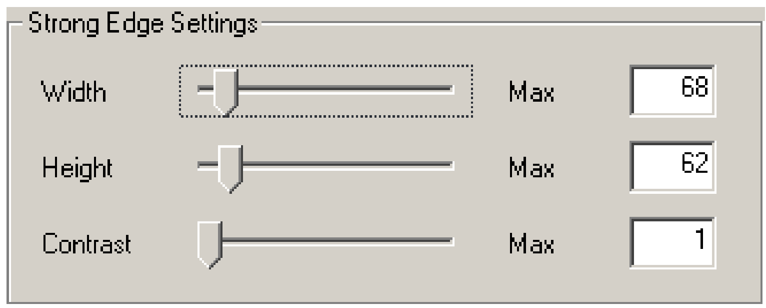
The software moves the stage until the found edge appears at the intersection of the crosshairs in the center of the box (see View B).



5.7.4 Strong Edge Settings

When the Strong Edge tool is displayed in the Image window, the Strong Edge tool setting window displays sliders to change the size of the Strong Edge box and the contrast threshold.

The percentage shown at the bottom of the window indicates the actual contrast of the last edge that was measured. You can change this percentage only by remeasuring the edge.



You can change the following settings in the Strong Edge tool settings window:

- **Width** and **Height** of the box. You can move the slider(s) or type the desired value(s) in the respective field(s).
- **Contrast threshold** at which the software will consider an edge valid.
  - Move the slider to the right to increase the threshold (to 10% in most cases) so that the software does not interpret dust particles as an edge.
  - Move the slider to the left to decrease the threshold.
  - Type the desired value (e.g., 10%) in the Contrast field.

Use the following guidelines when you change the contrast threshold:

- The percentage value must be lower than the actual contrast of the last measured edge. If you increase the value so that it is higher than the actual value, the software will not consider the edge valid when you measure the edge.
- It is recommended that the value controlled by the slider be approximately 1/2 the actual value. For example, if the actual value is 30%, the highest value that you should use is 15%.

5.7.5 Illumination Considerations with the Strong Edge Tool

- The Strong Edge tool can search for an edge both in a dark-to-light direction or a light-to-dark direction. The dark-to-light direction is recommended so that the desired edge can be found more easily. For example, going in a dark-to-light direction avoids erroneous edges that may be caused by improper illumination or dust particles.
- Strong Edge compensates for different light levels when measuring an edge. This means that the measured value for the edge will not change as the light level is changed.
- Strong Edge automatically changes the light level when you run or step through a routine during the routine playback. Any new light settings created during a routine run can be saved by resaving the part routine after the run has completed.

5.8 Edge Trace Tool

The edge trace tool enables you to measure many points on a line, arc, circle, Min/Max/Avg feature, or an irregular contour. The feature or contour does not have to fit entirely in the field of view; the stage will move automatically if required. Typically, the edge trace tool is used for the Measure Contour function and to measure circles and lines that are larger than the field of view because the measurement can occur in a single step.

You can access this tool by clicking  in the toolbox or selecting **Tools** / **Edge Trace** from the main menu.

When you select the edge trace tool, the system displays the Edge Trace Settings window at the bottom of the screen. Initially, only the cursor is displayed in the Image window; no other graphic is displayed.

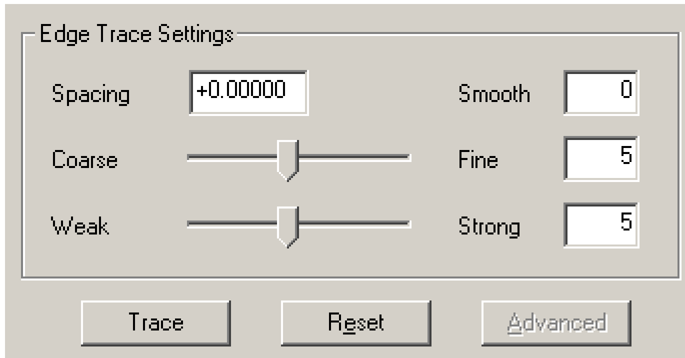
If you do not select a Measure function (e.g., Circle) before selecting the edge trace tool, the software activates the Measure Contour function automatically.

See *How to Use the Edge Trace Tool* for instructions on how to trace an edge.

> The edge trace tool cannot be used for distance, midpoint, plane, rotary, or sphere measurements.

5.8.1 Edge Trace Settings

The edge trace settings control window contains sliders, spacing and smoothness edit fields, and buttons to control the calculation and display of the edge trace points.



You can control the edge trace function in the following ways:

- Change the edge trace spacing value to control the spacing between each point.
- In the **Smoothness** field, specify the percentage of points that can be eliminated when the software calculates a line or circle feature. The default value is 0%. The maximum value is 50%.
- Use the **Coarse/Fine** slider to control the smoothing of the data points.
  - When you move the slider toward **Coarse**, the system smoothes the displayed data points by averaging many of the internal data points.
  - When you move the slider toward **Fine**, the system tracks the surface detail or variations more closely and includes them in the measurement.
- Use the **Weak/Strong** slider to control the path of the edge trace.
  - When you move the slider toward **Weak**, the system enlarges the search area. This slows the edge trace.
  - When you move the slider toward **Strong**, the system reduces the search area. This speeds up the edge trace.
- Click the **Trace** button to redo the edge trace after changing the spacing or moving any of the sliders.

- Click the **Reset** button to restore the system default settings.

5.8.2 Edge Trace Spacing

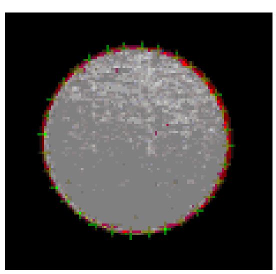
The edge trace spacing value controls the spacing between each point. The value is displayed in the spacing field in Edge Trace Settings window. The value is in the current measurement units (inches or millimeters).

The spacing value can be changed in the following ways:

- When you increase the spacing value, there is more space between each point and fewer points are used for the edge trace. If the entered value is too large (for example, larger than the distance from the start point to the end point), the system may display only the starting point.
- When you decrease the spacing value, there is less space between each point and more points are used for the edge trace.
- If you change the units, the system recalculates the default value and displays it in the current units.

5.8.3 How to Use the Edge Trace Tool

When you use the edge trace tool  the software traces the edge from the specified start point to the specified end point using the dark-to-light rule. The edge does not have to fit entirely within the field of view. The stage will move automatically if required. The software displays the measurement results in the Measurement window.



Before you begin to trace an edge, be sure to verify or change the spacing between each point by entering a value in the Spacing field in the Edge Trace Settings window.

You can trace an edge in the following ways:

- Specify the start point and activate the trace immediately. To do so, place the cursor on the desired edge and *double-click* the left mouse button quickly. The system traces the edge until it returns to the start point or reaches the maximum number of points.
- Specify a start point and an end point.
  - For the start point, place the cursor at the desired start point and click the left mouse button once. The software displays a dot with a box around it.
  - For the end point, place the cursor at the desired end point and click the left mouse button **twice** (or click the **Trace** button) to activate the trace. You can either double-click the left mouse button or click the mouse button once and then place the cursor over the displayed dot and click the mouse button again.
  - If you want to specify an end point that is not in the field of view, move the stage so that the desired end point location is displayed in the Image window.
- Specify the start point and end point as described above, and indicate the direction and size of the search area when the start point dot and box are displayed.
  - To change the size of the search area, place the cursor anywhere on the box, press and hold the left mouse button, and drag the box to the desired size. Then release the mouse button.
  - To indicate the direction, place the cursor over the start point dot, press and hold the left mouse button, and drag the rubber-band arrow in the direction that you want the trace to go. Then release the mouse button.

If you activate the trace by clicking on the end point, be sure that you click the point itself. If you click at any other location, the software removes the start and end points instead of starting the trace.

If you need to stop the edge trace measurement, press the **Stop** / **Start** button on the joystick (if equipped) or control panel (if equipped).

5.9 Focus Tool

The Focus tool is used to focus an edge or surface automatically and to measure the Z axis position of a part. The focus tool looks at image data from several Z axis positions and calculates the Z axis position that would yield the best focus, or highest contrast. You perform an edge focus when using the back light, and a surface focus when using the surface lights, and a grid focus using the grid light. You can also use laser focus to focus on a surface.

The image must be approximately in focus before you can start an autofocus. If it is not, focus the image manually by rotating the Z axis focus knob on top of the joystick. The image must have sufficient contrast in order for the focus to work.

To select the focus tool, click a focus icon in the toolbox or select **Focus** in the Tools menu. The Focus tool appears as a box in the center of the Image window.

 The **Basic Focus** tool appears as a box with a closed border. When you use this tool, the system performs an autofocus using general focus parameters.

You can change the size of the focus tool to any desired rectangular size. The maximum size is 632 pixels. The minimum size is 25 pixels. To change the size, select any edge or corner and hold the left mouse button while "dragging" the edge or corner to the desired size.

Use the Width and Height sliders or type a pixel value in the Autofocus Settings window.

When you use a Focus tool, it will automatically Measure a Point after completing the focus. Click **OK** to accept the point, or click **Cancel** to cancel the point measurement.

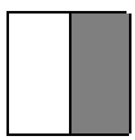
5.9.1 Focus Methods

You can use the Focus tool (Basic or Advanced) to focus an image automatically in two ways:

- Focus the image without creating a step in the routine. See *Edge Focus*, *Surface Focus* or *Grid Focus* for the procedure to do this.
- Include the autofocus of the image in a step in the routine. For example, select Measure Point, perform an autofocus, and click **OK** in the Measurement window. When you run the routine, the software plays back the autofocus just as it was done during the creation of the step.

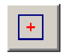
5.9.2 Edge Focus

When you perform an edge focus, they system looks for the sharpest contrast location on the edge of a part. It is used most often with the backlight; all other light sources are off.



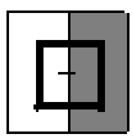
The edge focus is typically used to automatically set the plane where edge measurements will be made with an edge analysis tool, or to measure a height. See *How to Set the Z Axis to Zero* for the procedure to set the plane.

5.9.3 How to Focus on an Edge

1. Turn the ring and surface lights to zero. Turn the backlight On to an acceptable light level.
2. Click  in the toolbox.

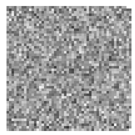
   The focus box appears in the center of the Image window with a white cross in the middle.
3. If you want, you can also change the size of the focus box in the Autofocus Settings window or by dragging the edge of the box with the mouse.
4. Position the desired edge inside the focus box.
5. Position the mouse cursor on an edge or surface in the Image window and press the left mouse button. The system performs an autofocus on the edge.

If the focus is successful, the cross turns green; if the focus is not successful, the cross turns red. The system also produces an audible signal (beeps for successful focus and buzzes for an unsuccessful focus) if the signals are turned on in System / Configuration / Sound.



5.9.4 Surface Focus

When you perform a surface focus, the system finds a surface by checking the contrast and focusing on the area within the Focus box. One Z axis value is returned from this focus.

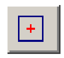


To measure a plane, you need to perform a surface focus on at least three different spots on the plane. This can be done in three separate Point steps or one Plane step.

You can also select the surface focus mode in the Autofocus Settings window when more than one surface of a part can be focused.

5.9.5 How to Focus on a Surface

1. Turn the back light to zero. Set the ring light, surface light, or programmable ring light so that the surface shows detail but does not become saturated. You may need to adjust the lights because, unlike edges, each surface has different characteristics based on material, color and finish.

2. Click  in the toolbox.

   The focus box appears in the center of the Image window with a white cross in the middle.

3. If you want, you can also change the size of the focus box in the Autofocus Settings window or by dragging the edge of the box with the mouse.

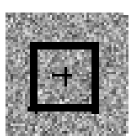
4. Select the desired surface focus mode in the Autofocus Settings window.

5. Position the desired surface inside the focus box.

6. Visually focus the image using the joystick.

7. Place the cursor on an edge or surface in the Image window and press the left mouse button.

The software scans the surface within the focus box (or on the grid), performs an autofocus on the surface, and displays the surface at the selected focus.



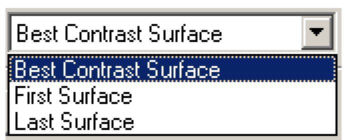
- If the focus is successful, the cross turns green; if the focus fails, the cross turns red.
- The system also produces an audible signal (beeps for a successful focus and buzzes for an unsuccessful focus) if the signals are turned on in System / Configuration / Sound.

5.9.6 Surface Focus Mode

The Surface Focus mode enables you to focus on an object when more than one surface of a part can be focused, such as a clear piece of plastic or a part with coating on it. This mode can also be used for layered parts. This mode can be used with the Basic Focus tool.

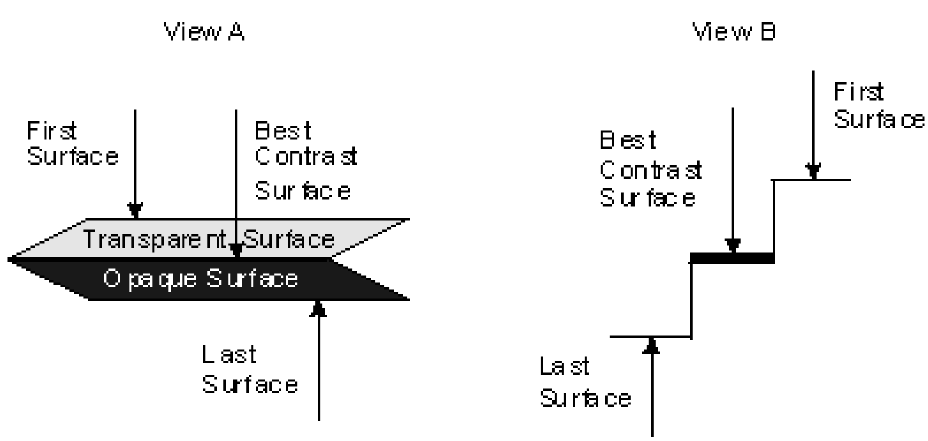
This mode scans the surface of the object along the Z axis. It allows you to choose which surface to focus on within the scan:

- **Best contrast surface**. The system performs a full scan and focuses on the surface with the maximum contrast.
- **First surface**. The system focuses on the top surface as it scans the object from top to bottom.
- **Last surface**. The system focuses on the bottom surface as it scans the object from bottom to top.



To perform a surface focus using the Surface Focus mode, see *How to Focus on a Surface*.

The examples here show how the system focuses on different surfaces that are in the field of view. If you select First Surface, the system focuses on the transparent surface on the top of a two-layered part; it focuses on the opaque surface if you select Last Surface. If you select Best Contrast Surface, the system focuses on the surface having the highest contrast.



5.9.7 Grid Focus

Grid focus projects a "checkerboard" pattern (LCD grid) or "honeycomb" pattern (LED grid) on the surface of the part, which creates an artificial contrast where none may exist. Typical applications include highly reflective surfaces and some translucent parts.

To perform an autofocus with the grid light:

1. Click  in the toolbox.
2. Turn on the surface light.
3. Zoom to a high magnification. (Grid focus is typically used at high magnifications).
4. Visually focus the image using the joystick.
5. Select the **Grid Light** check box in the Autofocus Settings window. The system displays a grid pattern on the screen.
6. If needed, change the light level (intensity) using a Surface light control.
7. Click anywhere in the Image window to perform an autofocus.
8. If you want to remove the grid pattern, clear the **Grid Light** check box.

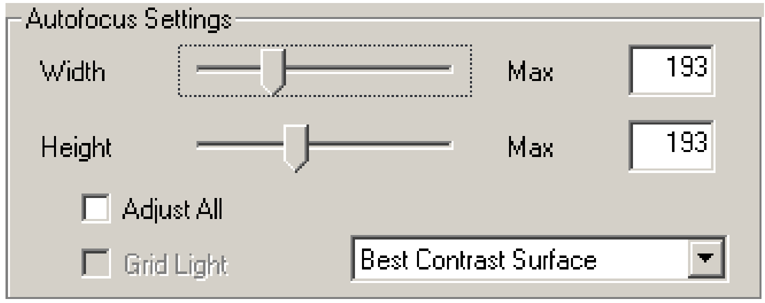
Tip It is recommended that you do not use the ring or programmable ring light with grid focus.

Grid focus is available only on systems that are equipped with the grid projector.

If your system is equipped with an LED grid light and it is enabled (see the GRID_PRESENT and GRID_TYPE configuration parameters), you cannot use it at magnifications below 3x.

- If you are at a low magnification and select the Grid Light check box, the software does not turn on the grid.
- If the grid is turned on and you zoom to a magnification below 3x, the software clears the grid check box, turns the grid off, and turns on the surface light to its current setting.

5.9.8 Focus Settings



You can change or select the following autofocus settings in the tool setting window when the Manual Focus Target or Basic Focus tool is displayed in the Image window:

- Change the size of the tool to any desired rectangular size by moving the **Width** and **Height** sliders or typing a pixel value in the respective fields. The maximum size is 632 pixels. The minimum size is 25 pixels.
- Select the **Adjust All** check box if you want to adjust both the Width and Height at the same time when you move either slider.
- Turn on the **Grid Light** (if equipped) to perform a grid focus, where the system projects a "checkerboard" pattern (LCD grid) or "honeycomb" pattern (LED grid) over the surface of the part, which creates an artificial contrast where none may exist.
- Select the focus mode, i.e., which surface the system should focus on as it scans the surface of the object along the Z axis. This enables you to focus on an object when more than one surface of a part can be focused, such as a clear piece of plastic or a part with coating on it. This focus mode can also be used for layered parts.

5.9.9 Focus Guidelines

Use the guidelines below to get the best results with the autofocus function.

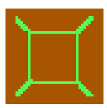
- Edges and surfaces are focused more accurately and repeatably at higher magnification settings. This is because the depth of field gets smaller as the magnification increases.
- The cross in the middle of the Focus box provides a visual indication whether an autofocus is successful or not (green = successful, red = failed). If you want, you can also use sound for an audible indication. This sound can be turned on in System / Configuration / Sound.
- Additional lenses that may be attached to the bottom of the optical system affect the magnification, working distance, and the depth of field. If you need to attach or remove a lens, make sure you select the appropriate lens in System / Configuration / Optics.
- Use surface illumination and do an autofocus on a surface if you are doing Z-axis measurements.

5.10 Centroid Tool

The Centroid tool enables you to measure the area and perimeter of an irregular contour. For example, this tool is useful for measuring leads, solder points and pins. The feature or contour must fit entirely in the field of view. The Centroid tool can be used with the Measure Centroid and Measure Circle functions.

To access this tool, click  in the toolbox or select **Tools** / **Centroid** from the main menu.

The software displays the tool as a box with corner handles in the center of the Image window. The software displays the Centroid Settings window next to the DRO window.



- Select any corner of the box and hold the left mouse button while "dragging" the corner to the desired size.
- Use the Width and Height sliders in the Centroid Settings window.

5.10.1 How to Use the Centroid Tool

When you measure a feature with the Centroid tool, the system measures the portion of the feature that lies within the boundaries of the Centroid tool.

To measure a feature in the Image window with the Centroid tool:

1. Select the desired Measure function, i.e., Measure Circle or Measure Centroid. If you do not select a Measure function before selecting the Centroid tool, the software activates the Measure Centroid function automatically when you click in the Centroid box to start the measurement.
2. Select and adjust the desired parameters in the Centroid Settings window and/or the Advanced Centroid Settings.

3. Click inside or outside the Centroid box in the Image window to start the measurement.

   The system measures the foreground pixels using the selected parameters and displays the measured area within the Centroid tool using the tool's color. The measurement results are displayed in the measurement step.

4. If the results are not what you expected, repeat Step 2 and click the **Remeasure** button as often as needed.

   You can click the **Reset** button if you want to reset all the check boxes and sliders to their default settings.

5. Click **OK** in the measurement step.

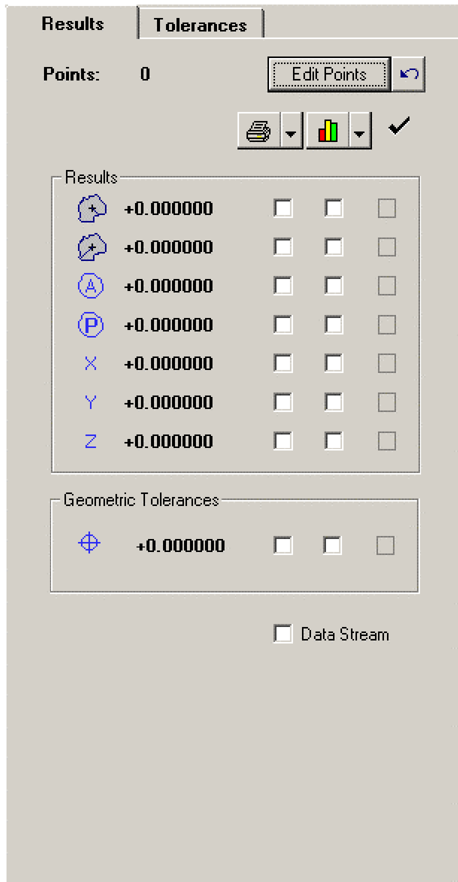
   A Centroid measurement appears as an octagon in the Model window.

   When you measure a circle with the Centroid tool:

   - The software fills in the measured area in the shape of a circle.
   - The measurement results display multiple points, which are taken around the perimeter of the circle.
   - The software does not calculate the circle over multiple regions, even if the check box is selected.

5.10.2 Centroid Measurement Results

This function reports the location and size of an irregular feature that was measured with the Centroid tool.



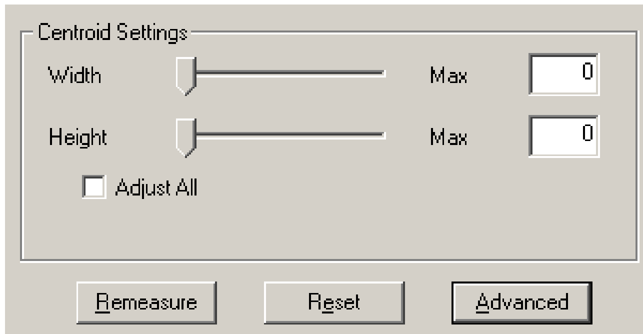
The Measurement window displays:

- Number of points in the measurement. The value is always 1. You can click the Points button to remove the measurement.
- Minimum radial distance from the center location
- Maximum radial distance from the center location
- Area of the centroid
- Perimeter of the centroid
- XYZ location of a centroid; this is the center of the mass of the foreground pixels
- True position tolerance

- Check box for data stream output (if the centroid was measured)

5.10.3 Centroid Settings Window

The Centroid Settings window contains check boxes, sliders and buttons to control the calculation and display of the centroid.



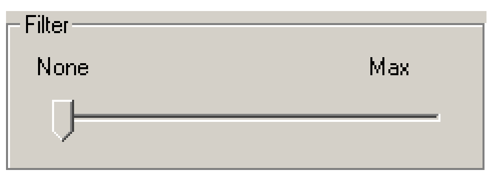
You can control the centroid function in the following ways:

- Change the size of the tool to any desired rectangular size by moving the **Width** and **Height** sliders or typing a pixel value in the respective fields. The maximum size is 632 pixels. The minimum size is 25 pixels.
- Select the **Adjust All** check box if you want to adjust both the Width and Height at the same time when you move either slider.
- Click the **Remeasure** button to remeasure the area within the boundaries of the tool, after changing the settings.
- Click the **Reset** button to remove the centroid and reset all the check boxes and slider bars to their default settings.
- Click the **Advanced** button to display the Advanced Centroid Settings in the Measurement window.

5.10.4 Advanced Centroid Settings

Click the **Advanced** button in the Centroid Settings window to display the Centroid advanced settings in the Measurement window.

- Select the **Multiple** check box to activate the slider for multiple regions.
- Use the Centroid **Contrast Threshold** slider to include or exclude features from the centroid calculation. This parameter is evaluated first when the measurement is started.
- Use the **Filter** slider to apply a low-pass, texture noise filter to "smooth" an area of the image before any subsequent processing takes place. This removes stray and undesirable pixels, e.g., dirt, from the calculation.
  - The range is 0 (none) to 10 (max) squared pixels, going from the center of the centroid box. For example, a value of 5 indicates that the area is 25 pixels.
  - Move the slider toward Max to increase the area over which the filter is applied.



- Select the **Fill** check box to fill in (smooth) any gaps in the centroid. This fills in any closed spaces within the measured region(s).
- Select the **Touch Boundary** check box to include any area that touches the boundary of the search area in the calculation.
- Select the desired illumination control, which enables you to specify the foreground pixels that are used in the calculation.

5.10.5 Centroid Multiple Regions

The Multiple check box and slider in the Advanced Centroid Settings window allow the centroid to be calculated over multiple regions within the centroid box. The smaller regions, which should be similar in size, are calculated in relation to the largest region.

Select the **Multiple** check box to activate the slider and allow multiple regions to be selected.

The Multiple slider controls the ratio of a region to the size of the centroid box.

- 0% (default) disables the checking of a region's size compared to the centroid box. However it does not disable the calculation of multiple regions. This means that all found regions will be displayed and included in the area and centroid calculation.
- When the ratio is greater than 0%, the found region must use a greater percentage of the centroid box than the percentage indicated by the slider.
- At 100%, the region must cover the entire area, i.e., be the size of the centroid box.

5.10.6 Centroid Contrast Threshold

Use the Centroid Contrast Threshold slider in the Advanced Centroid Settings window to include or exclude features from the centroid calculation. The center of mass is the center of the total area under (or over) the contrast threshold.

- The range is 0 (black) to 255 (white). A value of 0 (default) indicates that the thresholding is done automatically.
- When Dark is selected, moving the slider left includes only the darkest features and moving the slider right allows more lighter features to be considered as dark features and included in the calculation.
- When Light is selected, moving the slider right includes only the lightest features and moving the slider left allows darker features to be considered as light features and included in the calculation.

> 🗎 The calculated results will vary depending on how much the contrast threshold is changed.

5.10.7 Centroid Illumination Controls

The **Illumination** drop-down list in the Advanced Centroid Settings window allows you to select whether light or dark image pixels are considered to be foreground pixels.

- If **Auto** is selected (default), the system automatically determines the foreground and background pixels and displays the centroid either over the dark area or the light area.
- If **Dark** is selected, the image pixels with an intensity *below* the centroid contrast threshold are considered to be foreground pixels. The remaining pixels are considered to be background pixels.

- If **Light** is selected, the image pixels with an intensity *above* the centroid contrast threshold are considered to be foreground pixels. The remaining pixels are considered to be background pixels.

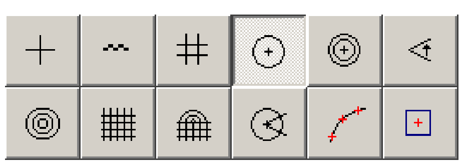
5.11 Manual Alignment Targets and Manual Focus Target

The manual alignment targets help you align geometric shapes and objects for accurate and precise location measurements:

- Crosshair
- Microgage
- Box
- Circle
- Protractor
- Double Circle
- Radius
- Grid
- Multiple targets
- Chart targets (radius, grid, combination)

Use the Manual Focus target to perform an autofocus on the desired edge or surface.

To select a manual target, click the **Targets** tab (Classic User Interface) in the toolbox and then click the desired icon, or select it from the **Target** menu.



You can also select the desired target and change the target size or spacing in the Target Settings window.

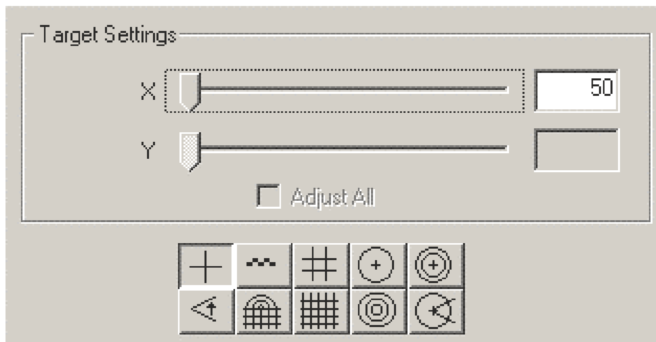
Click the target color icon above the Image window to change the target color.

When using an alignment target, the software hides the "?" cursor when you hold the right mouse button while moving the stage.

5.11.1 Target Settings

In the Target Settings control window, you can click the desired alignment target icon to select the desired target and change the target size or spacing.

This window displays different settings for each target.

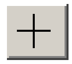


To change the size or spacing of a target, move the slider(s) left (decrease) or right (increase). The size or spacing, in pixels, is displayed in the field next to the slider.

You can also size the target directly in the Image Window by dragging its edge in or out. This method is not available for the Crosshair and Microgage targets and the charts.

You can also select the desired target from the **Target** menu.

5.11.2 Crosshair Target



The Crosshair target is used to manually align edges to the center of the screen.

The lines are very thin at the intersection and thicker farther out. You can change the amount of thin or thick line that is displayed using the X slider in the Target Settings window.

You may also move the stage so that a specified location appears at the intersection of the crosshairs in the center of the field of view. To do so:

1. Position the mouse cursor at the desired location in the Image window.
2. Press the left mouse button once. The software moves the stage and the specified location appears at the intersection of the crosshairs.

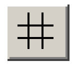
The software moves the stage only if the DRIVE_TO_CLICK_LOCATION parameter is on in the Video section of the Configuration Editor. The default setting is off.

5.11.3 Microgage Target

The Microgage target has a small crosshair at the center and alternating alignment blocks farther out.

It is unique because it creates a centerline that has no thickness. The top and bottom of the alternating rectangles are exactly co-linear.

5.11.4 Box Target



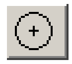
The box target is used to manually align square or rectangular features to the center of the screen. It is formed by the intersection of a set of horizontal and vertical parallel lines.

You can change the box target in the following ways:

- In the Image window:
  - Move both sets of lines together by dragging a corner (where the horizontal and vertical lines intersect).
  - Move one set of lines by dragging either the horizontal or vertical line.
- In the Target Settings window:
  - Move the X slider to change the distance between the two vertical lines
  - Move the Y slider to change the distance between the two horizontal lines
  - You can also select the **Adjust All** check box if you want to adjust both distances at the same time when you move either slider.

The approximate size of the box (to the nearest pixel) is displayed in the fields next to the X and Y sliders in the Target Settings window.

5.11.5 Circle Target



The circle target is used to manually align circular features such as holes and rounded corners to the center of the screen. The target has a small Crosshair indicating its center.

- Move the Radius slider in the Target Settings window OR
- Drag the circle in or out in the Image window.

The approximate size of the target (to the nearest pixel) is displayed in the Radius field in the Target Settings window.

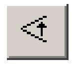
5.11.6 Double Circle Target

The double circle target consists of two concentric circles displayed on the screen at the same time.

The approximate diameter of each circle is displayed in the lower-left corner of the Image window.

If you create a routine, you can set the size of the two circles using the Measure Point function. When you run the routine, it positions the stage at the specified location and displays the two circles at their appropriate sizes.

5.11.7 Protractor Target



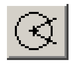
The protractor target is used for quick checks of the included angle between lines and for alignments to the intersection point. It is formed by two lines (legs of an angle) that begin at the small crosshair in the center of the Image window. The arrows show the included angle.

- In the Image window:
  - Move either leg by placing the cursor anywhere on the leg and dragging it with the mouse. The included angle changes as you move the leg.
  - Rotate the entire protractor around the crosshair. Place the cursor directly at the point where the leg and included angle arrow meet. Then hold the left mouse button and move the mouse. The size of the angle does not change.
- In the Target Settings window:
  - Move the Angle 1 slider to change the included angle.
  - Move the Angle 2 slider to rotate the entire protractor around the crosshair.

The approximate angles (included angle and leg 1) are displayed in the fields next to the angle sliders.

If you want, you can display an additional crosshair, e.g., for use with part alignments. To do so, specify a value of 1 for the DRAW_ALTERNATE_PROTRACTOR parameter in the Default Preferences section of the configuration file editor.

5.11.8 Multiple Targets



The Multiple option allows you to simultaneously display up to five alignment targets.

To display the currently selected targets, click  in the Target Settings window or select **Targets** / **Multiple** / **Display** from the main menu.

To select or change which targets are displayed:

1. Select **Targets** / **Multiple** / **Define** from the main menu.

   A list of targets appears in a pop-up dialog box with check marks next to the current selections.

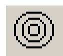
2. Select check box next to the desired target. A check mark appears in the check box and the target appears in the Image window.
3. Size the target as desired.
4. Repeat the two steps above for the other desired targets.
5. If you want to remove one of the targets, clear the appropriate check box. The check mark and the target disappear.
6. Click **OK**.

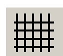
   All the multiple targets appear together in the Image window.

Click the color icon above the Image window to change the color of the selected targets.

5.11.9 Chart Targets

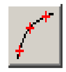
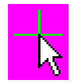
Charts are used to manually align circular or rectangular features to the center of the Image window.

 Radius chart. It is formed by a series of concentric circles centered around the Crosshair target. To change the spacing of the circles only, move the Radius slider in the Target Settings window. The approximate diameter of the inner circle (to the nearest pixel) and the radius increment to the next circle are displayed in the field next to the slider.

 Grid chart. It consists of a series of intersecting lines that form boxes centered around the Crosshair target. To change the spacing of the intersecting lines only, move the X slider in the Target Settings window. The approximate distance from one line to the next (to the nearest pixel) is displayed in the field next to the slider.

 Combination of both charts. This includes a radius chart above the horizontal axis and a grid chart below the axis. You can change the spacing of the circles and intersecting lines together only by sliding the X slider in the Target Settings window. The approximate diameter of the inner circle (to the nearest pixel) and the radius increment to the next circle or the distance from one line to the next, are displayed in the field next to the slider.

5.11.10 Point Entry Target

The Point Entry target enables you to specify multiple points in the Image window without bringing them to the center of the FOV and without pressing the Enter button on the joystick to accept the points.

When you select the target it appears as a standard Windows cursor with a small crosshair at the tip of the cursor. You can change the color of the small crosshair in the color palette; however, you cannot change the color of the Windows cursor itself.

To use the target, see the following topics:

- How to Use the Point Entry Target
- How to Use the Point Entry Target in Run Mode

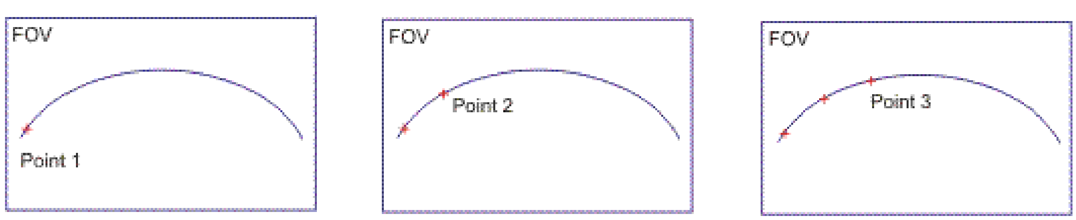
> 🗎 If you want to change the total number of points in the step, use the Change Step function; it is not possible to do this during step edit.

How to Use the Point Entry Target

When the Point Entry target is selected and the feature is displayed in the Image window, you can capture and accept the location of each individual point comprising the feature.

To specify a point, place the Point Entry target on the desired point and press the left mouse button. A small cross indicates the actual location of the selected point. The colors of the small crosses are the same as the color of the Point Entry target itself.

Create Mode



All points within the FOV continue to be displayed until the feature is complete. If you move the stage, the points track within the Image window, relative to current stage location, until moved outside of the FOV.

If the feature is located partially outside the FOV, you can select the points within the current FOV, and then move the stage with the joystick and continue to take points at the new stage location.

When you edit a step where the entire feature does not fit in the FOV, the software displays only those points that fit in the current FOV. When you move the stage from the original stage location of that step, points that were not visible previously will begin to show in the Image window as you move the stage.

- If you want to delete points, click the **Back** arrow in the step to remove points sequentially.

- If you want, you may change the stage locations for each specified point in the Edit window.

- To change the target color of all the points in an existing manual point entry step while creating or editing a routine, you may do one of the following:
  - Change the color and add another point.
  - Delete the last point, change the color, and add another point.

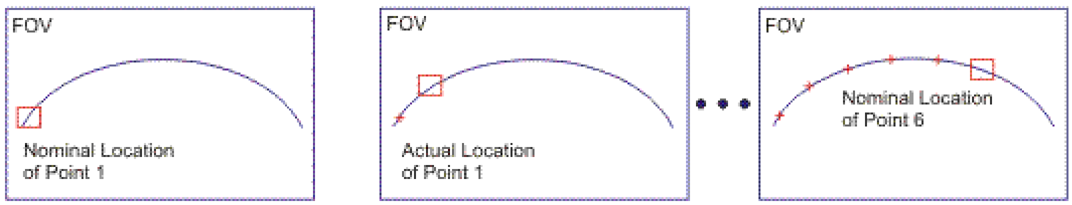
- To change the target color of all the points in an existing manual point entry step while step editing a routine, you need to change the target color first and then add another point.

How to Use the Point Entry Target in Run Mode

When a step measured with the Point Entry target is encountered in a step during Run mode, the software:

- Positions the stage where it was during step creation, allowing you to click the feature in the Image window to retake the actual points.
- Displays a small box over the nominal location of the first point that you are expected to click. The color of the small box is the same as the color of the Point Entry target itself.

Run Mode (only points that have been "picked" are drawn)



As you click in the center of the box, a small cross appears where the actual point is located. The software then moves the small box over the nominal location of the next point in the sequence. You will need to specify the same number of points that were specified during Create mode.

If any points are outside the FOV or they were created at a different XYZ stage location, the software moves the stage automatically, displays the feature at the programmed stage location, and continues to display the "picked" points at the programmed actual locations (as long as they are located within the new FOV). The software then displays a small box over the nominal location of the first point in the new FOV that you are expected to click.

🗎 If you want to delete points, click the **Back** arrow in the step to remove points sequentially.

5.11.11 Manual Focus Target

The Manual Focus target is used to automatically focus an edge or surface. It cannot be used to measure a point. The focus tool looks at image data from several Z axis positions and calculates the Z axis position that would yield the best focus, or highest contrast. You perform an edge focus when using the back light, and a surface focus when using the surface lights, and a grid focus using the grid light. You can also use laser focus to focus on a surface.

The image must be approximately in focus before you can start an autofocus. If it is not, focus the image manually by rotating the Z-axis focus knob on top of the joystick. The image must have sufficient contrast in order for the focus to work.

To select the Manual Focus target, click  in the toolbox. The target appears as a box in the center of the Image window.

You can change the size of the focus tool to any desired rectangular size. The maximum size is 632 pixels. The minimum size is 25 pixels. To change the size:

- Select any edge or corner and hold the left mouse button while "dragging" the edge or corner to the desired size.
- Use the Width and Height sliders or type a pixel value in the Autofocus Settings window.

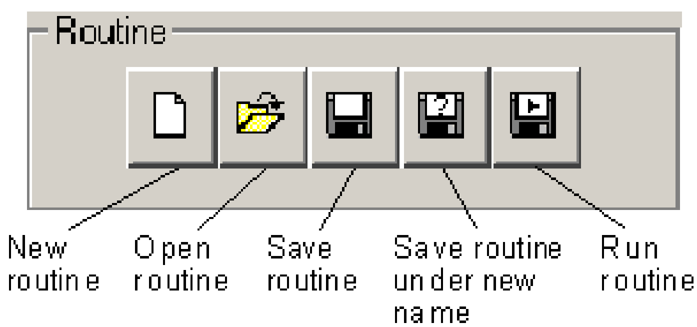
You can also change the color of the target by clicking the Color Palette icon in the Image window toolbar.

This page was left blank intentionally.

6. Creating, Opening, & Saving a Routine

6.1 Overview

The eBx software enables you to create and run automated inspection routines. As you create a routine, each measurement is saved as a step. The software "remembers" the steps and the sequence in which they were done. When you complete the measurements you can save them as an inspection routine. You can then run the routine to measure additional parts automatically.



You click the appropriate Routine icon in the toolbox or select the following functions from the **File** menu:

- Start a new routine
- Open an existing routine
- Delete a routine file
- Save the current routine
- Save the current routine under a new name
- Print a routine listing
- Run the current routine
- Stop the routine

6.2 Understanding File Management

When you first create an inspection routine as you measure features, the measurement steps reside in active memory. If you turn the machine off or start another routine without first saving the current routine, any measurements in active memory are lost.

You can save each inspection routine as a file on any of the computer's drives. For example, a routine can be named FASTRT01.BXI. Two elements make each file unique:

- File name, consisting of alphanumeric characters. You can use both upper and lower case letters.
- File type, consisting of three letters. The default file type for inspection routines is BXI. The use of other extensions is not recommended so that you do not load an incorrect file type by mistake.

When you name a file, you must also specify a location indicating where to store or find the file on the disk. The location consists of two elements:

- Folder (and/or subfolder), which is arranged in a hierarchy from general to specific. Each name in the folder is preceded by a backslash (\).
- Drive. The computer typically has a hard disk drive (C:), and may also have a floppy drive (A:), other hard drives, or network drives.

For example, to save file FASTRT01.BXI on (or get it from) the Routines folder on the hard disk, you would use FASTRT01.BXI and the C:\QVSI\BASICX\ROUTINES folder.

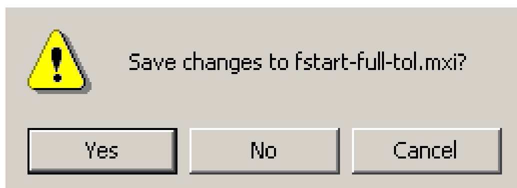
The folders, subfolders, and files are typically displayed in numerical, then alphabetical order.

6.3 Start a New Routine

Starting a new routine clears any measurements (or routine) that are in active memory. It does not clear the part setup zero and part setup alignment settings. This allows you to create a new routine using the same setup.

To start a new routine, click  or select **File** / **New** from the main menu.

If a routine is in active memory, and it was changed or has not been saved, the software displays a confirmation prompt indicating that the current data will be lost.



- Click **Yes** if you want to save changes to the current routine and start creating a new routine.
- Click **No** if you want to clear any routine in memory and start creating a new routine.
- Click **Cancel** to quit this function.

Tip If you want to clear the part setup zero and part setup alignment settings and initialize the XYZ offsets from stage home (scale zero), use System / Reset.

6.4 How to Open an Existing Routine

This function retrieves the specified routine from the hard disk or diskette and puts it into active memory so that you can run it or change it. If the routine is read-only, you can run it but not save it.

To open an existing routine:

1. Click  or select **File** / **Open** from the main menu. A dialog box displays the routine (BXI) files in the folder that was used last. Navigate to the desired folder, if different.
2. Specify the desired routine in one of the following ways:
   - Type the name of the routine in the File Name field and click **Open** (or press Enter on the keyboard).
   - Scroll through the list until you find the routine. Then highlight the routine name and click **Open** (or press Enter on the keyboard), or double-click the routine name.

Once the routine is in active memory you can use it to measure additional parts or make changes to the routine by adding measurements or using the Edit function.

When you open a routine:

- It is loaded with the most current datum settings in the routine. This allows you to make additional measurements on the part without having to run the routine to establish the current datums. However, your manual setup must be correct.
- The system internally recalculates the dimensions of each feature using the nominal point locations from the creation of the routine. The recalculated dimensions appear in the print listing, and the features displayed in the Model window are based on the recalculated dimensions.

6.5 Import File

SmartCAD creates a file with an MXT extension, which can be imported into eBx to create a routine.

To import SmartCAD files:

1. Select **File** / **Import** from the main menu. A window pops up displaying the SmartCAD (MXT) files in the folder that was used last. Navigate to the desired folder, if different.
2. Specify the desired file in one of the following ways:
    - Type the name of the file in the File Name box and click **Open** (or press Enter on the keyboard).
    - Scroll through the list until you find the file. Then highlight the file name and click **Open** (or press Enter on the keyboard), or double-click the file name.

6.6 Export File

A CAD file contains a template drawing of a part. This function exports the part routine to a CAD file.

To export a CAD file:

1. Select **File** / **Export** from the main menu. A dialog box displays the files in the folder that was used last. Navigate to the desired folder, if different.
2. Select the file format, e.g., DXF or IGS, if it is not already selected.
    - The software will output the following features to a DXF file: Point, Line, Circle, Midpoint, Intersection, Centroid as a Point, and a Contour as a series of points.
    - The software will output the following features to an IGS file: Point, Line, Circle, Sphere, Cylinder, Cone, and Plane. It also outputs an Intersection as a Point, Midpoint as a Point, a Centroid as a Point, and a Contour as a series of points.

3. Type the name of the file in the File Name field and click **OK** (or press Enter on the keyboard).

6.7 Save the Current Routine

This function allows you to save all your measurements as a routine.

To save the measurements:

1. Click  or select **File** / **Save** from the main menu. If you are creating a new routine and this is the first time you select Save, a dialog box displays the routine (BXI) files in the folder that was used last. Navigate to the desired folder, if different.
2. Type a unique name for the routine and click **Save** or press **Enter** on the keyboard.

If you specify the name of an existing routine, the software displays a confirmation message.

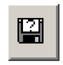
- Click **Yes** to replace the existing file.
- Click **No** to return to the dialog box to specify another file name.

If this is an existing routine, this saves all the measurements done since the file was opened or saved the last time.

 Save your measurements often as you create a routine.

6.8 Save the Current Routine Under a New Name

To save the current routine under a new name:

1. Click  or select **File** / **Save As** from the main menu. A dialog box displays the routine (BXI) files in the folder that was used last. Navigate to the desired folder, if different.
2. Type a unique name for the routine. You can also change the folder if you want. This copies the routine in active memory and saves it in the designated folder under a new name.

If you type the name of an existing routine, a confirmation prompt asks whether you want to overwrite the file or type a new name. Click the desired button.

6.9 How to Delete an Existing Routine

This function deletes the specified routine from its current location. You can delete any kind of file, for example, routines and print files.

To delete an existing routine:

1. Click or select **File** / **Delete** from the main menu. A dialog box displays the routine (BXI) files in the folder that was used last. Navigate to the desired folder, if different.
2. Specify the desired routine in one of the following ways:
   - Type the name of the routine in the File Name field and click **Open** (or press Enter on the keyboard).
   - Scroll through the list until you find the routine. Then highlight the routine name and click **Open** (or press Enter on the keyboard), or double-click the routine name.
   - The software displays a confirmation box to verify that you want to delete the file.
3. Click **OK** (or press Enter on the keyboard) to delete the file.

It is strongly recommended that you use this function to delete only those kinds of files that you created with the measurement software. Do not delete any files that are not related to the routine. Also, you may want to back up the files that you plan to delete.

You cannot delete write-protected files. The software displays a message if you try to delete such a file.

You can also delete the file using the Windows File Manager function.

7. Entering Nominal Values & Tolerances

7.1 Overview

Nominal values refer to the ideal dimensions of a feature. *Tolerances* are values that indicate how much the dimensions can vary.

Click the **Tolerances** tab in a measurement step to enter and change the nominal and tolerance values.

When you measure a feature, the software compares the actual measurements with the specified nominal values by subtracting the nominal value from the actual value. The amount that the actual value differs the nominal value is given as the deviation.

Once the deviation is calculated, the software checks if the actual value is within the specified tolerance range.

- If the actual value exceeds the tolerance range (it is out of tolerance):
  - A red Fail indicator (see *Pass / Fail Indicators*) is displayed next to the measured value in the Measurement Results window.
  - An amount is shown in the Exceeded column on the printout.
- If the actual value is within the tolerance range:
  - A green or yellow Pass indicator is displayed next to the measured value in the Measurement Results window.
  - A representation of the percentage within the upper or lower tolerance is shown in the form of pluses (value is above the nominal) and minuses (value is below the nominal). Each plus/minus represents 25 percent of the tolerance.

The example here shows a sample printout and how the percentage is calculated and displayed. The printout shows two minuses in the Exceeded column for a diameter measurement. This indicates that the percent of the tolerance used (36.8%) is between 26 and 50 percent of the lower tolerance.

```
======================================================================
Feature   Unit    Nominal   Actual     Tolerances     Deviation     Exceeded
======================================================================

Step 6 - Datum -C- Hole
Diameter   in   +00.25000 +00.24816  +0.0050  -0.0050 -00.00184       --
```

Calculation of the tolerance percentage $\frac{\text{Deviation}}{\text{Tolerance}}$ $\frac{0.00184}{0.00500}$ = 36.8% (displayed as two minus signs)

If the measurement results are output, the software also outputs the nominal values, tolerances and deviation results.

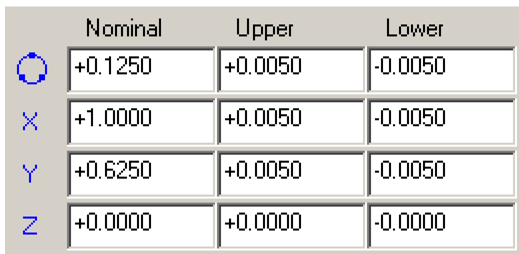
If the measurement results are sent to the data collector or stats file, it sends the data that is specified in the Stats Output templates.

- Nominal feature dimensions are not the same as nominal point locations.
- Nominal feature dimensions are the ideal dimensions of a feature as described above.
- Nominal point locations are the ideal coordinates of each point that make up a feature. The coordinates of each point are used to calculate the dimensions of a feature. You can edit the nominal point locations by clicking on the Edit Points button in the Measurement window and changing the values. The edited values become the nominal location that the stage will be driven to when the routine is run.

7.2 How to Enter Nominal Values and Tolerances

You need to enter nominal values and tolerances in order to determine the quality of the part. You can enter them either before or after you measure a feature. We recommend that you measure the feature first and then enter the nominal values and tolerances.

Click the **Tolerances** tab in a measurement step to enter and change the nominal and tolerance values.



You can enter and change the nominal and tolerance values in the following ways:

- Use the mouse to place the cursor in a box.
  - While holding the left mouse button down, drag the mouse over the desired digits to highlight them. After releasing the button, type the new digits. This overwrites the highlighted digits.
  - Enter new values at the cursor location. They are inserted in front of existing values. This is used most often for editing and changing existing values.
  - Double-click the left mouse button to highlight the entire box and to replace the value with the new entry.
- Use the Tab key to move forward from one box to the next and [Shift] + [Tab] to move the cursor backward. When the tab key is used the entire box is highlighted and the entire value in the box is replaced. This is used most often for first-time entry.

7.3 Tips for Entering Nominal Values and Tolerances

When entering nominal values and tolerances, keep the following in mind:

- You do not need to enter leading zeros (i.e., to the left of the decimal point) unless you are making changes.
- It is not necessary to enter the positive (+) sign. However, a negative (-) sign is necessary when required.
- You do not enter signs for ANSI tolerance values. However, they appear on the printed report (upper is +; lower is -).

If you do not enter nominal and tolerance values for a feature, no deviations are reported. If you want to specify zero as a nominal value or tolerance, go to the Tolerances window, enter the values, and click OK. This will accept the zeros as valid nominal and tolerance values.

You can change the nominal and tolerance values at any time.

For similar features, the system "carries forward" the previous tolerance information from one feature to the next, as described under the topic Understanding How Tolerances Are Carried Forward.

When you do the measurement first, the software inserts the actual measured values in the nominal feature size and location fields (it does not enter any values in the tolerance fields). This allows you to see the signs and enter the correct signs for the nominal values. For example, if the measurement results show an angle that is negative (for example, -45°), you must also enter the nominal as a negative value. This also reduces the amount of typing because you only need to change some of the values rather than enter all of them.

7.4 Understanding How Tolerances Are Carried Forward

When you measure a feature, the software automatically inserts the actual measured values in the nominal feature size and location fields. However, it does not enter any values in the tolerance fields. You need to enter the tolerance values manually.

When you enter the size, location and form tolerances for a feature, the software "carries forward" the tolerances to the next measurement of the same feature. That is, the software automatically uses the same tolerances that were used in the last identical or similar type of feature.

For example, the QVI training part has three holes (circles) along its outer edges; the tolerances for these holes are the same. After you measure the first hole using the Circle function and enter the tolerances, the software "remembers" the tolerances you entered. It carries them forward when you use the Circle function to measure the other holes. You do not need to enter any tolerances for the other holes.

The software also carries tolerances for similar types of features. Several examples are listed below.

- If you used Measure Line, tolerances are carried forward for Construct Line.
- If you used Measure Circle, tolerances are carried forward for Construct Circle.

> You can change tolerances at any time. You do not have to use what is carried forward.

7.5 Geometric Tolerances

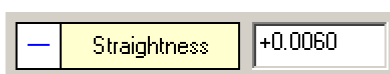
The following tolerances are displayed with the measurement results of a feature after you specify the applicable nominal and tolerance values.

- Position tolerances are displayed for point, circle, midpoint, sphere, intersection, and width measurements.
- Form tolerances are displayed for line, circle, plane, and sphere measurements.
- Orientation tolerances are displayed for line measurements, including reference lines in a width.
- Profile tolerances (upper and lower) are displayed for line, circle, sphere, and plane measurements.
- Concentricity tolerances are displayed for circle and sphere measurements.

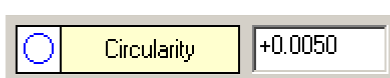
7.5.1 Form Tolerances

Geometric tolerances of form apply to single features or elements of single features such as line, circle, plane, sphere, cone, or cylinder.

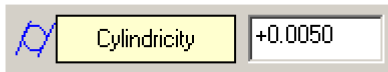
- **Straightness** is a condition where an element of a surface or an axis is a straight line. A straightness tolerance specifies a tolerance zone, e.g., defined by two parallel lines, within which the line must lie.



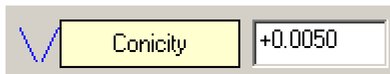
- **Circularity** (roundness) is a condition where all points of a circular element such as a circle or sphere, are equidistant from a common axis or center. A circularity tolerance specifies a tolerance zone, e.g., defined by two concentric circles, within which the circle must lie.



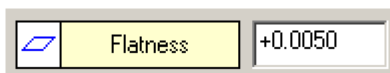
- **Cylindricity** is a condition on a surface of revolution in which all points of the surface are equidistant from the centerline axis. If a cylindricity tolerance is specified, it defines a tolerance zone bounded by two concentric cylinders within which the surface must lie.



- **Conicity** is the region between two similar, perfect, and coaxial cones (inscribed cone and circumscribed cone) that are some distance apart and within which the entire conical part lies.



- **Flatness** is the condition of a surface having all elements in one plane. A flatness tolerance specifies a tolerance zone defined by two parallel planes within which the plane must lie.



To specify a form tolerance for a feature:

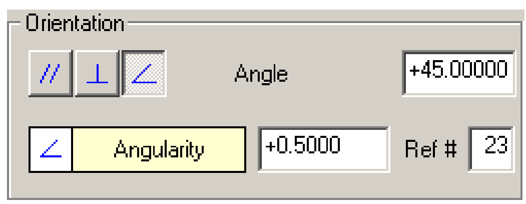
1. Measure or construct the desired feature.
2. Click the **Tolerances** tab.
3. Enter the form tolerance.
4. Click the **Results** tab to check the pass/fail status of the form deviation.
5. Click **OK** to complete this step.

7.5.2 Orientation Tolerances

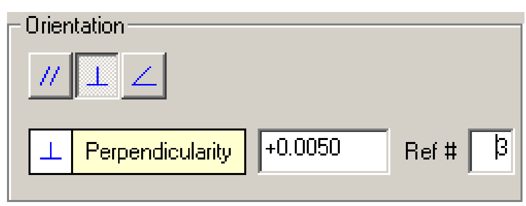
Three geometric tolerances of orientation report the maximum deviation (in the selected units, such as inches) from the perfect form. The tolerances are based on the angle between the line in the current step and a selected reference line.

To specify an orientation tolerance and display the deviation:

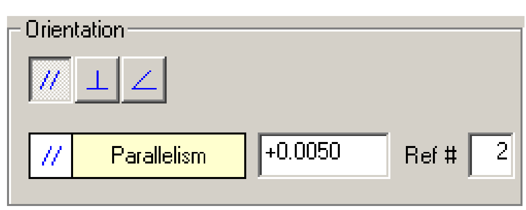
1. Measure or construct a line that can be used as a reference line. For example, use the datum line or the reference feature on the blueprint as the reference line.
2. Measure or construct a line, or construct a width.
3. Click the **Tolerances** tab.
4. Click a button to select the desired orientation tolerance.
   - Angularity. This refers to the basic angle that you specify between the two lines.

   - Perpendicularity. The two lines are nominally perpendicular to each other and the basic angle is 90° (you do not enter the angle).

   - Parallelism. The two lines are nominally parallel to each other and the basic angle is 0° (you do not enter the angle).

5. Enter the tolerance value. If you selected Angularity, also enter the basic angle from the dimension drawing.

6. Specify the reference step by typing the step number or selecting the feature in the Model window.
7. Click the **Results** tab to view the deviation between the actual angle and the basic angle.
8. Click **OK** to complete this step.

> If the lines have different lengths, the order in which the lines are selected may yield different results. It is recommended that you select the longer line first to ensure more accurate results.

7.5.3 Position Tolerance

A true position tolerance is a geometric tolerance of location. It defines a zone within which the center point or axis of a feature is permitted to vary from the nominal location. True position tolerances are applied on the basis of the material condition. You can specify position tolerances for features that have a location, e.g., point, arc or circle, sphere, intersection, width.

To specify a position tolerance and display the position deviation:

1. Measure or construct the desired feature.
2. Click the **Tolerances** tab and enter the nominal values and tolerances for the *location*.

3. Select the appropriate radio button to select the material condition.

    - RFS (Regardless of Feature Size). This is the default. If the blueprint shows a circled S or does not show a circled M or L in the feature control box, the tolerance should be applied RFS. Any deviation in the size of the feature will not change the position tolerance.
    - MMC (Maximum Material Condition). Use MMC if there is a circled M in the feature control box on your dimension drawing or blueprint.
    - LMC (Least Material Condition). Use LMC if there is a circled L in the feature control box on your dimension drawing or blueprint.
    - If you select MMC or LMC, you must also specify whether the feature is an ID (inner dimension) or OD (outer dimension) in order to calculate the appropriate "bonus" tolerance. Bonus tolerances are determined by comparing the feature's actual measured size (up to its size limit) to the MMC or LMC size and adding the difference to the position tolerance as a "bonus." The default is ID.

4. Enter the position tolerance value in the Position Tolerance field. No sign is necessary.
5. Click the **Results** tab to view the position deviation.
6. Click **OK** to complete this step.

> If the true position tolerance has a value other than zero, the X and Y nominal values, actual values and deviations can be printed, even if the X and Y nominal values are zero.

7.5.4 Position Tolerance Calculation

The following example shows how the elements of a position tolerance are calculated. The calculation is based on the actual measurements and the nominal and tolerance values specified for the location.

Calculated true position tolerance in the printout after the measurement is completed

```
==============================================================================
Feature       Unit   Nominal      Actual      Tolerances        Deviation   Exceeded
==============================================================================
 Step 5 - Upper-left hole measurement
Diameter      in   +00.10000  +00.09943  +0.00300  -0.00300  -00.00057    -
X Location    in   +00.00000  -00.00007                      -00.00007
Y Location    in   +01.25000  +01.25015                      +00.00015
Position      in              +00.00033  +0.00743            +00.00033    +
```

1

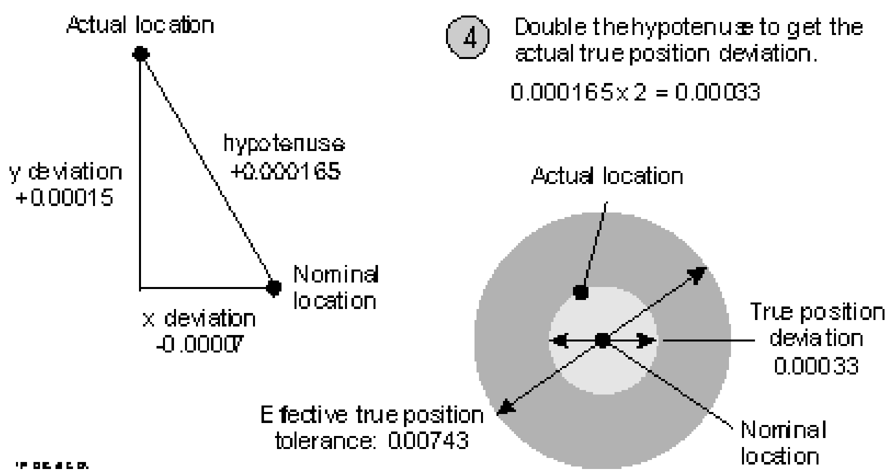
.10000	Nominal
(+) - 0.00300	LowerTolerance
.09700	Maximum Material Condition

2

.09943	Actual Size
- 0.09700	At MMC
.00243	Bonus

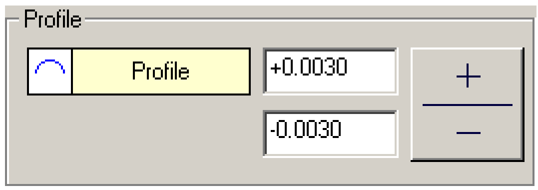
3

0.00500	Basic T.P. Tolerance
+ 0.00243	Bonus
0.00743	Effective T.P. Tolerance



7.5.5 Profile Tolerances

A profile tolerance is a geometric tolerance of size, location and form. It specifies a uniform boundary along the true profile of a feature. You can specify profile tolerances for lines, arcs or circles (shown here), spheres, and planes.



To specify a profile tolerance and display the profile deviation of a feature:

1. Measure or construct the desired feature.
2. Click the **Tolerances** tab and enter the nominal values and tolerances for *size and location*.
3. Enter the profile upper and lower tolerances.

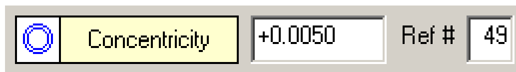
   The profile button (+/-) indicates which side has the positive deviation and which side has the negative deviation. If you want to apply the positive deviation to the other side, click the profile button. This switches the + and - on the button and re-displays the actual profile deviations.
4. Click the **Results** tab to view the profile deviations.
5. Click **OK** to complete this step.

The profile deviations are based on the size and location nominal values of the selected step. The nominal values must be specified correctly to ensure the correct calculation of the profile deviations.

If one number on the blueprint denotes bilateral tolerances, you must split the number in half and apply the values as upper and lower tolerances.

7.5.6 Concentricity Tolerance

A concentricity tolerance is a geometric tolerance that specifies a cylindrical tolerance zone whose axis coincides with a datum axis and within which all cross-sectional axes of the feature being controlled must lie. The specified tolerance and the datum reference apply only on an RFS basis.



1. Measure or construct a circle or sphere that can be used as a reference feature.
2. Measure or construct a circle or sphere.
3. Click the **Tolerances** tab.
4. Enter the tolerance value.
5. Specify the reference feature by typing the step number of the reference feature or by selecting the feature in the Model window.
6. Click the **Results** tab to view the deviation between the current feature and the reference feature.
7. Click **OK** to complete this step.

7.5.7 Tolerance Standards

Geometric tolerances in the Tolerances window and on a printed report are based on one of the following standards:

- ANSI (ASME) Y14.5M. The upper tolerance is indicated by a plus (+) sign; the lower tolerance is indicated by a minus (-) sign. These signs are entered automatically. You cannot enter the signs.
- ISO. These tolerances apply to size and location features. The default upper tolerance is indicated by a plus (+) sign; the default lower tolerance is indicated by a minus (-) sign. To specify High Upper/Low Upper (+/+) tolerances and Low Lower/High Lower (-/-) tolerances, you must enter the signs in the respective fields.

To select the desired standard, select the ANSI or ISO radio button. The selected tolerance is applied to all dimensions of the feature. You can change the tolerance standard for any step.

🖹 Form tolerances (e.g., roundness, straightness, flatness) and position tolerances are only positive.

🖹 Profile tolerances for lines, arcs, circles, and planes have a lower and upper tolerance.

🖹 Plane profile +/- tolerances are signed as positive and negative.

🖹 The default tolerance standard is ANSI. This is based on a value of 0 for the USE_ISO_TOLERANCING parameter in the INPUT.INI file. A non-zero value indicates the use of ISO tolerance standards.

8. Measure Functions

8.1 Overview

This section describes how to measure the following:

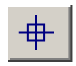
- Point
- Line
- Circle
- Distance
- Plane
- Midpoint
- Contour
- Sphere
- Cone
- Cylinder
- Min/Max/Avg

The Measure Sphere, Measure Cone, and Measure Cylinder functions are only available if the system is equipped with a touch probe.

8.2 How to Measure a Point

You can measure a point with the weak edge point tools, the strong edge tool, laser tool, and any of the manual targets.

Use this procedure to measure a point on an edge with the Strong Edge tool.

1. Click  in the toolbox to select the Strong Edge tool. The system displays the tool in the Image window.
2. [Classic User Interface] Click the **Measure** tab.
3. Click in the toolbox or select **Measure** / **Point** from the main menu.
4. Click the desired point on the edge.

   The software automatically moves the stage so the point appears at the intersection of the crosshairs. Then it performs an edge analysis.
5. Press **Enter** on the joystick to accept the point.

   Measurement results appear in the Point Measurement Results window.
6. If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.
7. Click **OK** to complete the measurement.

8.3 Locate Point

The Locate Point function enables you to define a location and elevate the camera or move the stage to that location without measuring anything. For example, it can be used to move the stage out of the way to provide access for changing parts at the end of an inspection routine.

To use this function, follow the steps below.

1. Select a manual alignment target.
2. Click in the toolbox or select **Measure** / **Point** from the main menu.

   The software displays the Point Measurement Results window.
3. Move the stage to the desired location and press **Enter** on the joystick to accept the location of the point.
4. Select the **Locate Point** check box to activate it.
5. Click **OK** in the Measurement window.

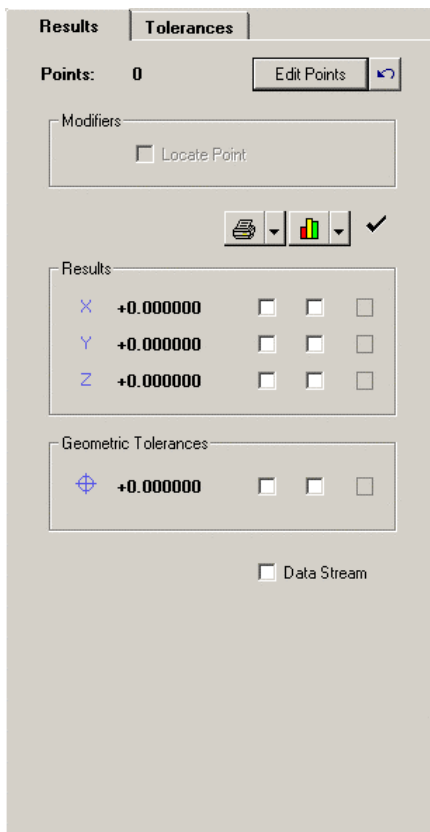
When you run the routine, the software goes to the specified location but it does not measure the point.

> The Locate Point function can be used only with the manual alignment targets. It cannot be used with the automatic edge measurement tools.

8.4 Point Measurement Results

A point is a single location.

- The number of points used in the measurement (you can edit or delete the entered data point)
- Check box for Locate point, which enables you to define a specific location (if the point was measured)
- XYZ coordinates of the point. If the point is constructed from a feature such as a circle or line, the XYZ coordinates indicate the center or midpoint of the selected feature
- Position tolerance
- Check box for data stream output (if the point was measured)

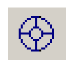


If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.

8.5 How to Measure a Line

You can measure a line or straight edge with any tool or manual target, except the weak edge point tools.

Use this procedure to measure a line with FeatureFinder, which is the tool used most often. You must specify a start and end point on the edge.

1. Click  in the toolbox to select the FeatureFinder tool.
2. Position the FeatureFinder tool on the first point on the edge where you want to start the measurement.
   - To specify the starting point and measure the line automatically, double-click the starting point. The system automatically determines whether the feature is a line or a circle, and measures it. This works best when the edge is of high contrast.
   - To measure a straight line, press and hold the left mouse button to indicate the location of the first point. While holding down the left mouse button, drag the line along the edge and release the button when you are at the second point. This completes the line measurement.
   - The software automatically performs an edge analysis and displays the search area and the points that it found within that area in the Image window. The measurement results appear in the Line Measurement Results window.
3. If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances. You can also change the weak edge settings and remeasure the circle.
4. Click **OK** to complete the measurement.

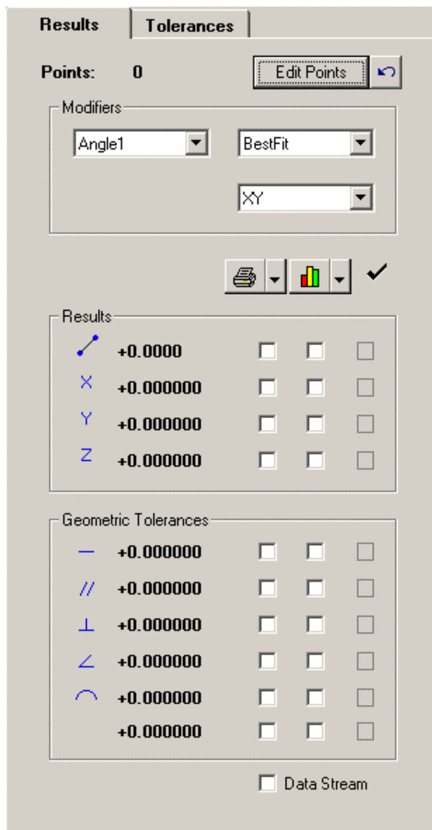
You can also measure the line using other tools such as Strong Edge.

Tip If you are measuring a line that also goes into a curve, do not use points that are close to or on the curve. This may cause inaccurate measurement results.

8.6 Line Measurement Results

The Measure Line function reports the orientation, location and form of a line or straight edge as calculated from the entered data points.

Measurement results appear in the Measurement window when at least two points or features are entered.



The Measurement window displays:

- The number of points or features used in the measurement (you can edit or delete the entered points or features)
- Modifiers, which allow you to select how the line is calculated:
  - **Angle or supplementary angle**, relative to the currently defined X axis
  - Line calculation type. Select the desired type from the drop-down list

- Coordinate plane in which the angle is reported

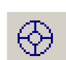
- Results, which include the **angle** and the XYZ coordinates of the **midpoint** of the line
- Geometric tolerances, including straightness, parallelism, perpendicularity, angularity, and profile
- Check box for data stream output

If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.

8.7 How to Measure a Circle

You can measure a circle with any tool or manual alignment target, except the weak edge point tools.

Use this procedure to measure a circle with FeatureFinder, which is the tool used most often. When using FeatureFinder, you must be very precise in placing each point **on** the desired edge and be sure that the points are spaced evenly from one other.

1. Click  in the toolbox to select the FeatureFinder tool.
2. Position the FeatureFinder tool at the first point on the edge where you want to start the measurement.
3. Press the left mouse button to display the first point. It will say either Arc or Circle.
   - If it is Arc and you want to measure a circle, click the same spot again to display Circle.
   - If it is Circle and you want to measure an arc, click the same spot again to display Arc.
4. Position the FeatureFinder tool on the second point on the edge and press the left mouse button to display the second point.

5. Position the FeatureFinder tool on the third point on the edge and press the left mouse button to display the last point.

   The software automatically performs an edge analysis and displays the search area and the points that it found within that area in the Image window. The measurement results appear in the Circle Measurement Results window.

6. If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances. You can also change the weak edge settings and re-measure the circle.

7. Click **OK** to complete the measurement.

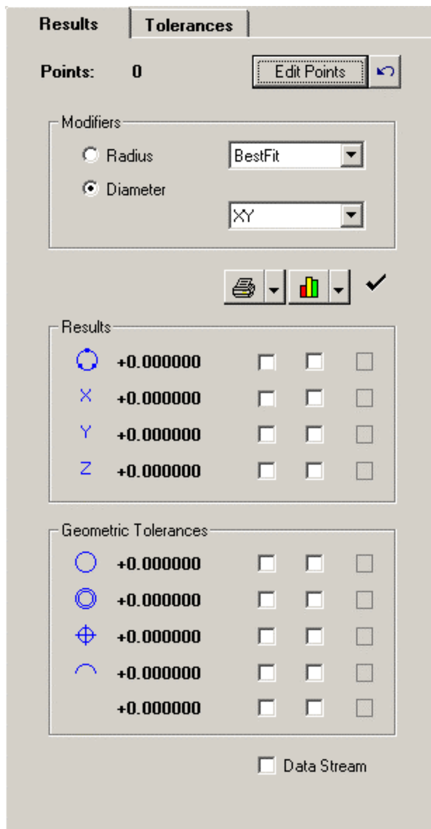
> If you are remeasuring the arc or circle, it is strongly recommended that you do **not** decrease the number of points; doing so may decrease the accuracy.
>
> You can also measure the circle using other tools such as Strong Edge.

Tip The circle function can also be used as an arc to measure rounded or curved edges. If you are measuring a curved edge that blends into a line, do not specify a point that is very close to or on the line; this may cause inaccurate measurement results.

8.8 Circle Measurement Results

The Measure Circle function reports the size, location, orientation and form of a circle or arc (rounded or curved edge) as calculated from the entered data points.

Measurement results appear in the Measurement window when the minimum number of required points or features is entered.



The Measurement window displays:

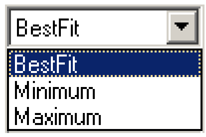
- The number of points or features used in the measurement (you can edit or delete the entered points or features)
- Modifiers, which allow you to select how the circle is calculated:
  - Diameter (default) or Radius
  - Circle calculation type. Select the desired type from the drop-down list.
  - Coordinate plane in which the radius/diameter is reported
  - Results, which include the diameter or radius and the XYZ coordinates of the center of the circle
- Geometric tolerances, including roundness, concentricity, position, and profile
- Check box for data stream output

If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.

8.9 Circle Calculation

When you measure or construct a circle, the Measurement window displays the types of calculations in a drop-down list.

Select the type of calculation from the drop-down list.



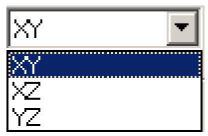
- **Best Fit**. This is the default mode. This is the best-fit circle from all the data points.
- **Minimum Contact**. This returns the largest circle that encompasses all the points such that all the points are on or outside of the circle.
- **Maximum Contact**. This returns the smallest circle that encompasses all the points such that all the points are on or within the circle.

You can measure or construct the points either before or after selecting the type of calculation. You can change the calculation type after measuring the points and view the different results.

8.10 Coordinate Plane

When you measure or construct a circle, measure or construct a line, or measure or construct a contour, the Measurement window displays a drop-down list for the coordinate plane.

To select the coordinate plane in which the radius/diameter or angle will be reported, click the desired coordinate plane: XY Plane (default), XZ Plane, YZ Plane, or XYZ Plane.



The model:

- Displays lines in the XZ or YZ axis planes as a two-dimensional XY line, projected into the XY plane.
- Does not display circles in the XZ or YZ axis planes.

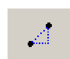
> You can measure the XZ and YZ lines and circles and send the measurements to the printer, but you cannot use them for other constructions.

8.11 How to Measure a Distance

You can measure the distance between two points (typically on edges) with the Strong Edge tool or any manual target. You can also measure a Z distance with the Laser Focus tool.

Use this procedure to measure a distance with the Strong Edge tool, which is the tool used most often. You must specify a start and end point.

1. Click  in the toolbox to select the Strong Edge tool.

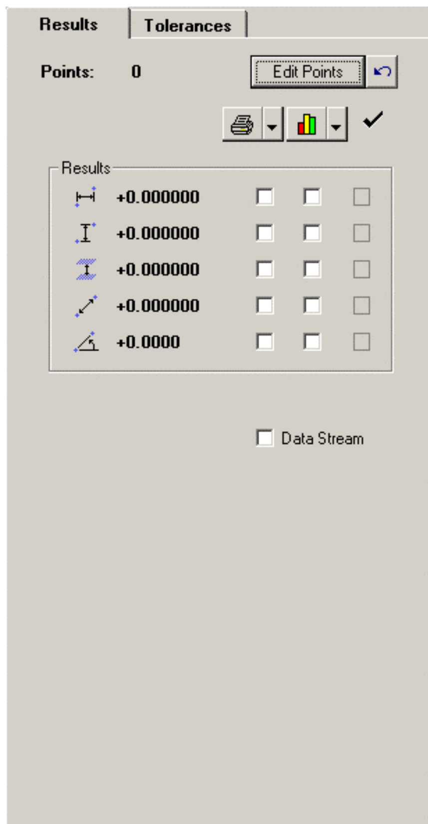
   The system displays the tool in the Image window.
2. [Classic User Interface] Click the **Measure** tab.
3. Click  in the toolbox or select **Measure** / **Distance** from the main menu.
4. Move the stage until the desired edge or surface appears in the Image window.
5. Position the mouse cursor on the desired point on the edge and press the left mouse button.

   The software automatically moves the stage so the point appears at the intersection of the crosshairs. Then it performs an edge analysis.
6. Press **Enter** on the joystick to accept the point.
7. Repeat Steps 3, 4, and 5 for the second point. The measurement results appear in the Distance Measurement Results window.
8. If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.
9. Click **OK** to complete the measurement.

8.12 Distance Measurement Results

The Measure Distance function reports the distance and angular separation between two points. However, the distance is not displayed in the Model window.

Measurement results appear in the Measurement window when the two points are entered.

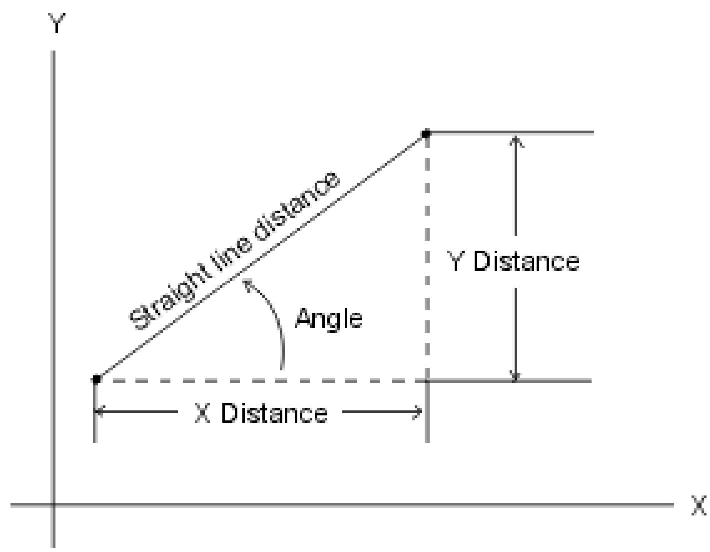


The Measurement window displays the results in the following order:

- The number of points or features used in the measurement
- **X distance** from one point to the other
- **Y distance** from one point to the other
- **Z distance** from one point to the other
- **Straight line distance** between the two points
- **Angle** of the straight line between the two points, based on 0° (the positive X axis)

- Check box for data stream output

If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.

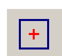


8.13 How to Measure a Plane

You can measure a plane with the Focus tool, laser tool, or a touch probe. This ensures that the Z-axis coordinate is measured.

Use this procedure to measure a plane with the Basic Focus tool. You must enter a minimum of three points to display the plane angle; a minimum of four points is required to display the flatness and profile.

> The number of points and their distribution may affect the results. For best results, make sure the measured points are spread as evenly as possible around the plane.

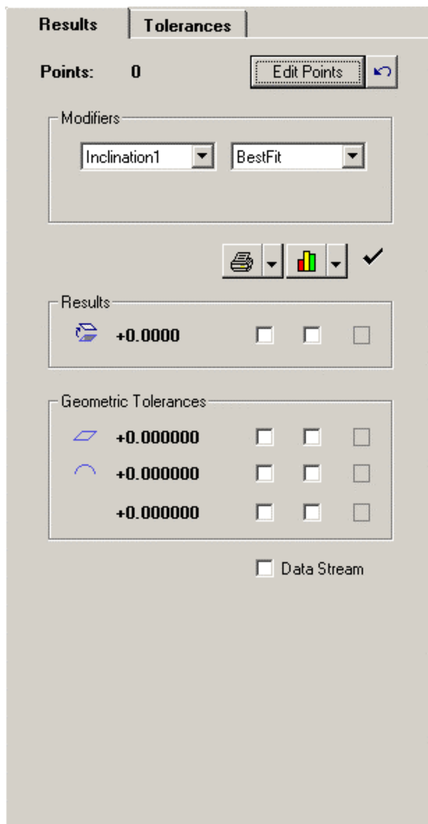
1. Click  in the toolbox to select the Focus tool.

   The system displays the tool in the Image window.
2. [Classic User Interface] Click the **Measure** tab.
3. Click in the toolbox or select **Measure** / **Plane** from the main menu.
4. Turn the backlight off and turn the surface light on.
5. Move the stage until the first surface point that you want to focus on appears in the Image window.
6. Focus the surface manually by twisting the focus knob on the joystick.
7. Click in the Image window to perform an autofocus on the surface.
8. Press **Enter** on the joystick to accept the focus point.
9. Repeat Steps 5 through 8 for the other focus points. The measurement results appear in the Plane Measurement Results window.
10. If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.
11. Click **OK** to complete the measurement.

8.14 Plane Measurement Results

The Measure Plane function reports the orientation, form and location of a plane, as calculated from the entered data points. However, this function does not return a location. In the Model window you cannot tell which points were used to measure the plane.

Measurement results appear in the Measurement window when the minimum number of required points or features is selected.



The Measurement window displays:

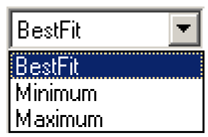
- The number of points or features in the measurement.
- Inclination angle of the plane. You can select and display the negative value of Angle 1 from the drop-down list.
- Plane calculation type. Select the desired type from the drop-down list.
- Geometric tolerances:
    - Flatness. This indicates the largest difference in the Z axis between entered points, measured perpendicular to the plane (whether minimum, maximum or best fit).
    - Profile +. This indicates the perpendicular distance from the measured plane to the point that is farthest above the plane. This is 0 when you use Maximum Contact because there are no higher points.
    - Profile -. This indicates the perpendicular distance from the measured plane to the point that is farthest below the plane. This is 0 when you use Minimum Contact because there are no lower points.
- Check box for data stream output

You can measure or construct the points either before or after selecting the type of plane calculation. You can change the calculation type after measuring the points and view the different results.

8.15 Plane and Line Calculation

When you measure or construct a plane or measure or construct a line, the Measurement window displays the types of calculations in a drop-down list.

Select the type of calculation from the drop-down list.



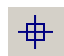
- **Best Fit**. This is the default mode. This is the least-square calculation of the best-fit plane or line from all the data points.
- **Minimum Contact**. This returns the plane or line constructed from three points such that all other points are above those three points.
- **Maximum Contact**. This returns the plane or line constructed from three points such that all other points are below those three points.

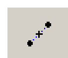
You can measure or construct the points either before or after selecting the type of calculation. You can change the calculation type after measuring the points and view the different results.

8.16 How to Measure a Midpoint

You can measure the midpoint between two points with the Strong Edge tool or any manual target.

Use this procedure to measure a midpoint with the Strong Edge tool. You must specify a start and end point.

1. Click  in the toolbox to select the Strong Edge tool.

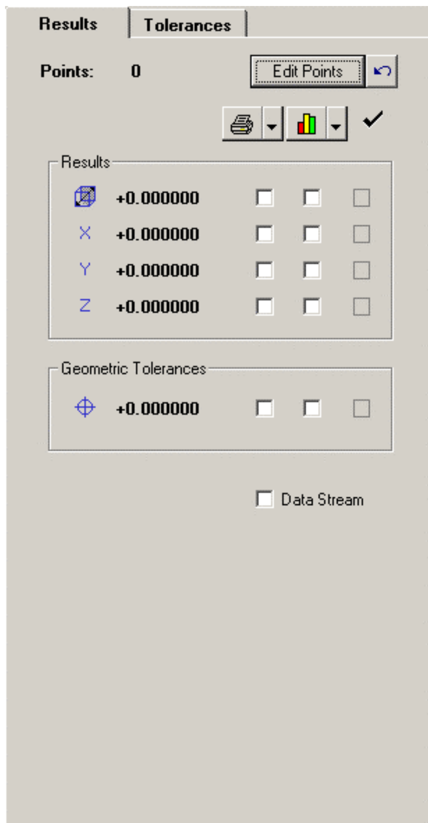
   The system displays the tool in the Image window.
2. [Classic User Interface] Click the **Measure** tab.
3. Click  in the toolbox or select **Measure** / **Midpoint** from the main menu.
4. Move the stage until the desired edge or surface appears in the Image window.
5. Click the desired point on the edge.

   The software automatically moves the stage so that the point appears at the intersection of the crosshairs. Then it performs an edge analysis.
6. Press **Enter** on the joystick to accept the point.
7. Repeat Steps 4, 5, and 6 for the second point. The measurement results appear in the Midpoint Measurement Results window.
8. If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.
9. Click **OK** to complete the measurement.

8.17 Midpoint Measurement Results

The Measure Midpoint function reports the 3D distance and midpoint between two points. The distance is three-dimensional; the calculations are based on all three axes. It is displayed as an asterisk in the Model window.

Measurement results appear in the Measurement window when the two points are entered.



The Measurement window displays:

- The number of points used in the measurement.
- **3D straight line distance** between the two entered points
- **XYZ coordinates** of the midpoint
- Position tolerance
- Check box for data stream output

8.18 How to Measure a Contour

You can measure the location and size (length and area) of an irregular feature that has a smooth edge using the Edge Trace tool, any of the manual alignment targets, the laser tool, or the strong edge tool.

To measure a contour, you must specify at least two points.

Use the following procedure to measure a contour with the Edge Trace tool. You must specify a start and end point.

1. Click  in the toolbox to select the Edge Trace tool.

   The system displays the tool in the Image window.

2. [Classic User Interface] Click the **Measure** tab.

3. Click  in the toolbox or select **Measure** / **Contour** from the main menu.

4. Move the stage until the desired edge appears in the Image window.

5. Measure the feature with the Edge Trace tool.

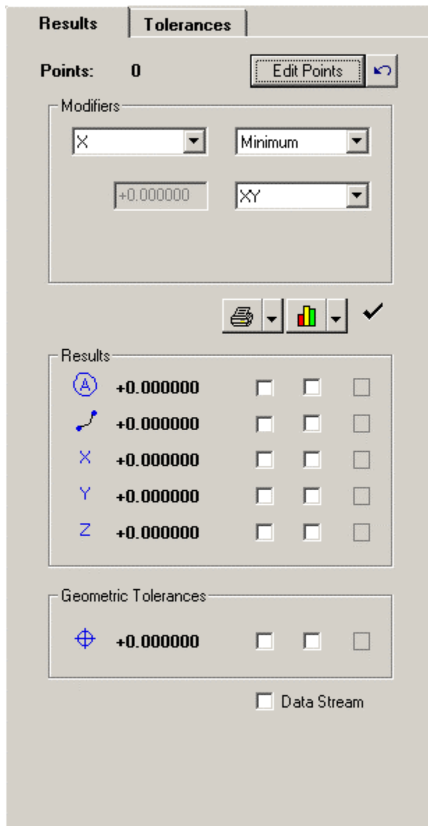
   The measurement results appear in the Contour Measurement Results window.

6. If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.

7. Click **OK** to complete the measurement.

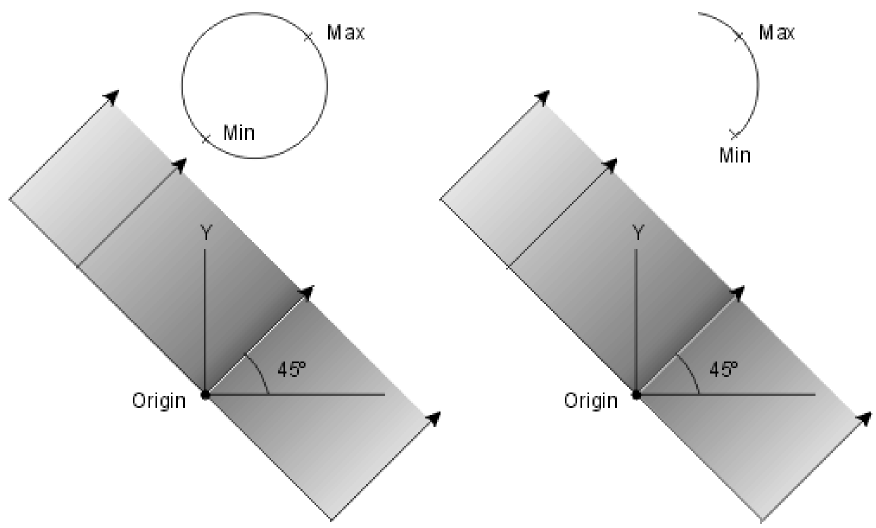
8.19 Contour Measurement Results

The Measure Contour function reports the location and size (length and area) of an irregular feature as calculated from the entered data points.



The Measurement window displays:

- The number of points or features used in the measurement (you can edit or delete the entered points or features)
- Location drop-down list
  - You can select X, Y, Z, Angle, Centroid, or Radius.
  - If you select Angle, the Angle field becomes active to specify the desired angle. If none is specified, the angle is 0 degrees. The system searches for and displays the maximum (highest) or minimum (lowest) XYZ point at the specified angle.



Note: In these examples, the software searches for a point (Min or Max) along a 45° vector. The vector intersects the feature; however, the Max and Min intersection point does not have to be along the same vector.

- Fit drop-down list
  - You can select Minimum or Maximum.
  - This list is disabled if Centroid is selected in the Location drop-down list.
- Coordinate plane in which the contour is reported
- The area of a closed contour
- The length of the contour
- The XYZ location of the closed contour based on the selected modifiers; if Centroid is selected, this is the center location of the mass
- The position tolerance of the closed contour

- Check box for data stream output (if the contour was measured)

If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.

8.20 How to Measure a Sphere

> The Measure Sphere function is only available if the system is equipped with a touch probe.

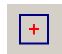
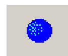
You can measure a sphere optically with the Focus tool and the Strong Edge tool, or by using a laser tool or a touch probe.

To measure a sphere, you must specify at least four points. Use the following guidelines to enter the data points:

- At least one point should be at the top or bottom of the sphere. If the point is measured optically, it should include an autofocus to capture the Z-axis measurement.
- For best results, make sure that the measured points are spread as evenly as possible around the circumference of the sphere.
- To ensure that a hemisphere is displayed in the Model window when Spherical Radius is selected:
  - Make sure that all the points are on the same side of the equator.
  - The first two points and the center location of the sphere determine the plane of the equator.

Use this procedure to measure a solid sphere with the Basic Focus tool and the Strong Edge tool.

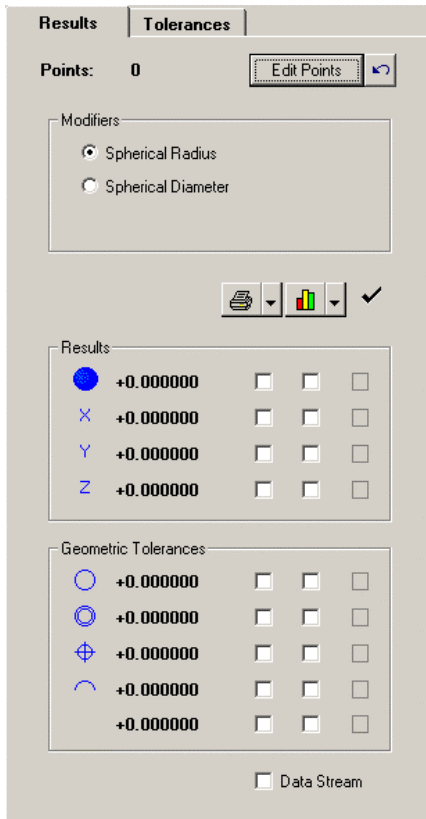
1. Zoom to the lowest magnification and turn the backlight off and turn the surface light on.
2. Move the stage until the top of the sphere appears in the Image window.
3. Zoom to the highest magnification.

4. Move the Z-axis up until you see the granular surface of the sphere.
5. Focus the surface manually by twisting the focus knob on the joystick.
6. Click  in the toolbox to select the Focus tool.
7. [Classic User Interface] Click the **Measure** tab.
8. Click  in the toolbox or select **Measure** / **Sphere** from the main menu.
9. Position the mouse cursor on the surface and press the left mouse button to perform an autofocus.
10. Press **Enter** on the joystick to accept the focus point.
11. Zoom to the lowest magnification and turn the surface light off and the backlight on.
12. Click  in the toolbox or select **Tools** / **Strong Edge** from the main menu.
13. Move the stage until an edge of the sphere is in the center of the tool and, if necessary, adjust the focus manually to bring the edge into focus.
14. Press the left mouse button to measure the edge and press **Enter** on the joystick to accept the point.
15. Repeat Steps 13 and 14 for the other edges. The measurement results appear in the Sphere Measurement Results window.
16. If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.
17. Click **OK** to complete the measurement.

8.21 Sphere Measurement Results

The Measure Sphere function reports the size, location and form of a sphere, calculated from the entered data points.

Measurement results appear in the Measurement window when the minimum number of required points or features is selected.



The Measurement window displays:

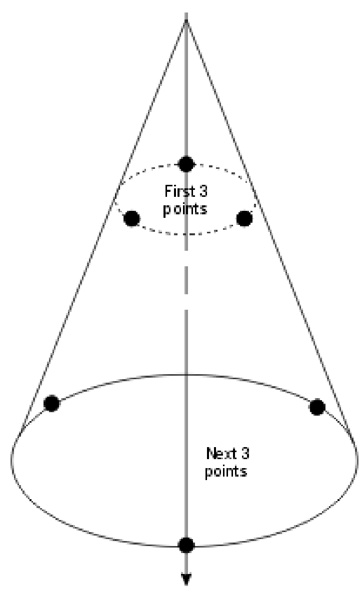
- The number of points or features used in the measurement
- Spherical Diameter and Spherical Radius radio buttons
- **Diameter** or **Radius**, whichever is selected
- **XYZ coordinates** of the center of the sphere
- **Geometric tolerances**, including Circularity, Concentricity, Position, and Profile
- Check box for data stream output

In the Model window, a measured sphere appears as a solid-line circle and a constructed sphere appears as a dashed-line circle.

8.22 How to Measure Cone

The Measure Cone function is only available if the system is equipped with a touch probe.

The cone function reports the orientation, location and size of a cone, calculated from the entered data points. You can measure a cone with the Focus tool, laser tool, or a touch probe.



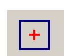
When you measure a cone, you must enter at least six points, in two groups of 3 points each. The order in which the two groups are measured does not matter.

- Measure three points near or at one end of the cone, which defines a circle or arc.
- Measure three points near or at the other end of the cone, which defines a circle or arc.
- After the first six points, you can measure other points between the two circles.
- For more accurate results, increase the distance between the two sets of points and increase the total number of points.

- The vector direction points away from the apex of the cone.
- The line going through the two centers is used to calculate the cone's centerline axis.
- To display a partial cone in the Model window when Radius is selected:
  - Make sure that the paths of the two groups of points do not form complete circles.
  - The first point determines where the start angle begins, then the path goes through the second point, and the last point determines the end angle of the path.

If the points are not entered as specified above, e.g., when you perform a laser scan or touch probe AutoPath, you must specify the nominal values for size, location, XY angle, and elevation before measuring the cone.

Use this procedure to measure a cone with the Basic Focus tool.

1. Click  in the toolbox to select the Focus tool.

   The system displays the tool in the Image window.
2. [Classic User Interface] Click the **Measure** tab.
3. Click  in the toolbox or select **Measure** / **Cone** from the main menu.
4. Turn the backlight off and turn the surface light on.
5. Move the stage until the first surface point that you want to focus on appears in the Image window.
6. Focus the surface manually by twisting the focus knob on the joystick.
7. Position the mouse cursor on the surface and press the left mouse button to perform an autofocus.
8. Press **Enter** on the joystick to accept the focus point.
9. Repeat Steps 5 through 8 for the other two focus points in the first group.
10. Repeat Steps 5 through 8 for the three focus points in the second group. The measurement results appear in the Cone Measurement Results window.

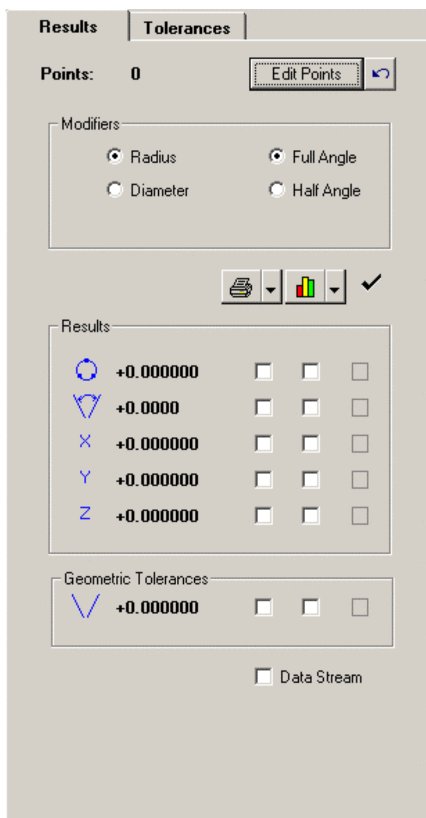
11. If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.
12. Click **OK** to complete the measurement.

A cone is displayed in the Model window as two circles, one each at the top and bottom, connected by lines at the 0, 90, 180, and 270 degree points on the circles. One circle is smaller than the other.

8.23 Cone Measurement Results

The Measure Cone function reports the orientation, location and size of a cone, calculated from the selected features.

Measurement results appear in the Measurement window when the minimum number of required points or features is entered.



The Measurement window displays:

- The number of points or features used in the measurement (you can edit or delete the entered points or features)
- Modifiers, which allow you to select how the cone is calculated:
  - Full Angle or Half Angle of the taper. This value is always positive.
  - Diameter (default) or Radius
- Results, which include best-fit diameter or radius at the center of the cone, angle of the taper, and XYZ location of the midpoint of the centerline axis of the cone
  - If you do not enter any nominal and tolerance values for the diameter and XYZ location, the reported diameter is at the midpoint of the cone’s centerline axis.
  - If you enter a nominal diameter and XYZ location, the reported diameter and XYZ location is at the point on the cone's centerline axis that is closest to the nominal XYZ location.
  - If you enter a nominal diameter of 0, the actual XYZ location is at the apex.
- Conicity tolerance
- Check box for data stream output

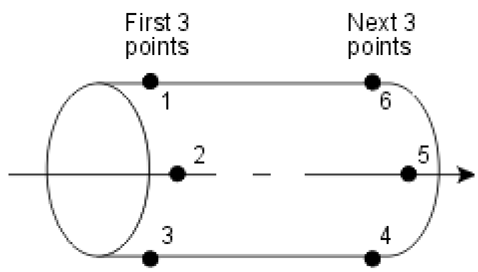
If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.

A cone is displayed in the Model window as two circles, one each at the top and bottom, connected by lines at the 0, 90, 180, and 270 degree points on the circles. One circle is smaller than the other.

8.24 How to Measure a Cylinder

The Measure Cylinder function is only available if the system is equipped with a touch probe.

The Cylinder function reports the size, location, orientation, and form of a cylindrical object, calculated from the entered data points. You can measure a cylinder with the Focus tool, laser tool, or a touch probe.



When you measure a cylinder, you must enter at least six points, in two groups of 3 points each. The order in which the two groups are measured does not matter.

- Measure three points near or at one end of the cylinder, which defines a circle or arc.
- Measure three points near or at the other end of the cylinder, which defines a circle or arc.
- After the first six points, you can measure other points between the two circles.
- For more accurate results, increase the distance between the two sets of points and increase the total number of points.
- The vector direction is from the first set of points to the second set of points.
- The line going through the two centers is used to calculate the centerline of the cylinder.

- To display a partial cylinder in the Model window when Radius is selected:
    - Make sure the paths of the two groups of points do not form complete circles.
    - The first point determines where the start angle begins, then the path goes through the second point, and the last point determines the end angle of the path.

Use this procedure to measure a cylinder with the Basic Focus tool.

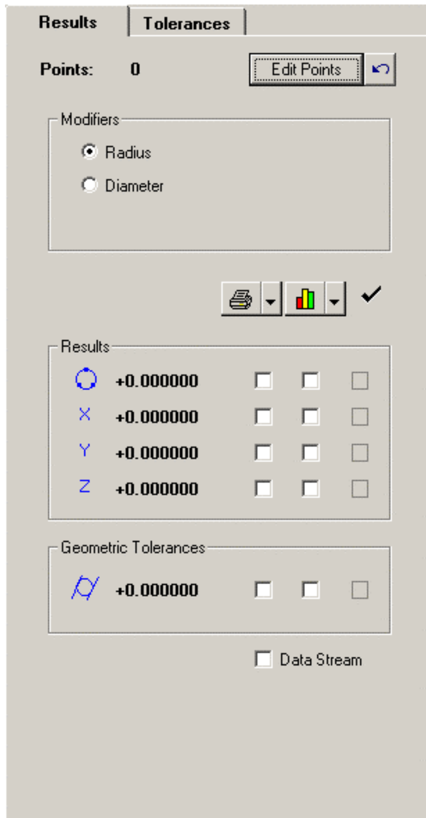
1. Click  in the toolbox to select the Focus tool.

   The system displays the tool in the Image window.
2. [Classic User Interface] Click the **Measure** tab
3. Click  in the toolbox or select **Measure** / **Cylinder** from the main menu.
4. Turn the backlight off and turn the surface light on.
5. Move the stage until the first surface point that you want to focus on appears in the Image window.
6. Focus the surface manually by twisting the focus knob on the joystick.
7. Position the mouse cursor on the surface and press the left mouse button to perform an autofocus.
8. Press **Enter** on the joystick to accept the focus point.
9. Repeat Steps 5 through 8 for the other two focus points in the first group.
10. Repeat Steps 5 through 8 for the three focus points in the second group. The measurement results appear in the Cylinder Measurement Results window.
11. If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.
12. Click **OK** to complete the measurement.

A cylinder is displayed in the Model window as two circles, one at each end, connected by lines at the 0, 90, 180, and 270 degree points on the circles. Both circles are the same size.

8.25 Cylinder Measurement Results

The Measure Cylinder function reports the size, location, orientation, and form of a cylindrical object, calculated from data points taken on the surface of the cylindrical object.



The Measurement window displays:

- The number of points or features used in the measurement. You can edit or delete the entered points or features.
- Modifiers, which allow you to select how the cylinder is calculated: **Diameter** (default) or **Radius**
- Results, which include best-fit diameter or radius at the center of the cylinder and XYZ location of the midpoint of the cylinder's centerline axis.

  If you do not enter any nominal and tolerance values for the diameter and XYZ location, the reported diameter is at the midpoint of the cylinder's centerline axis.
- Cylindricity tolerance

- Check box for data stream output

If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.

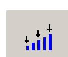
A cylinder is displayed in the Model window as two circles, one at each end, connected by lines at the 0, 90, 180, and 270 degree points on the circles. Both circles are displayed the same size.

8.26 How to Measure Min/Max/Avg Points

You can measure Min/Max/Avg points (typically on edges) with the Strong Edge and Focus tools, the Laser Focus tool, or any manual target.

Use this procedure to measure the points with the Strong Edge tool. You must measure at least two points.

1. Click  in the toolbox to select the Strong Edge tool.

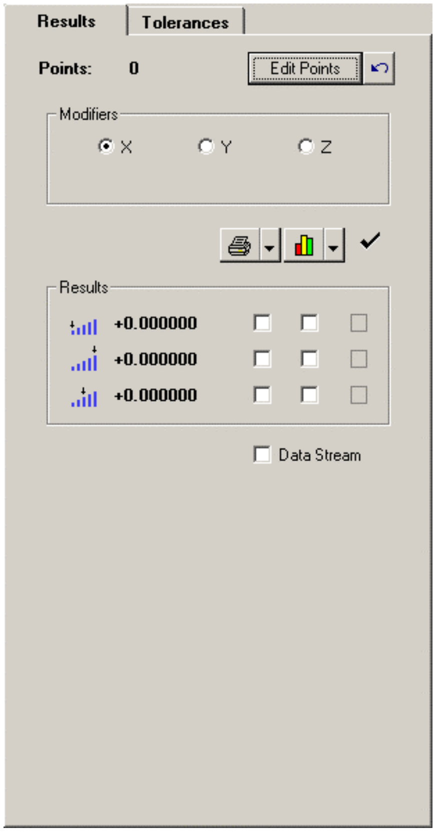
   The system displays the tool in the Image window.
2. [Classic User Interface] Click the **Measure** tab.
3. Click  in the toolbox.
4. Move the stage until the desired edge or surface appears in the Image window.
5. Place the cursor on the desired point on the edge and press the left mouse button.

   The software automatically moves the stage so that the point appears at the intersection of the crosshairs. Then it performs an edge analysis.
6. Press **Enter** on the joystick to accept the point.
7. Repeat Steps 4, 5 and 6 for the other points. The measurement results appear in the Min/Max/Avg Measurement Results window.
8. Select on the appropriate radio button to select the desired coordinate.

9. If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.
10. Click **OK** to complete the measurement.

8.27 Min/Max/Avg Measurement Results

The Measure Min/Max/Avg functions display the minimum, maximum and average value of the X, Y, Z, R, or A coordinates of a series of points. You must enter at least two points before any valid values are displayed.



The Measurement window displays:

- The number of points used in the measurement
- **X, Y, Z** (or R, A) radio buttons
- **Minimum, Maximum, and Average** values of the selected coordinate
- Check box for data stream output

To display the minimum, maximum and average values, select the radio button next to the desired coordinate. You can also select another coordinate to display the values for that coordinate.

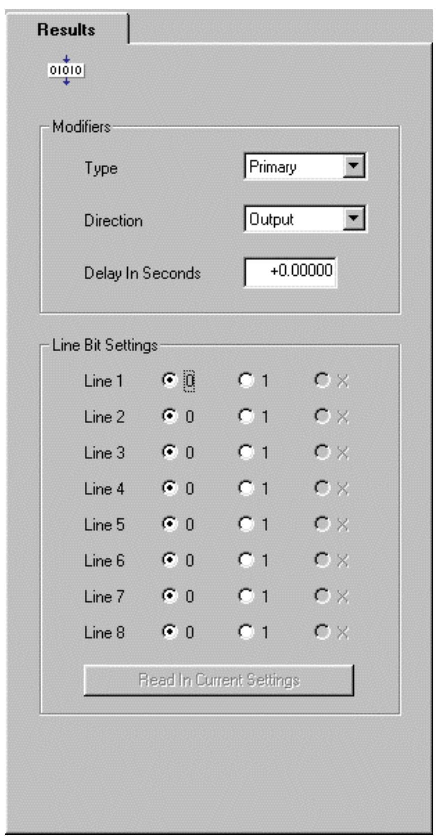
For example, when you specify three points on a vertical edge and select the Y coordinate, the minimum Y value is the one closest to zero and the maximum Y value is the one farthest away from zero.

8.28 Digital I/O Step

The Digital I/O function allows the software to communicate with other factory automation systems and control external devices. For example, this function is used to send and receive commands from external devices such as indexers, robots and automated fixtures.

A Digital I/O step has eight input bits and eight output bits. Each bit is on a separate line. The input bits are used for the commands coming in from the external devices. The output bits send commands to the external devices.

When you select **Measure / Digital I/O** (it must be enabled first), the output information is displayed in the Image and Measurement windows. The Image window displays the text that was entered in the Digital I/O Text in the System / Configuration menu.



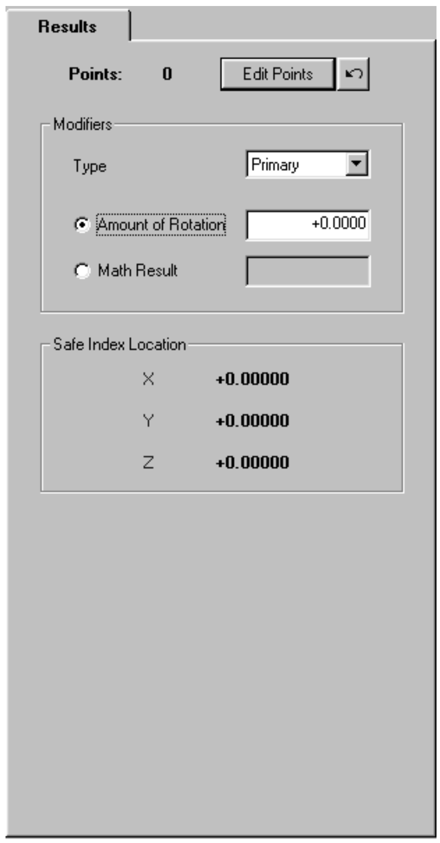
The Measurement window displays the following Digital I/O step information:

- **Direction drop-down list**, where you can select whether Input or Output information is displayed in the Image and Measurement windows. Output is the default direction.
- **Type drop-down list**, where you can select whether this step is associated with the primary rotary (default) or the secondary rotary, if it was enabled in the rotary configuration.
- **Eight radio buttons for each line**, which can be toggled between 0 (line is Off) and 1 (line is On). The default bit setting for each line is 0. For an Input step, the buttons can also be toggled to X (it does not matter whether the line is On or Off).
  - The Input logic is positive logic. Applying a 5 volt signal to a digital input will result in the software reading and displaying a "1" for the corresponding digital input line. Also, any disconnected inputs will read as a logic "1." Applying a 0 volt input will cause the software to display a "0" for the corresponding digital input line.
  - The Output logic is negative or inverted logic. Selecting an output of "1" for a digital output line causes a low level signal (0 volts) to appear at the corresponding output. Selecting an output of "0" for a digital output line produces a high (5 volts) output. On power up, the digital outputs default to 5 volts.
- **Delay (wait) period** in seconds. You can define a wait period for an operation to occur before the routine continues with the next step.
  - For an Output step, the routine pauses for the defined period after the command is sent.
  - For an Input step, the routine pauses to wait for the input bits to be received. If the expected bits are not received during this period, a message is displayed allowing you to continue or stop the routine.
- **Read in Current Settings button**. If this is an Input step, you can click this button to read the current settings from the digital I/O input port.

8.29 Rotary Indexer

If a rotary indexer is installed and the rotary has been configured in System / Configuration / Rotary, use this function to include a step in the inspection routine to rotate the indexer.

Before selecting this function, select a target or tool: manual alignment target (recommended), Focus tool, or Strong Edge tool.



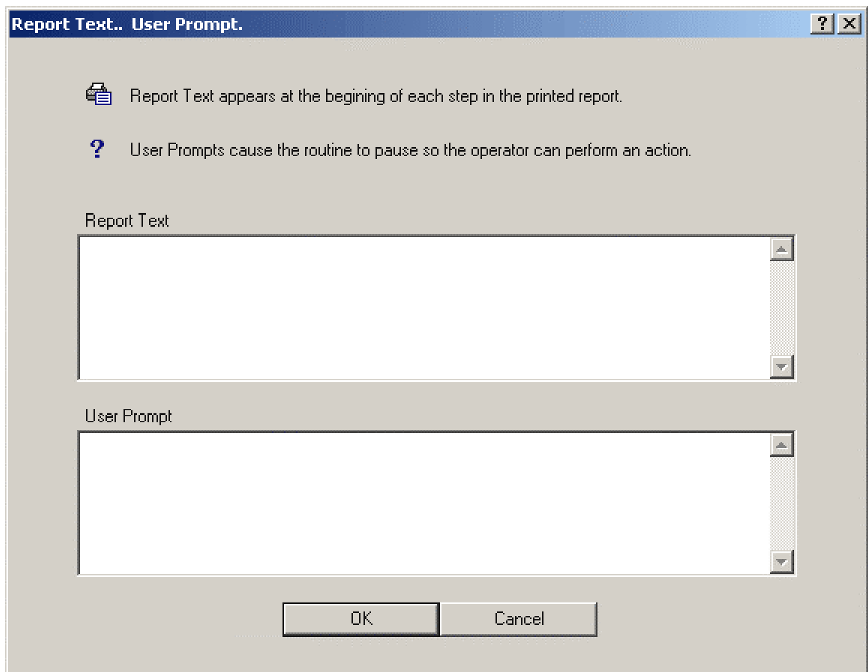
When you select Measure/ Rotary, the Measurement window displays:

- Type drop-down list to select the primary or secondary rotary
- Amount of Rotation / Math Result radio buttons, which are used to indicate and specify the relative move (rotation) of the indexer (the initial value is set to zero)
    - If you select the Amount of Rotation radio button, you may enter the relative move of the rotary in decimal degrees or degrees:minutes:seconds (specified in Units). A Plus (+) sign indicates a clockwise direction; a minus (-) sign indicates a CCW direction. The actual precision of the move depends on the resolution of your rotary table.
    - If you select the Math Result radio button, you can enter the step number containing the result of a math calculation. In this case, the value calculated in the specified math step is used for the relative move. The number of the entered step must be below the number of the current step.
- Safe X, Y and Z indexer locations. This indicates the coordinates of a safe location on the stage so that the optics are clear of the part when the indexer moves. You must move the stage to a "safe position" and press **Enter** on the joystick to specify the coordinates.
- Prompt / Text button. Click this button if you want to enter a one-line prompt that will appear when this step is encountered during the run of the routine. When you run the routine, it stops at this step and puts you in a manual mode so you can fine-tune the indexer.
- Again button. Click this button to repeat the relative move.

8.30 Feature Text and Prompt

When you click the **Prompt** / **Text** button in the Measurement window, the software displays a dialog box where you can type a comment and/or prompt for the **feature** being measured.

- Any text that you enter will appear in the Comment field in the routine listing and in the selected output when you run the routine.
- Any prompt that you enter will appear when the software encounters the feature, e.g., during a run or when you edit the step. The routine will pause and you will need to perform an action to continue the routine.



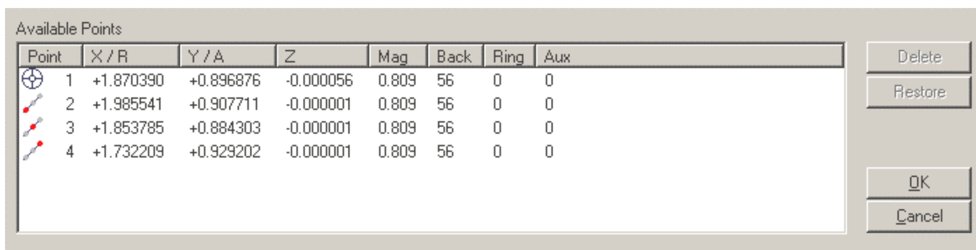
To enter the text or prompt:

1. Place the cursor in the desired field and click the left mouse button once.
2. Type the desired text. If you reach the end of the line, characters will wrap to the next line.
3. If you want to go to another line, press **Enter** on the keyboard and type the desired text.

4. Click **OK** or click Cancel to return to the Run window.
   - OK saves the text and any changes/additions you made.
   - Cancel saves only your original text; any additions or changes are not saved.

8.31 How to Edit Measured Points

You can edit the coordinates of all the data points used to measure a feature whenever the Measurement window displays the measurement results (this does not apply to the results of a construction).



To edit the displayed values for any of the points that you specified when you measured a feature:

1. Click the **Edit Points** button in a measurement step. This opens a dialog box with the specified points below the Image window.
2. Position the mouse cursor over the desired value or press the Tab key to get to the value, and double-click the value.
3. Change the value as desired: highlight the digits to be changed and type the new value.
4. Repeat Steps 2 and 3 for other values, if you want.
5. Click **OK** at the bottom of the dialog box to close it.
6. If you measured the feature with a weak edge tool, click the **Remeasure** button in the Weak Edge Settings window to view the effect of your changes.

The edited values become the nominal location that the stage will be driven to when the routine is subsequently run. All measurements based on the modified feature are recalculated.

- You can delete only points that were measured with a Strong Edge or Focus tool, or a manual alignment target. You cannot delete any of the three FeatureFinder points.

- For weak edge features, you can edit the start, end and stage locations of the feature measurement. However, entering nominal values in the feature measurement will override any changes made in this dialog box.

- If you measured a feature with a weak edge tool, i.e., FeatureFinder or Weak Edge Point, the Edit Points window also displays a Reverse button. This button enables you to reverse the search direction; however, the software still searches in a dark-to-light direction. This function makes it easier to measure difficult edges where the initial search direction may be incorrect.

8.32 Search Function

The Search function is used to measure features that have a direct location correlation with other features. When you turn this function on in a step, you can include a reference feature. The system automatically fills in the offset values between the two features (the software adds the deviation from the location of the reference feature to the current feature's location). When you click Remeasure or run the routine, the system measures the current feature using the deviations from the reference feature.

This function is available in every Measure step except Digital I/O. It is available on a per axis basis (i.e., multiple features are allowed as references). Any individual axis may be pursued at its original measured / nominal location, rather than relative to a reference feature.

Be sure that the reference steps used in the Search function contain the appropriate location nominal values. If the values are not specified, the software automatically copies the actual values and uses them instead. The software does not display any messages when it does this.

Do not use Step Edit when using the Search function. Step Edit allows the stage to move, producing an offset change. To edit steps while using the Search function, select Edit / Change from the main menu, or click the **Change a step** icon in the toolbox.

8.32.1 How to Use the Search Function

1. Measure the reference feature(s), i.e., the feature(s) that will be specified in the Search Settings window.
2. Click the **Tolerances** tab and specify the nominal values for the desired location dimensions.
3. Measure the current feature in which the Search function will be enabled.
4. Click the **Search** button in the current step to display the Search Settings window.
5. Select the check box of the desired X, Y, and or Z (or RAZ) coordinates. This enables the field for entering the reference step number.
   - The software automatically fills in the step number of the previous valid step and checks that the reference step number is valid.
   - Valid steps are measured or constructed features that have a location and Math steps.
6. If you want to specify a different step, enter the step number or click the feature in the Model window.
   - The software checks that the reference step number is valid.
   - You cannot enter a step number greater than or equal to the current step.
7. Click **OK** in the Search Settings window to accept the changes or keep the current settings.
8. Remeasure the current feature (if measured with a weak edge tool) or run the routine.

When you click Remeasure or run the routine, the system measures the current feature using the deviations from the reference feature. If the reference feature fails when the routine is run, the software will try to measure the current feature at the nominal location and ignore the offset.

The Search button is disabled for the first step in the routine.

- The following features are not considered valid reference steps: Distance (measure and construct), Digital I/O, Datum Origin, Datum Axis, Datum Level, Branch, and User Input.

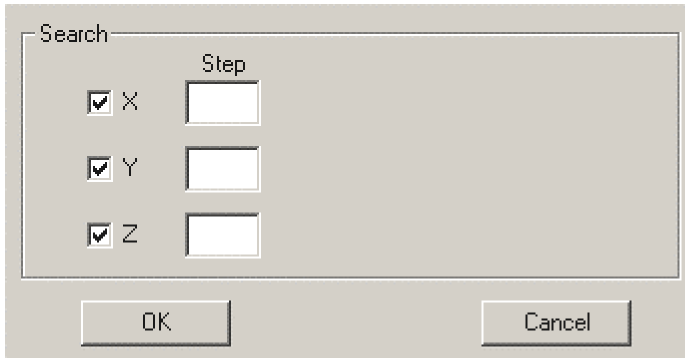
- You may specify a Math step as a reference step. In this case the deviation from the nominal of the result of the math expression will be used as the axis value for the search. This allows complex math expressions to be entered (such as X5+WID10) and used for the search. An offset value is put into the Offset edit box, but it is grayed out.

- When steps are inserted or deleted, the reference step numbers are updated appropriately. If a Search reference step is deleted, the search for that axis is also turned off.

- When steps with the Search function are copied, the reference step numbers are updated if the reference steps are included in the copy.

8.32.2 Search Settings Window

The Search Settings window contains the XYZ (or RAZ when using polar coordinates) check boxes and fields for specifying the reference steps for the search.



When using the Search function, you can do the following:

- Select the check box of the desired X, Y, and or Z (or RAZ) coordinates. This enables the field for entering the reference step number.
  - The software automatically fills in the step number of the previous valid step.
  - If you want to specify a different step, enter the step number or click the feature in the Model window. You cannot enter a step number greater than or equal to the current step.
- Click **OK** to accept the changes or keep the current settings.
- Click **Cancel** to disregard any changes and close the window.

8.32.3 Search Function Examples

Several examples of the Search function are described below. Be sure to specify the appropriate nominal values in the reference steps.

- To measure the depth at the center of a trough or channel, measure the edges of the trough, construct the width, and then use a focus step. In the focus step, include a reference to the width to perform the focus at the actual center of the width.
- To measure a countersink diameter that is concentric to a through-hole, measure the through hole first, followed by the countersink diameter. In the second measurement, include a reference to the first hole to measure the true diameter of the hole.
- To measure the size of a fillet (corner) radius, measure the edges of the corner, construct an intersection, and then measure the corner (arc). In the arc step, include a reference to the intersection step. The deviation values represent the actual distance from the center of the measured arc to the intersection point. When the system measures the arc, the deviations are included in the measurement.
- To measure features such as pins that may not be parallel to one another, measure the two long edges and construct a width. Then measure a point at the end of the pin and include a reference to the Width step for the specified axis. The deviations are displayed and included in the Point measurement.

This page was left blank intentionally.

9. Construct Functions

9.1 Overview

This section describes how to construct the following:

- Point
- Line
- Circle
- Intersection
- Angle between intersecting features
- Width
- Plane
- Contour
- Sphere
- Distance
- Midpoint
- Cone
- Cylinder
- Min/Max/Avg Points

> The Construct Sphere, Construct Cone, and Construct Cylinder functions are only available if the system is equipped with a touch probe.
>
> Gage functions, math function, branch functions, and data extraction are also covered in this section.

9.2 How to Construct a Point

The point function allows you to define a reference point in space or to select a feature from the model without positioning the stage.

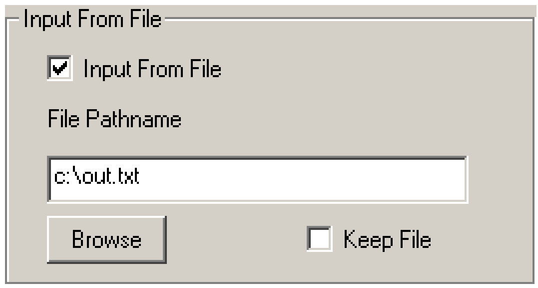
To construct a point:

1. [Classic User Interface] Click the **Construct** tab.
2. Click in the toolbox or select **Construct** / **Point** from the main menu.
3. Select the desired feature in the Model window or enter the desired value in the Tolerances window.

   The construction results appear in the Point Measurement Results window.
4. If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.
5. Click **OK** to complete the construction.

9.3 Point Input from a File

When you construct a Point, you can input the point using an XYZ (or RAZ) location that is specified in a file. The coordinates must be in the current part coordinate system, in an XYZ or RAZ format, and in the units of the step.



To input a point from a file:

1. Construct a Point step.
2. Click the **Tolerances** tab.
3. Select the **Input from File** check box.
4. Specify the path and name of the file, or click the Browse button to search for the file.
5. If you want to reuse the file, select the **Keep File** check box.
6. Click the **Results** tab to return to the measurement results window.

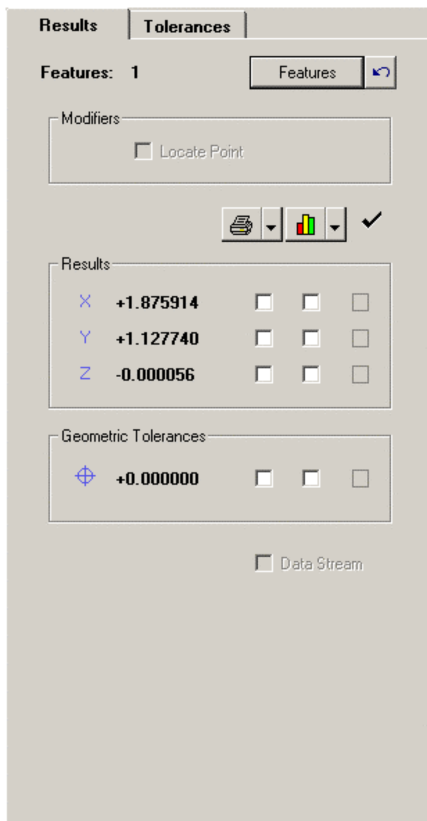
When the software encounters the constructed Point step during the run of the routine, it:

- Displays a message that it is searching for the file
- Reads only the first line (maximum of 60 characters per line) in the specified file
- Deletes the file after reading the point input, unless you select the Keep File check box

9.4 Point Construction Results

A point is a single location.

- The number of points used in the measurement. You can edit or delete the entered data point.
- XYZ coordinates of the point. If the point is constructed from a feature such as a circle or line, the XYZ coordinates indicate the center or midpoint of the selected feature.
- Position tolerance.



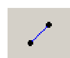
If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.

9.5 How to Construct a Line

The Construct Line function reports the orientation, location, and form of a line, as calculated from the selected features.

> Measurement results appear in the Measurement window when at least two features are selected. These can be previously measured or constructed features such as other lines, circles or points.

To construct a line:

1. [Classic User Interface] Click the **Construct** tab.
2. Click  in the toolbox or select **Construct** / **Line** from the main menu.
3. Click a feature in the Model window to select it. The feature appears in a contrasting color (green).
4. Repeat the previous step to select the required number of features or the desired features. Click the Remove Features button if you want to remove a selected feature. To select multiple features at once, press and hold the Alt key while clicking and dragging in the Model window with the left mouse button. This allows you to draw a box around a group of features, all of which are selected when you release the mouse button.

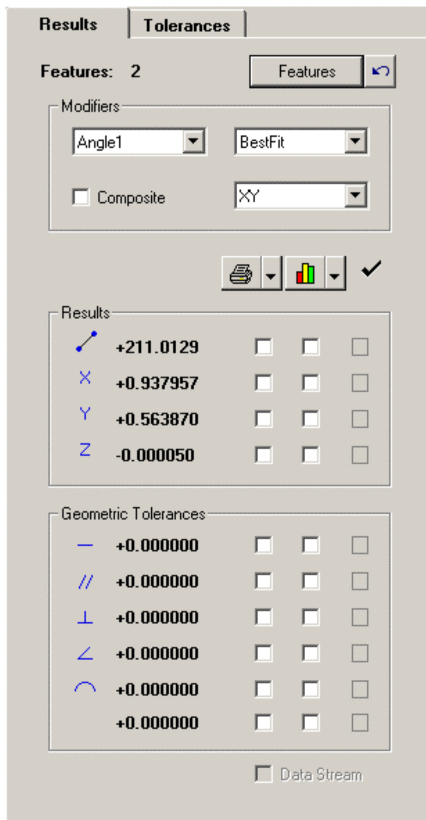
   After you select the minimum number of features, the Measurement window displays the Line Measurement Results.
5. If you want to construct a composite line, select the **Composite** check box.
6. If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.
7. Click **OK** after selecting the features.

The Model window displays the constructed line as a dashed line. The system redraws the model only if required to display all the features.

9.6 Line Construction Results

The Construct Line function reports the orientation, location and form of a line or straight edge as calculated from the entered data points.

Measurement results appear in the Measurement window when at least two points or features are entered.



The Measurement window displays:

- The number of points or features used in the measurement. You can edit or delete the entered points or features.
- Modifiers, which allow you to select how the line is calculated:
  - Angle or supplementary angle, relative to the currently-defined X axis
  - Line calculation type. Select the desired type from the drop-down list
  - Coordinate plane in which the angle is reported
  - Composite check box, which indicates how the software calculates the constructed line. This appears only if the line is constructed.
- Results, which include the angle and the XYZ coordinates of the midpoint of the line
- Geometric tolerances, including straightness, parallelism, perpendicularity, angularity, and profile

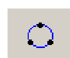
If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.

9.7 How to Construct a Circle

The Construct Circle function reports the size, location, orientation, and form of a circle, as calculated from the selected features. For example, you can construct a bolt circle from other circles in a circular pattern.

> Measurement results appear in the Measurement window when at least three features are selected. These can be previously measured or constructed features such as other lines, circles or points.

To construct a circle:

1. [Classic User Interface] Click the **Construct** tab.
2. Click  in the toolbox or select Construct / Circle from the main menu.
3. Click a feature in the Model window to select it. The feature appears in a contrasting color (green).
4. Repeat the previous step to select the required number of features for the desired feature. Click the Remove Features button if you want to remove a selected feature. To select multiple features at once, press and hold the Alt key while clicking and dragging in the Model window with the left mouse button. This allows you to draw a box around a group of features, all of which are selected when you release the mouse button.

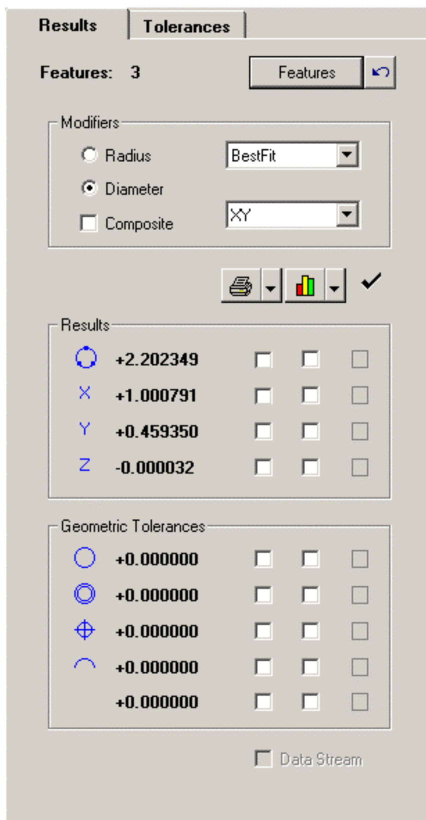
   After you select the minimum number of features, the Measurement window displays the Circle Construction Results.
5. If you want to construct a composite circle, select the **Composite** check box.
6. If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.
7. Click **OK** after selecting the features.

The Model window displays the constructed circle with dashed lines. The system redraws the model only if required to display all the features.

9.8 Circle Construction Results

The Construct Circle function reports the size, location, orientation and form of a circle or arc (rounded or curved edge) as calculated from the entered data points.

Measurement results appear in the Measurement window when the minimum number of required points or features is entered.



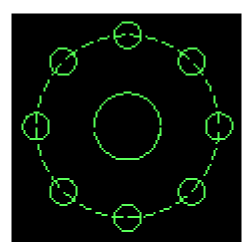
The Measurement window displays:

- The number of points or features used in the measurement. You can edit or delete the entered points or features.
- Modifiers, which allow you to select how the circle is calculated:
  - **Diameter** (default) or **Radius**
  - Circle calculation type. Select the desired type from the drop-down list.
  - Coordinate plane in which the radius/diameter is reported
  - Composite check box, which indicates how the software calculates the constructed circle or arc. This appears only if the circle is constructed.
  - Results, which include the diameter or radius and the **XYZ coordinates** of the center of the circle
- Geometric tolerances, including roundness, concentricity, position, and profile

If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.

9.9 Bolt Circle

A bolt circle is constructed from other circles that are in a circular pattern. The center points of the other circles already exist because they are the locations of previously measured features. You can determine the center point and diameter of the bolt circle using the calculated centers of the other circles.



For example, a bolt circle is constructed from the smaller circles that are in a circular pattern on the QVI training part. The center of the bolt circle is the same as the center of the large circle in the middle.

9.10 Composite Features

The composite check box indicates how the software calculates the constructed line, constructed circle, constructed cone, constructed cylinder or constructed contour.

- If you selected the check box, the software uses the XYZ locations of the individual data points that make up (compose) each reference feature. In this case, you can construct a line from a single line or a circle from a single circle.
  - A reference feature can be measured or constructed.
  - If the reference feature is a previously constructed composite feature, the software uses the data points of the features in the original composite construction.
- If you left the check box unchecked, the software uses the actual location of each reference feature (for example, the midpoint of a line or the center of a circle).

> When you construct a composite feature from previous composite features, the software searches through the features until it finds a non-composite feature and it uses the data points of the non-composite features in the construction.

9.11 Composite Feature Examples

Composite circle examples:

- If you measure a feature with a relatively shallow arc (e.g., a circle that is much larger than the field of view), you can measure the arcs with the FeatureFinder line tool and construct a composite circle from the measured lines.
- Any individual measured or constructed reference feature can be constructed as a composite circle. For example, to also output the minimum and maximum diameters, measure a circle in one step and construct composite circles in the next two steps.

Composite line examples:

- If there is a midpoint between two circles and you want to construct a line between the two circles that goes through the midpoint, simply select the midpoint in the Line step and select the Composite check box. When you click the OK button, the composite line is drawn between the centers of both circles through the midpoint.
- You can construct a composite line between the two measured lines that are used in an intersection. To do so, measure the two lines with FeatureFinder, construct the intersection, and select the intersection in the Line step. When you select the Composite check box and click the OK button, the composite line is drawn between the midpoints of the lines used in the construction.

9.12 How to Construct an Intersection

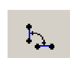
The Construct Intersection function reports the orientation and location of a point where two features meet.

Measurement results appear in the Measurement window when the two features are selected. These can be previously measured or constructed lines or circles.

You can construct the following kinds of intersections:

- Between two lines (reports the angle between the two lines and the intersection point)
- Between a line and a circle (reports one or two points)
- Between two circles (reports one or two points)

To construct an intersection:

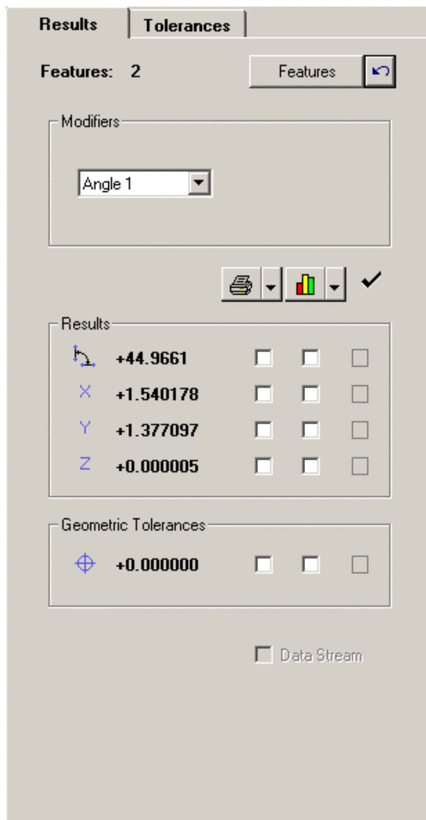
1. [Classic User Interface] Click the **Construct** tab.
2. Click  in the toolbox or select **Construct** / **Intersection** from the main menu.
3. Click a feature in the Model window to select it. The feature appears in a contrasting color (green).
4. Repeat the previous step to select the second feature. Click the Remove Features button if you want to remove a selected feature. To select multiple features at once, press and hold the Alt key while clicking and dragging in the Model window with the left mouse button. This allows you to draw a box around a group of features, all of which are selected when you release the mouse button. After you select the two features, the Measurement window displays the Intersection Construction Results.
5. If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.
6. Click **OK** after selecting the features.

The Model window displays the constructed intersection as a cross. The system redraws the model only if required to display all the features.

9.13 Intersection Construction Results

The Construct Intersection function reports the orientation and location of a point where two features meet.

🗎 Measurement results appear in the Measurement window when the two features are selected.



The Measurement window displays:

- The number of features used in the construction.
- Drop-down list where you can select one of four angles.
- **1 of 2** and **2 of 2** radio buttons (if there are two intersection points). If the 2 of 2 button is active, this indicates that there are two intersection points. Click this button to display the angle and coordinates of the second intersection point (2 of 2).
- **Angle** between the two features.
- **XYZ coordinates** of the intersection point.

- Position tolerance.

9.14 Angles between Intersecting Features

The software can report any one of four intersection angles between two lines, even if the two lines do not physically meet. The angles are reported as positive values going in a counterclockwise (CCW) direction.

- Angle 1 is the included angle between the endpoints
- Angle 2 is 360° - Angle 1
- Angle 3 is 180° - Angle 1 (if this value is negative, it is added to 360°)
- Angle 4 is 360° - Angle 3

The included angle is calculated from the endpoint of the first line to the endpoint of the second line. The endpoint is determined by the vector direction in which each line was measured (or selected):

- For lines measured with the FeatureFinder or Edge Trace tools, the vector direction of the line is based on the dark-to-light rule. For example, points are measured clockwise for a rectangular through-hole and counter-clockwise for a solid object or a rectangular filled object.
- For constructed lines or lines measured with the Strong Edge tool, the vector direction of the line is based on the direction in which the first two points were measured (or selected).
- The order in which the angles are displayed depends on which line was selected (not measured) first.

9.15 How to Construct a Width

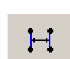
The Construct Width function reports the size, location and orientation of a width.

> Measurement results appear in the Measurement window when the two features are selected. These can be previously measured or constructed features such as other lines, circles or points.

You can construct the following kinds of widths (see *Width Calculation* for a description of how the widths are calculated):

- Between two points
- Between a point and a line. The location of the width lies at the midpoint of the perpendicular dropped from the point to the line.
- Between a point and a circle
- Between two lines. The width is calculated as a perpendicular between two best-fit lines.
- Between a line and a circle. The location of the width lies at the midpoint of the perpendicular dropped from the center of the circle to the line.
- Between two circles reports the orientation, location, and form of a line, as calculated from the selected features.

To construct a width:

1. [Classic User Interface] Click the **Construct** tab.
2. Click  in the toolbox or select **Construct** / **Width** from the main menu.
3. Click a feature in the Model window to select it. The feature appears in a contrasting color (green).

4. Repeat the previous step to select the second feature. Click the Remove Features button if you want to remove a selected feature. To select multiple features at once, press and hold the Alt key while clicking and dragging in the Model window with the left mouse button. This allows you to draw a box around a group of features, all of which are selected when you release the mouse button.

   After you select the two features, the Measurement window displays the Width Construction Results.

5. If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances. For an orientation tolerance, the software compares the specified reference feature with the width centerline.

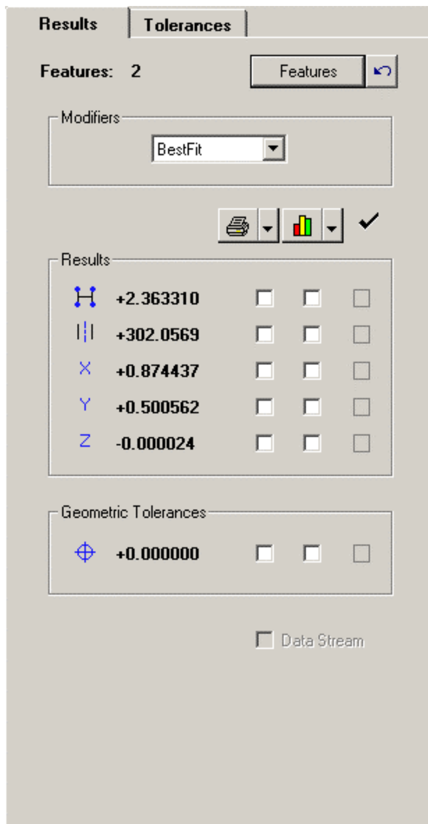
6. Click **OK** after selecting the features.

The Model window displays the constructed width as a cross. The system redraws the model only if required to display all the features.

9.16 Width Construction Results

The Construct Width function reports the size, location and orientation of a width.

Measurement results appear in the Measurement window when the two features are selected. These can be previously measured or constructed features such as other lines, circles or points.



The Measurement window displays:

- The number of features used in the construction
- Width modifier to select the best-fit width, minimum width or maximum width
- Selected width
- **Angle** of the width centerline
- The **XYZ location** of the center of the constructed width

- **Geometric tolerances**, including position and orientation (parallelism, perpendicularity, and angularity)

> The position tolerance calculation of a width between two lines is based on ASME Y14.5. When using a True Position tolerance with a width, you must enter nominal values for X, Y, Z, and C/L Angle so that the correct true position can be calculated. Large true position errors may result if you do not enter the correct nominal values.

9.17 Width Calculation

The calculations of the value and location of a constructed width are described below.

- For **two points**, the width is the straight line distance between the points, projected into the XY plane. There is no minimum or maximum width in this case. The location is the midpoint between the two points.
- For a **point and a line**, the width is the perpendicular distance from the point to the line, projected into the XY plane. There is no minimum or maximum width in this case. The location is the midpoint of the perpendicular.
- For a **point and a circle**, the width is the perpendicular distance from the point to the center of the circle, projected into the XY plane. The minimum width is this distance minus the circle radius. The maximum width is the distance plus the circle radius. The location is the midpoint of the distance.
- For **two lines**, the width is the distance between the two best fit lines, measured perpendicular to the width centerline and projected into the XY plane. The minimum width is measured between the two closest points, perpendicular to the best fit centerline. The maximum width is measured between the two farthest points, perpendicular to the best fit centerline. The location is the midpoint of the distance.
- For a **line and a circle**, the width is the perpendicular distance from the center of the circle to the line, projected into the XY plane. The minimum width is this distance minus the circle radius. The maximum width is the distance plus the circle radius. The location is the midpoint of the distance.

- For **two circles**, the width is the distance between the centers of the two circles, projected into the XY plane. The minimum width is this distance minus the radius of both circles. The maximum width is the distance plus the radius of both circles. The location is the midpoint of the distance.

9.18 How to Construct a Plane

The Plane function reports the orientation, form and location of a plane, as calculated from the entered data points.

> Measurement results appear in the Measurement window when the minimum number of required features is selected.

When you construct a plane, you must select a minimum of three points to display the plane angle; a minimum of four points is required to display the profile or flatness. Use the following guidelines to select the data points:

- Use previous features that included autofocus with the Focus tool. This ensures that the Z-axis coordinate is measured.
- The number of points and their distribution may affect the results. For best results, make sure that the measured points are spread as evenly as possible around the plane.

To construct a plane:

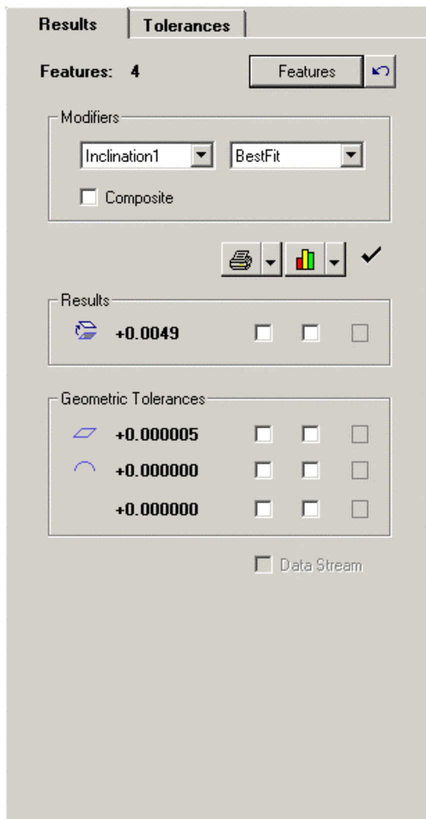
1. [Classic User Interface] Click the **Construct** tab.
2. Click in the toolbox or select **Construct** / **Plane** from the main menu.
3. Click a feature in the Model window to select it. The feature is displayed in a contrasting color (green).

4. Repeat the previous step to select the required number of points or the desired points. Click the Remove Features button if you want to remove a selected feature. To select multiple features at once, press and hold the Alt key while clicking and dragging in the Model window with the left mouse button. This allows you to draw a box around a group of features, all of which are selected when you release the mouse button. After you select the minimum number of points, the Measurement window displays the Plane Construction Results.
5. If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.
6. Click **OK** after selecting the features.

9.19 Plane Construction Results

The Construct Plane function reports the orientation, form and location of a plane, as calculated from the entered data points. However, this function does not return a location. In the Model window you cannot tell which points were used to measure the plane.

> Measurement results appear in the Measurement window when the minimum number of required points or features is selected.



The Measurement window displays:

- The number of points or features in the measurement.
- **Inclination angle** of the plane. You can select and display the negative value of Angle 1 from the drop-down list.
- Plane calculation type. Select the desired type from the drop-down list.
- Geometric tolerances:
  - **Flatness**. This indicates the largest difference in the Z axis between entered points, measured perpendicular to the plane (whether minimum, maximum or best fit).
  - **Profile +**. This indicates the perpendicular distance from the measured plane to the point that is farthest above the plane. This is 0 when you use Maximum Contact because there are no higher points.
  - **Profile -**. This indicates the perpendicular distance from the measured plane to the point that is farthest below the plane. This is 0 when you use Minimum Contact because there are no lower points.

You can measure or construct the points either before or after selecting the type of plane calculation. You can change the calculation type after measuring the points and view the different results.

9.20 How to Construct a Contour

The Construct Contour function reports the location and size (length and area) of an irregular feature as calculated from the selected features.

Measurement results appear in the Measurement window when at least two features are selected. These can be previously measured or constructed features such as other lines, circles or points.

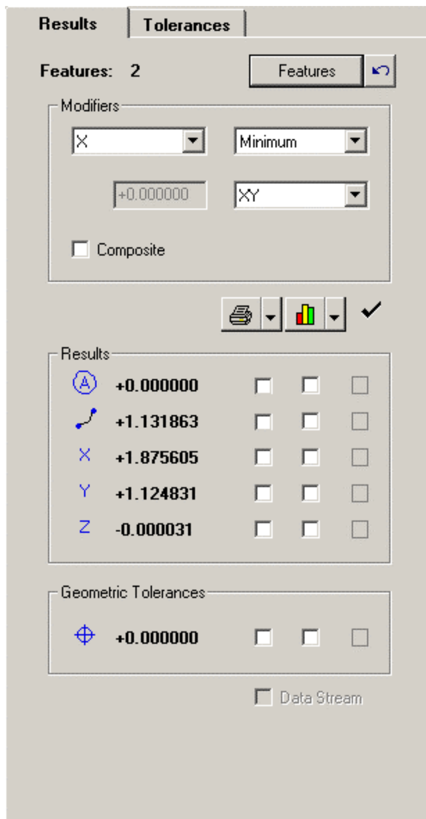
To construct a contour:

1. [Classic User Interface] Click the **Construct** tab.
2. Click  in the toolbox or select **Construct** / **Contour** from the main menu.
3. Click a feature in the Model window to select it. The feature is displayed in a contrasting color (green).
4. Repeat the previous step to select the required number of features or the desired features. Click the Remove Features button if you want to remove a selected feature. To select multiple features at once, press and hold the Alt key while clicking and dragging in the Model window with the left mouse button. This allows you to draw a box around a group of features, all of which are selected when you release the mouse button.
   - After you select the minimum number of features, the Measurement window displays the Contour Construction Results.
   - If you want to construct a composite contour, select the Composite check box.
5. If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.
6. Click **OK** after selecting the features.

The Model window displays the constructed contour with dashed lines. The system redraws the model only if required to display all the features. To select the contour in the Model window, click the features rather than the contour itself.

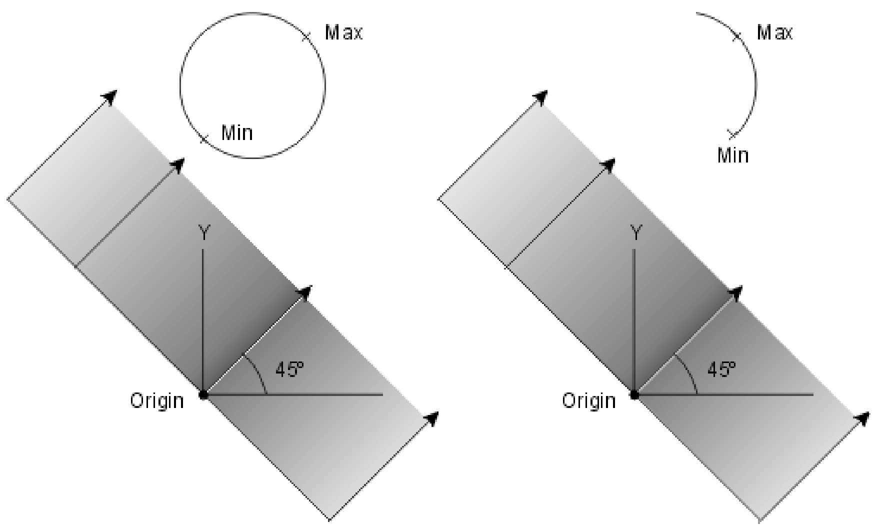
9.21 Contour Construction Results

The Construct Contour function reports the location and size (length and area) of an irregular feature as calculated from the entered data points.



The Measurement window displays:

- The number of points or features used in the measurement. You can edit or delete the entered points or features.
- Location drop-down list.
  - You can select X, Y, Z, Angle, Centroid, or Radius.
  - If you select Angle, the Angle field becomes active to specify the desired angle. If none is specified, the angle is 0 degrees. The system searches for and displays the maximum (highest) or minimum (lowest) XYZ point at the specified angle.



Note: In these examples, the software searches for a point (Min or Max) along a 45° vector. The vector intersects the feature; however, the Max and Min intersection point does not have to be along the same vector.

- Fit drop-down list.
  - You can select Minimum or Maximum.
  - This list is disabled if Centroid is selected in the Location drop-down list.
- Coordinate plane in which the contour is reported.
- Composite check box, which indicates how the software calculates the constructed contour. This appears only if the contour is constructed.
- The area of a closed contour.
- The length of the contour.

- The XYZ location of the closed contour based on the selected modifiers; if Centroid is selected, this is the center location of the mass.
- The position tolerance of the closed contour.

If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.

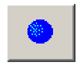
9.22 How to Construct a Sphere

The sphere function reports the size, location and form of a sphere, calculated from the selected data points.

When you construct a sphere, you must select at least four points. Use the following guidelines to select the data points:

- If the points were measured optically, at least one of the first four points should include an autofocus to capture the Z-axis measurement. This is typically the point measured at the top or bottom of the sphere. You can select this point either before or after the "equator" points.
- For best results, make sure that the selected points are spread as evenly as possible around the sphere.
- To ensure that a hemisphere is displayed in the Model window when Spherical Radius is selected:
  - Make sure that all the points are on the same side of the equator.
  - The first two points and the center location of the sphere determine the plane of the equator.

To construct a sphere:

1. [Classic User Interface] Click the **Construct** tab.
2. Click  in the toolbox or select **Construct** / **Sphere** from the main menu.
3. Click a feature in the Model window to select it. The feature is displayed in a contrasting color (green).
4. Repeat the previous step to select the required number of points or the desired points. Click the Remove Features button if you want to remove a selected feature. To select multiple features at once, press and hold the Alt key while clicking and dragging in the Model window with the left mouse button. This allows you to draw a box around a group of features, all of which are selected when you release the mouse button.

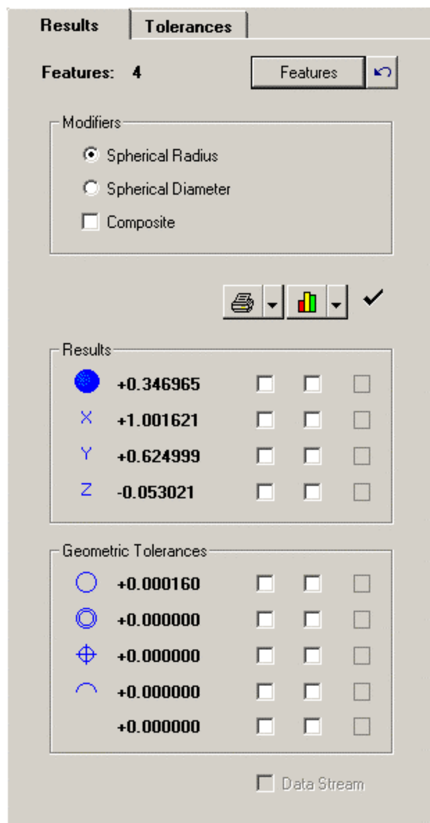
   After you select the minimum number of features, the Measurement window displays the Sphere Construction Results.
5. If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.
6. Click **OK** after selecting the features.

The Model window displays the constructed sphere as a dashed-line circle. The system redraws the model only if required to display all the features.

9.23 Sphere Construction Results

The Construct Sphere function reports the size, location and form of a sphere, calculated from the entered data points.

Measurement results appear in the Measurement window when the minimum number of required points or features is selected.



The Measurement window displays:

- The number of points or features used in the measurement.
- Spherical Diameter and Spherical Radius radio buttons.
- **Diameter** or **Radius**, whichever is selected.
- **XYZ coordinates** of the center of the sphere
- **Geometric tolerances**, including Circularity, Concentricity, Position, and Profile.

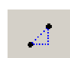
In the Model window, a measured sphere appears as a solid-line circle and a constructed sphere appears as a dashed-line circle.

9.24 How to Construct a Distance

The Construct Distance function reports the distance and angular separation between two features.

Measurement results appear in the Measurement window when two features are selected. These can be previously measured or constructed features such as other lines, circles or points.

To construct a distance:

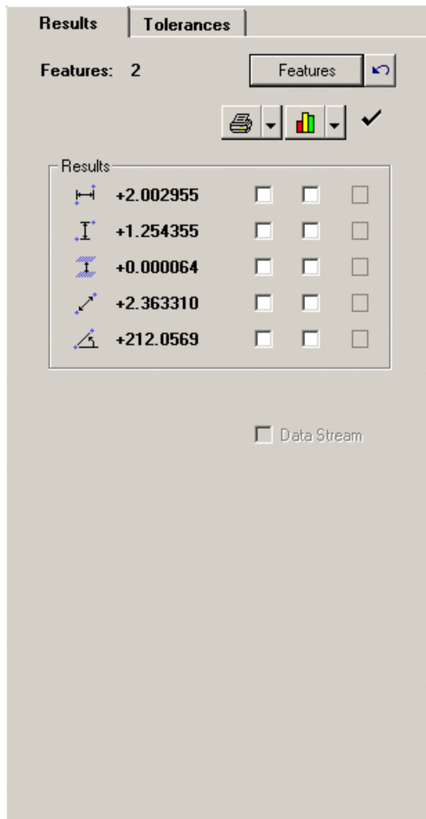
1. [Classic User Interface] Click the **Construct** tab.
2. Click  in the toolbox or select **Construct** / **Distance** from the main menu.
3. Click a feature in the Model window to select it. The feature appears in a contrasting color (green).
4. Repeat the previous step for the second feature. Click the Remove Features button if you want to remove a selected feature. To select multiple features at once, press and hold the Alt key while clicking and dragging in the Model window with the left mouse button. This allows you to draw a box around a group of features, all of which are selected when you release the mouse button.

   The Measurement window displays the Distance Construction Results.
5. If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.
6. Click **OK** after selecting the features.

9.25 Distance Construction Results

The Construct Distance function reports the distance and angular separation between two points. However, the distance is not displayed in the Model window.

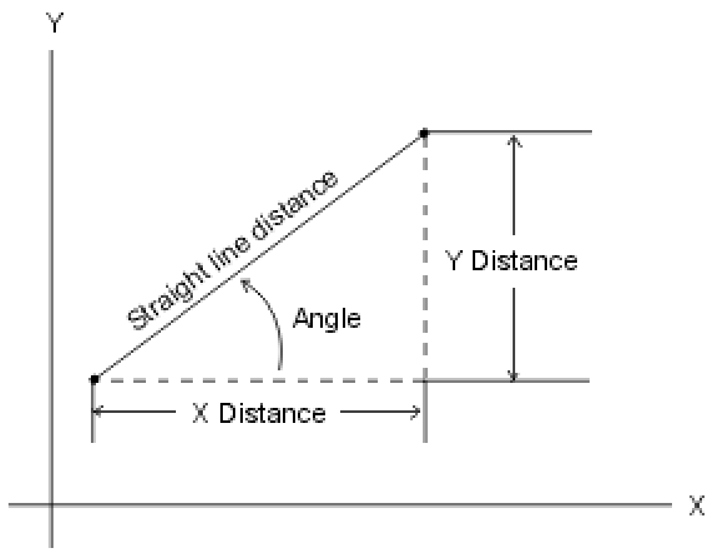
Measurement results appear in the Measurement window when the two points are entered.



The Measurement window displays the results in the following order:

- The number of points or features used in the measurement.
- **X distance** from one point to the other.
- **Y distance** from one point to the other.
- **Z distance** from one point to the other.
- **Straight line distance** between the two points.
- **Angle** of the straight line between the two points, based on 0° (the positive X axis).

If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.



9.26 How to Construct a Midpoint

The midpoint function reports the midpoint between two features.

> Measurement results appear in the Measurement window when the two features are selected. These can be previously measured or constructed features such as other lines, circles or points.

To construct a midpoint:

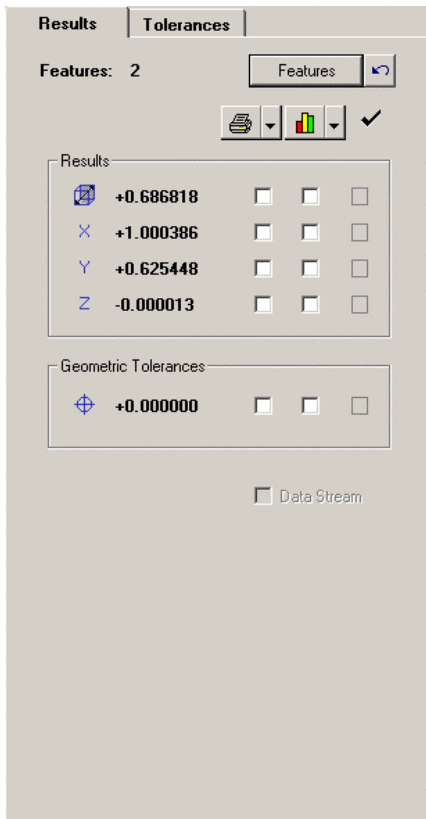
1. [Classic User Interface] Click the **Construct** tab.
2. Click  in the toolbox or select **Construct** / **Midpoint** from the main menu.
3. Click a feature in the Model window to select it. The feature is displayed in a contrasting color (green).
4. Repeat the previous step to select the second feature. Click the Remove Features button if you want to remove a selected feature. To select multiple features at once, press and hold the Alt key while clicking and dragging in the Model window with the left mouse button. This allows you to draw a box around a group of features, all of which are selected when you release the mouse button.

   After you select the two features, the Measurement window displays the Midpoint Construction Results.
5. If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.
6. Click **OK** to complete the construction.

9.27 Midpoint Construction Results

The Construct Midpoint function reports the 3D distance and midpoint between two points. The distance is three-dimensional; the calculations are based on all three axes. It is displayed as an asterisk in the Model window.

Measurement results appear in the Measurement window when the two points are entered.



The Measurement window displays:

- The number of points used in the measurement.
- **3D straight line distance** between the two entered points
- **XYZ coordinates** of the midpoint
- Position tolerance

9.28 How to Construct a Cone

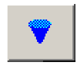
The Cone function reports the orientation, location and size of a cone, calculated from the selected features.

> Measurement results appear in the Measurement window when the minimum number of required features is selected. These can be previously measured or constructed features. For example, a cone can be constructed from points, or from a previous cone or two circles if the Composite check box is selected.

If you construct a cone from measured or constructed points, you must select at least six points, in two groups of 3 points each. The order in which the two groups are selected does not matter. Use the following guidelines to select the data points (it may be easier to select points in Top View):

- Select three points near or at one end of the cone, which defines a circle or arc.
- Select three points near or at the other end of the cone, which defines a circle or arc.
- After the first six points, you can select other points between the two groups.
- For more accurate results, increase the distance between the two sets of points and increase the total number of selected points.
- The vector direction points away from the apex of the cone.
- The line going through the two centers is used to calculate the cone's centerline axis.
- To display a partial cone in the Model window when Radius is selected:
  - Make sure that the paths of the two groups of points do not form complete circles.
  - The first point determines where the start angle begins, then the path goes through the second point, and the last point determines the end angle of the path.

To construct a cone:

1. [Classic User Interface] Click the **Construct** tab.
2. Click  in the toolbox or select **Construct** / **Cone** from the main menu.
3. Click a feature in the Model window to select it. The feature appears in a contrasting color (green).
4. Repeat the previous step to select the required number of points or the desired points. Click the Remove Features button if you want to remove a selected feature. To select multiple features at once, press and hold the Alt key while clicking and dragging in the Model window with the left mouse button. This allows you to draw a box around a group of features, all of which are selected when you release the mouse button.

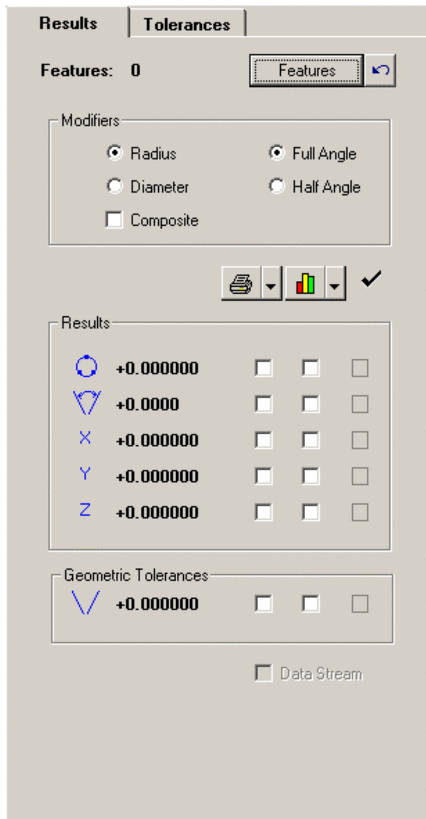
   After you select the minimum number of features, the Measurement window displays the Cone Construction Results.
5. If you want to construct a composite cone, select the **Composite** check box.
6. If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.
   - If you do not enter any nominal and tolerance values for the diameter and XYZ location, the reported diameter is at the midpoint of the centerline axis of the cone.
   - If you enter a nominal diameter and XYZ location, the reported diameter and XYZ location is at the point on the centerline axis of the cone that is closest to the nominal XYZ location.
   - If you enter a nominal diameter of 0, the actual XYZ location is at the apex.
7. Click OK after selecting the features.

The software displays a cone in the Model window as two circles, one each at the top and bottom, connected by lines at the 0, 90, 180, and 270 degree points on the circles. One circle is smaller than the other.

9.29 Cone Construction Results

The Construct Cone function reports the orientation, location and size of a cone, calculated from the selected features.

> Measurement results appear in the window when the minimum number of required points or features is entered.



The Measurement window displays:

- The number of points or features used in the measurement. You can edit or delete the entered points or features.
- Modifiers, which allow you to select how the cone is calculated:
  - Full Angle or Half Angle of the taper. This value is always positive.
  - Diameter (default) or Radius
  - Composite check box, which indicates how the software calculates the constructed cone. This appears only if the cone is constructed.
- Results, which include best-fit diameter or radius at the center of the cone, angle of the taper, and XYZ location of the midpoint of the cone's centerline axis.
  - If you do not enter any nominal and tolerance values for the diameter and XYZ location, the reported diameter is at the midpoint of the centerline axis of the cone.
  - If you enter a nominal diameter and XYZ location, the reported diameter and XYZ location is at the point on the centerline axis of the cone that is closest to the nominal XYZ location.
  - If you enter a nominal diameter of 0, the actual XYZ location is at the apex.
- Conicity tolerance

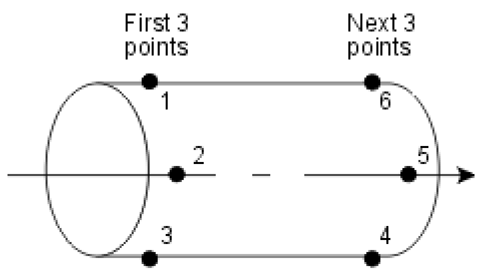
If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.

A cone is displayed in the Model window as two circles, one each at the top and bottom, connected by lines at the 0, 90, 180, and 270 degree points on the circles. One circle is smaller than the other.

9.30 How to Construct Cylinder

The cylinder function reports the size, location, orientation, and form of a cylindrical object, calculated from the selected features.

Measurement results and the OK button appear in the Measurement window when the minimum number of required features is selected. These can be previously measured or constructed features. For example, you can construct a cylinder from points, or from a previous cylinder or two circles if the Composite box is checked.



If you construct a cylinder from measured or constructed points, you must select at least six points, in two groups of 3 points each.

- Select three points near or at one end of the cylinder, which defines a circle or arc.
- Select three points near or at the other end of the cylinder, which defines a circle or arc.
- After the first six points, you can select other points between the two groups. For more accurate results, increase the distance between the two sets of points and increase the total number of selected points.
- The vector direction is from the first group of points to the second group.
- The line going through the two centers is used to calculate the cylinder's centerline axis.

- To display a partial cylinder in the Model window when Radius is selected:
  - Make sure that the paths of the two groups of points do not form complete circles.
  - The first point determines where the start angle begins, then the path goes through the second point, and the last point determines the end angle of the path.

To construct a cylinder:

1. [Classic User Interface] Click the **Construct** tab.
2. Click  in the toolbox or select **Construct** / **Cylinder** from the main menu.
3. Click a feature in the Model window to select it. The feature appears in a contrasting color (green).
4. Repeat the previous step to select the required number of points or the desired points. Click the Remove Features button if you want to remove a selected feature. To select multiple features at once, press and hold the Alt key while clicking and dragging in the Model window with the left mouse button. This allows you to draw a box around a group of features, all of which are selected when you release the mouse button.

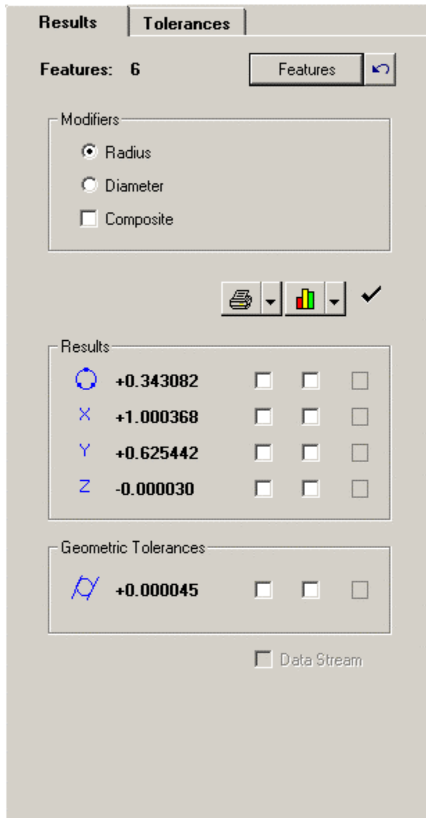
   After you select the minimum number of features, the Measurement window displays the Cylinder Construction Results.
5. If you want to construct a composite cylinder, select the **Composite** check box.
6. If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.

   If you do not enter any nominal and tolerance values for the diameter and XYZ location, the reported diameter value is at the midpoint of the centerline axis of the cylinder.
7. Click **OK** after selecting the features.

The software displays a cylinder in the Model window as two circles, one at each end, connected by dashed lines at the 0, 90, 180, and 270 degree points on the circles. Both circles are the same size.

9.31 Cylinder Construction Results

The Construct Cylinder function reports the size, location, orientation, and form of a cylindrical object, calculated from data points taken on the surface of the cylindrical object.



The Measurement window displays:

- The number of points or features used in the measurement. You can edit or delete the entered points or features.
- Modifiers, which allow you to select how the cylinder is calculated:
  - Diameter (default) or Radius
  - Composite check box, which indicates how the software calculates the constructed cylinder. This appears only if the cylinder is constructed.
- Results, which include best-fit diameter or radius at the center of the cylinder and XYZ location of the midpoint of the cylinder's centerline axis.

  If you do not enter any nominal and tolerance values for the diameter and XYZ location, the reported diameter is at the midpoint of the cylinder's centerline axis.
- Cylindricity tolerance

If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.

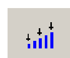
A cylinder is displayed in the Model window as two circles, one at each end, connected by lines at the 0, 90, 180, and 270 degree points on the circles. Both circles are displayed the same size.

9.32 How to Construct Min/Max/Avg Points

This function reports the minimum, maximum and average X, Y or Z points of the selected features.

Measurement results appear in the Measurement window when at least two features are selected. These can be previously measured or constructed features such as other lines, circles or points.

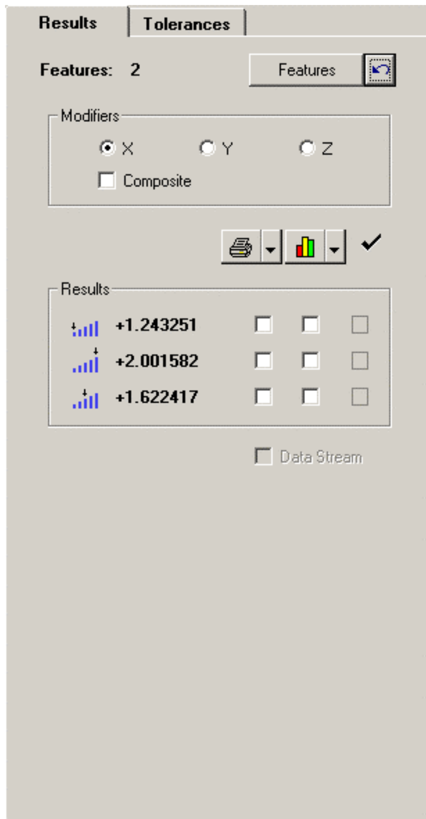
To construct the minimum, maximum and average points:

1. [Classic User Interface] Click the **Construct** tab.
2. Click  in the toolbox or select **Construct** / **Min**/**Max**/**Avg** from the main menu.
3. Click a feature in the Model window to select it. The feature appears in a contrasting color (green).
4. Repeat the previous step to select the other feature(s). Click the Remove Features button if you want to remove a selected feature. To select multiple features at once, press and hold the Alt key while clicking and dragging in the Model window with the left mouse button. This allows you to draw a box around a group of features, all of which are selected when you release the mouse button.

   After you select two features, the Measurement window displays the Min/Max/Avg Construction Results.
5. If you want, you can select the desired output destinations and the dimensions to be output, and you can click the Tolerances tab to enter nominal values and tolerances.
6. Click **OK** to complete the construction.

9.33 Min/Max/Avg Point Construction Results

The Construct Min/Max/Avg functions display the minimum, maximum and average value of the X, Y, Z, R, or A coordinates of a series of points. You must enter at least two points before any valid values are displayed.



The Measurement window displays:

- The number of points used in the measurement.
- **X, Y, Z** (or R, A) radio buttons
- **Minimum, Maximum, and Average** values of the selected coordinate

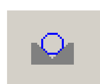
To display the minimum, maximum and average values, select the radio button next to the desired coordinate. You can also click another coordinate to display the values for that coordinate.

For example, when you specify three points on a vertical edge and select the Y coordinate, the minimum Y value is the one closest to zero and the maximum Y value is the one farthest away from zero.

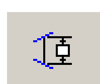
9.34 Gage Functions

The **Construct / Gage** menu includes the gage functions:

Gage Ball reports the center location of a circle of defined size that is tangent to two intersecting lines.



Gage Diameter constructs a line of defined length, perpendicular to the bisector of two intersecting lines, and reports the location of its midpoint.

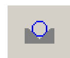
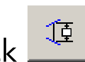


These functions are typically used to measure features such as the pitch diameter of threads and hydraulic parts.

You may select either measured or constructed lines to create gage locations. In either case, the two lines do not need to physically intersect but they **cannot** be parallel.

9.35 How to Use the Gage Functions

To use either gage ball or gage diameter:

1. Measure or construct the two reference lines before using the gage functions.
2. [Classic User Interface] Click the **Construct** tab.
3. Click  to select the Gage Ball function or click  to select the Gage Diameter function.

   The Measurement window displays the Radius / Diameter radio buttons, 1 of 2 / 2 of 2 gage location radio buttons, a field to enter the gage diameter or radius, the XYZ location, and the position tolerance of the selected gage.
4. Select the **Diameter** or **Radius** radio button.
5. Select the **1 of 2** radio button (if not already selected) to display the primary gage location. Select the 2 of 2 radio button if you want to select and display the mirror (secondary) gage location.
6. Click inside the box next to the gage function and enter a diameter or radius value.
7. Select the two desired lines the Model window. Select them in the order such that the gage ball or line appears in a counter-clockwise direction between the first line and the second line.

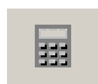
   The software displays a prompt to move the stage.
8. Click **OK** in response to the prompt.

   The software moves to the XYZ location and the field-of-view rectangle indicates the location where the ball or line will be.
9. Click **OK** to save the gage construction.

   The actual (primary) gage ball or line appears in the Model window.

9.36 Math Functions

The Math function enables you to perform mathematical operations to calculate a result, which can be saved in a math step. You can use measured values from other steps of the routine in the mathematical operations. You can also compare the calculated result with the nominal and tolerance values entered with the step.



The Math function is comprised of the following components:

- Math window, which contains:
  - Expression box where you can enter the mathematical expression.
  - Calculated result, which can be selected for output.
  - Buttons to calculate the result or clear an expression.
  - Valid arithmetic and trigonometric functions, and arithmetic and logical comparison operators.
  - Numerical pad for numeric entry.
- Variables window that pops up after you select a feature. It displays the number of the selected step along with the valid variables.

The following topics provide more detailed information about the Math function and the related Branch function.

- Mathematical Expressions describes different kinds of expressions and their syntax.
- Math Tags identifies the tags for all the eBx features.
- Arithmetic Functions describes the trigonometric and arithmetic functions.
- Operators describes the arithmetic and logical/comparison operators.
- How to Use the Math Function provides a step-by-step procedure with an example.

You can use the Math function as a "calculator" for mathematical operations.

9.36.1 Mathematical Expressions

The Math and Branch functions can use many different kinds of mathematical expressions.

A simple expression typically consists of a function and an argument. Arguments can be numeric values or eBx variables, which consist of math tags with step numbers. For example, in the expression "TAN (ANG5)", TAN is a trigonometric function and ANG5 is a variable argument. The result will be the tangent of the angle of the line measured in Step 5.

A complex expression may also include arithmetic and logical/comparison operators. For example, the expression "SQT (X5 - 0.1) ^ 2" means the square root of the argument in which a value of 0.1 is subtracted from the X value in Step 5, and that value is squared.

You can create a valid expression in any combination of the following ways:

- Place the cursor in the Expression box and type the entire expression or parts of the expression.
- Select functions, arithmetic operators and logical/comparison operators from the Math or Branch window.
- Select variables by clicking on features in the Model window.

When creating a mathematical expression, you must always type the alphabetic characters from the keyboard. You can enter numeric values from the keyboard or by using the numerical pad.

9.36.2 Mathematical Expression Syntax

Mathematical expressions must have the proper syntax. Some guidelines are listed below.

- Spaces are not required but they can be used for readability.
- In a variable, a math tag precedes the step number. Example: DIA5.
- All functions automatically insert an open parenthesis "(" character. You must enter a close parenthesis ")" character when you complete the expression.
- When using parentheses, you must have an equal number of open and close parentheses. Example: AVG(X1,X2).
- Trigonometric functions can have only one argument (see *Arithmetic Functions*). Example: SIN(ANG3).
- The AVG, MIN, and MAX functions typically have at least two arguments. Multiple arguments must be separated by commas. Example: MAX(DIA2, DIA3)
- Commas, which are required only with the Min, Max and Avg functions, can be specified in two ways:
  - Right-click in the Model window.
  - Enter it from the keyboard (the mouse cursor must be active in the Expression box).
- If you need to continue an expression, you must complete the expression in one step and refer to it (using the result) in the next step.
- **Calc** calculates the result of the expression
- **CE** (clear expression) removes the entire expression.
- **C** (clear) removes the expression and the result.
- **Back** removes the last entry in the Expression box. It acts like the Backspace key on a keyboard.

9.36.3 Math Tags

Tags are labels or identifiers for measurement features. The math tags associated with the eBx features are listed below. These tags consist of a maximum of three characters and are used in argument variables. The tags, and their definitions, are shown in the following list.

A - A axis location

ABT - Angle between

AGL - Angularity

ANG - Angle

ANA - Angle 1

ANB - Angle 2

ANC - Angle 3

AND - Angle 4

AVG - Average

CND - Cone diameter

CNR - Cone radius

CON - Concentricity

CYD - Cylinder diameter

CYL - Cylindricity

CYR - Cylinder radius

DIA - Diameter

FTA - Full taper angle

FLT - Flatness

HTA - Half taper angle

INA - Inclination angle 1

INB - Inclination angle 2

INP - User input

MAX - Maximum

MIN - Minimum

MND - Minimum diameter

MNR - Minimum radius

MNW - Minimum width

MXD - Maximum diameter

MXR - Maximum radius

MXW - Maximum width

PAR - Parallelism

PER - Perpendicularity

PHD - Height/depth

PRM - Profile -

PRP - Profile +

R - R axis location

RAD - Radius

RES - Math result

RND - Roundness

SLD - Straight line distance

SMA - Supplementary angle

SPD - Sphere diameter

SPR - Sphere radius

STR - Straightness

TDD - 3D distance

WCA - Width centerline angle

WID - Width

X - X axis location

XD - X distance

Y - Y axis location

YD - Y distance

Z - Z axis location

ZD - Z distance

9.36.4 Arithmetic Functions

The list below identifies the available math and branch functions and shows the required arguments.

ABS (absolute value)

One argument, ranging from a numeric value to a complex expression

AVG (average)

Two or more arguments, separated by commas

MIN (minimum)

Two or more arguments, separated by commas

MAX (maximum)

Two or more arguments, separated by commas

SQT (square root)

One argument, which must be a positive value

SIN (sine)

One argument, which must be an angle

COS (cosine)

One argument, which must be an angle

TAN (tangent)

One argument, which must be an angle

LOG (logarithm)

One argument, ranging from a numeric value to a complex expression

INV (inverse)

Sets the Inverse function for sin, cos, tan, log, and avg. The Inv function is automatically turned off after the arithmetic function is used.

ASN (arc sine)

One argument, which must be a value between -1 and +1

ACS (arc cosine)

One argument, which must be a value between -1 and +1

ATN (arc tangent)

One argument, which must be a value between -1 and +1

ILG (inverse logarithm)

One argument, ranging from a numeric value to a complex expression

The following apply to the Branch functions only:

IN

In tolerance (uses only one variable)

OUT

Out of tolerance (uses only one variable)

MISsed

Missed (uses only a single step)

9.36.5 Operators

The arithmetic and logical/comparison operators are displayed in the Math and Branch windows.

The two lists below list the operators in their order of precedence, from highest to lowest. In Branch functions, the *arithmetic operators are evaluated before the logical/comparison operators*.

Arithmetic Operators (used with Math and Branch)

(Open parenthesis

) Close parenthesis

^ Raise to a power

* Multiply

/ Divide

+ Add

- Subtract

Logical/Comparison Operators (used only with Branch)

< Less than

> Greater than

<= Less than or equal to

>= Greater than or equal to

& And

| Or

= Equal to

!= Not equal to

Selecting PI inserts a numeric constant, 3.14159, in an expression.

9.36.6 How to Use the Math Function

A sample procedure for using the Math function is provided in the example below. In this example, the Math function is used to calculate a position tolerance that can then be sent to the statistics file.

It is assumed that the datum features and datums have been done in Steps 1 to 4 of the sample routine. Step 5 contains a hole measurement with a nominal X location of 0.1, a Y location of 0.75, and a Z location of 0.0.

1. Select **Construct** / **Math** from the main menu.

   The Math step (6) appears with a blank Expression box and the Model window is displayed.

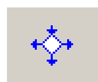
2. Click the **square root** button. "SQT(" appears in the Expression box.
3. Click **(** to begin the first variable.
4. Select Step 5 by clicking the circle in the Model window. This displays a list of variables in a pop-up window.
5. Click the X value in the Variables pop-up window. X5 appears in the expression.
6. Click the **-** operator. If you want, you can also type it in the expression.
7. Type **.1)^2** in the expression. This completes the first variable. The expression at this point is SQT((X5-.1)^2. If you want, you can click the **)** and **^** operators.
8. Click the **+** operator and **(**.
9. Repeat Step 4 and click the Y value in the Variables window.
10. Click the **-** operator.
11. Type **.75)^2** in the expression. This completes the second variable. The expression at this point is SQT((X5-.1)^2 + (Y5-.75)^2.
12. Click **)** to complete the square root function in the expression.

13. Click the ***** operator and type **2** at the end of the expression.
    - The completed expression is SQT((X5-.1)^2 + (Y5-.75)^2)*2.
    - (This is the formula for calculating a distance between two points, multiplied by 2.)
14. Click the **Calc** button in the Measurement window.

    The position tolerance appears under Result in the Math window.

9.37 Branch Function

The Branch function enables you to skip one or more steps and go to a specific step in a routine. This is done using IF-THEN-ELSE Go To conditional statements. The condition may be based on measured results or on calculated values from a mathematical expression.



The Branch function is comprised of the following components:

- Branch window, which contains:
  - Expression box where you can enter a function or mathematical expression.
  - Conditional statements with fields for the steps of a routine
  - Buttons to calculate the result or clear an expression.
  - Valid arithmetic and logical/comparison operators.
  - Numerical pad for numeric entry.
- Variables window that pops up after you select a feature in the Model window. It displays the number of the selected step along with the valid variables.

The following topics provide more detailed information about the branch function and the related math function.

- Mathematical expressions describes different kinds of expressions and their syntax.
- Math tags identifies the tags for all the eBx features.
- Arithmetic functions describes the trigonometric and arithmetic functions.
- Operators describes the arithmetic and logical/comparison operators.
- How to use the branch function provides step-by-step procedures in two examples.

⚠ Do not specify the current step number in either Go To statement. This will result in an infinite loop.

⚠ Be very careful if you specify a step number in either Go To statement that is before the current step number. Branching backwards may result in an infinite loop.

⚠ Be very careful about branching past a datum reset step. Since all measurement locations are based on the current datum set, skipping the datum set will most likely result in invalid location measurements.

> 🗎 You must click the **Calc** button in order for the OK button to become active.
>
> 🗎 If a Branch step goes around a datum step when branching to the specified step, the software uses the current datum frame for the specified step. The values of the skipped datum step have no effect on the specified step because the skipped datum step was not executed.

9.37.1 How to Use the Branch Function

Sample procedures for using the Branch function are provided in two examples below.

- Example 1 provides a procedure to branch to the end of a routine to skip empty cavities on a fixture.
- Example 2 provides a procedure to end the run of a routine if a critical dimension is not measured or is out of tolerance.
- Example 3 shows how the Branch function is used to branch around datums.

9.37.2 Branch Example 1

In this example, the Branch function is used to go to the end of the routine when an edge is missed due to an empty cavity on a fixture; the routine then measures the next part. It is assumed that Continue Run after Missed Edges is checked in the Run Options when you run the routine.

1. Select **Construct** / **Branch** from the main menu.

   The Branch step (2) appears with a blank Expression box and the Model window is displayed.

2. Click **Miss** to select the Missed Edge function. "MIS(" appears in the Expression box.

3. Type in the step number for the control feature, for example, **1** to represent Step 1.

4. Click **)** to complete the expression.

   The completed expression is MIS(1).

5. Type **9999** in the *THEN* Go To Step field. This represents the last step at the end of the routine.

6. Click the **Calc** button and then click **OK**.

7. Write the remaining steps of the routine and then run it. Remember to select the **Part Repeat** and **Continue Run after Missed Edges** check boxes in the Run Options.

When you run the routine, it will go to the end of the routine if an edge is missed in Step 1. Otherwise it will run through all the other steps in the routine.

9.37.3 Branch Example 2

In this example, the Branch function is used to go to the end of the routine when a critical dimension is missed or out of tolerance. It is assumed that the datum features and datums have been done in Steps 1 to 4 of the sample routine. Step 5 contains a critical diameter measurement.

1. Select **Construct** / **Branch** from the main menu.

   The Branch step (6) appears with a blank Expression box and the Model window is displayed.

2. Click **Miss** to select the Missed Edge function. "MIS(" appears in the Expression box.
3. Type **5** in the expression to specify Step 5.
4. Click **)** to complete the expression. The completed expression at this point is MIS(5).
5. Click **Or** to specify the OR operator.
6. Click **Out** to select the Out of Tolerance function. "OUT(" appears at the end of the expression.
7. Select Step 5 by clicking the appropriate feature in the Model window. This displays a list of variables in a pop-up window.
8. Click the **DIA** value in the Variables pop-up window. DIA5 appears in the expression.
9. Type **9999** in the *THEN* Go To Step field. This represents the last step at the end of the routine.
10. Click the **Calc** button and then click **OK**.

When you run the routine, it will stop if an edge is missed in Step 5 or if it is out of tolerance.

9.37.4 Branch Example 3

Sometime parts vary enough to be outside the field of view (FOV) at low magnification. In this case, it may be necessary to create a routine that can find a part that varies by a FOV. To measure the FOV, drag the boundaries of the manual box target to their extremes. Then read the X and Y values shown in the lower left corner of the FOV.

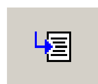
The print listing below shows how the Branch function is used to branch around datums. The print listing is from a sample routine created to set a datum on a part that varies by a FOV in the X direction.

To allow the routine to continue when a feature is missed, enable Continue Run After Missed Edges.

Step	Feature	X/R Location	Y/A Location	Z Location	Size	Reference
1	Point	+0.000000	+0.000000	+0.000000		
Constructed point to reset the origin to.						
2	Centroid	+0.000000	+0.000000	+0.000000	+0.006913	
First Centroid looks for the feature in the first FOV.						
3	Branch Expression: MIS(2)					6 4
If the feature was not found in Step 2, go to Step 6. If the feature was found, go to Step 4.						
4	Origin	+0.000000	+0.000000			2
The feature was found, therefore, set it as the origin.						
5	Branch Expression: 1=1					13 13
Feature was found, therefore, go to Step 13.						
6	Origin	+0.000000	+0.000000			1
Feature was not found, reset origin onto Step 1, the constructed point.						
7	Centroid	+0.400000	+0.000000	+0.000000	+0.006913	
Second Centroid looks for the feature in the second FOV.						
8	Branch Expression: MIS(7)					11 9
If the feature was not found in Step 7, go to Step 11. If the feature was found go to Step 9.						
9	Origin	+0.000000	+0.000000			7
The feature was found, therefore, set it as the origin.						
10	Branch Expression: 1=1					13 13
The feature was found therefore go to Step 13.						
11	Point	+0.000000	+0.000000	+0.000000		
Prompt: Please reseat the part. Routine will start again.						
The feature was not found in either search. Reset the part.						
12	Branch Expression: 1=1					2 2
The feature was not found, therefore the routine will automatically start again.						
13	Point	+0.000000	+0.000000	+0.000000		
Location of the found feature on the part.						

9.38 User Input

This function enables you to enter text and data during the run of a routine. For example, you can enter part measurements made on a different instrument, such as surface finish and hardness. The entered data and/or text become part of the output.



When the routine is run, the system pauses the run and displays a prompt. You need to enter the text and/or data and then continue the run.

See the following procedures for entering text and data:

- User Input When Creating the Routine
- User Input When Running the Routine

9.38.1 User Input When Creating a Routine

You can enter the following text or data in a separate user input step when you create the routine or any time you edit or change the routine:

- A feature prompt of up to 60 characters. The system displays this prompt when you run the routine.
- The nominal value as well as the upper and lower tolerance values.

To enter the text and/or data, follow the steps below.

1. Select **Construct** / **User Input** from the main menu.

   The software displays the User Input step.

2. Enter the desired values.
3. Click the **Results** tab.
4. If you want to enter a prompt or report text, click the **Prompt** / **Text** button and type the text.
5. If you want to output the entered information, select the desired output and select the appropriate check box.
6. Click **OK**.

9.38.2 User Input When Running a Routine

You can enter the following text or data when you run the routine:

- Any text of up to 60 characters. The entered text can also be sent to stats using the @P output character.
- An actual measurement value. The entered value can also be used in another step such as Math or Branch.

During the run of the routine, the system pauses when it gets to the User Input step and it displays a User Input window. To enter the text and/or data, follow the steps below.

1. Type the desired text.
2. Enter the actual measurement value.
3. Click **OK**.

The routine continues with the run and the entered text and/or data appears in the output.

9.39 Data Extraction

The data extraction function enables you to construct a feature (Line, Circle, Plane, Sphere, Contour, Cone, Cylinder) from a subset of points taken from one or more measured reference features or composite features.

You can select one or more subsets in a reference feature in the Model window. You can select subsets from one feature or multiple features. Each subset is considered a "feature" and increments the Features counter in the constructed step.

Typically data is extracted from features that have many points. For example, it is common to construct lines and circles from contours that were measured with an Edge Trace or Laser Scan tool.

> If you copy an extracted feature, you may also need to copy the reference feature; otherwise the copy may cause an error to occur.

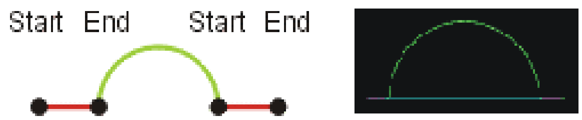
9.39.1 How to Extract Data

The data extraction function enables you to construct a feature (Line, Circle, Plane, Sphere, Contour, Cone, Cylinder) from a subset of points taken from one or more measured reference features or composite features.

To extract data from one or more reference features, follow the steps below. It is assumed that you have measured a feature that has many points, e.g., Contour.

1. Select the feature you want to construct, e.g., Circle. The software displays the Model window.
2. Decide which features you want to extract. This will affect how they are extracted. For example:
   - Control-Click in pairs to extract data points in subsets.
   - Click a feature to extract all the data points in that feature.
   - Make sure you extract the subsets in the same order (direction) in which the points were specified, e.g., left to right, clockwise, counter-clockwise. (See *How Data Extraction Is Done* for more information.)
3. Control-Click the Start point to begin creating a subset. The software highlights the entire feature in the Model window, automatically selects the Composite check box, and the cursor changes to a cross.
4. Control-Click the End point to finish creating a subset. The software creates and draws the subset in the picked color, changes the cursor back to normal, and increments the Features counter in the step. The software also grays out the Composite check box when at least one subset is defined.
5. Repeat Steps 3 and 4 if you want to select other features or subsets.
6. Click **OK**. The constructed feature is displayed in the Model as dashed lines and the subsets are no longer highlighted.

When you click a constructed step to edit it, all of its reference features and subsets are highlighted and the feature itself turns to a bright blue.



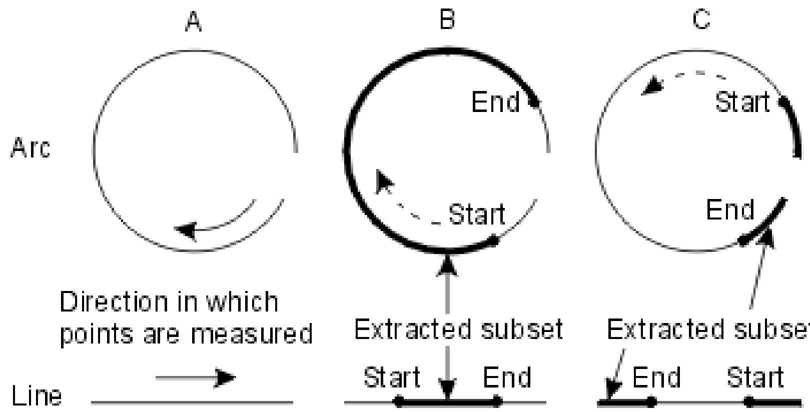
- To perform a Control-Click, press and hold the **Ctrl** key on the keyboard, press the left mouse button, and then release the **Ctrl** key.

- If the extracted subset is not what was expected, reverse the order of the Start and End points.

- If you copy an extracted feature, you may also need to copy the reference feature; otherwise the copy may cause an error to occur.

9.39.2 How Data Extraction Is Done

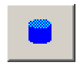
The illustration below shows how the software extracts data when you use the data extraction function to specify the Start and End points of a subset for an arc and a line.



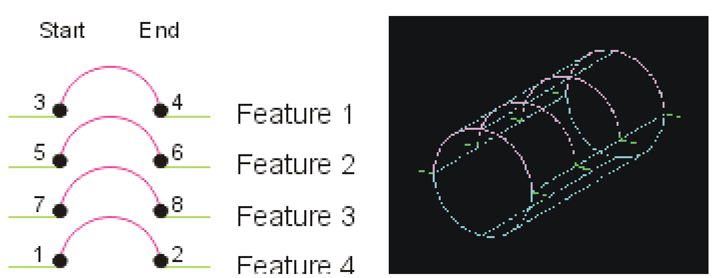
- **Figure A**: The data points are measured in a clockwise direction for the arc and a left-to-right direction for the line.
- **Figure B**: When you specify the Start and End points in the direction in which the data points are measured (shown by a dashed arrow), the extracted subset is also created by going in the same direction.
- **Figure C**: When you specify Start and End points in the opposite direction (shown by a dashed arrow) from which the points are measured, the extracted subset is still created by going in the direction in which the data points are measured (shown in Fig. A) rather than the direction in which the Start and End points were specified. If this is not what was expected, the Start and End points should be specified in the opposite direction, i.e., the same direction in which the points are measured.

9.39.3 How to Extract a Cylinder from Multiple Features

In the example below, use the data extraction function to construct a cylinder from four subsets of points taken from four measured Contour steps.

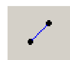
1. [Classic User Interface] Click the **Construct** tab.
2. Click  in the toolbox or select **Construct** / **Cylinder** from the main menu.

   The software displays the Model window.
3. Control-Click the Start point of the first curve segment to begin creating a subset.
   - The first subset needs to be at one end of the cylinder.
   - The software highlights the entire feature in the Model window, automatically selects the Composite check box, and the cursor changes to a cross.
4. Control-Click the End point of the segment to finish creating a subset. The software creates and draws the subset in the picked color, changes the cursor back to normal, and increments the Features counter in the step. The software also grays out the Composite check box.
5. Repeat Steps 2 and 3 to create subsets from the other three contours.
   - The second subset needs to be at the other end of the cylinder.
   - You can create the remaining two subsets in any desired order.
6. Click **OK**. The constructed cylinder appears in the Model as dashed lines and the subsets are no longer highlighted.

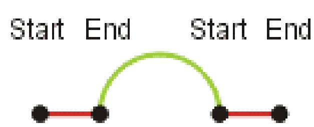


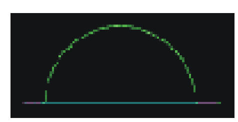
9.39.4 How to Extract a Line from One Feature

In the example below, use the data extraction function to construct a line from two subsets of points taken from one measured Contour.

1. [Classic User Interface] Click the **Construct** tab.
2. Click  in the toolbox or select **Construct** / **Line** from the main menu.

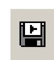
   The software displays the Model window.
3. Control-Click the Start point of the first line segment to begin creating a subset. The software highlights the entire feature in the Model window, automatically selects the Composite check box, and the cursor changes to a cross.
4. Control-Click the End point of the segment to finish creating a subset. The software creates and draws the subset in the picked color, changes the cursor back to normal, and increments the Features counter in the step. The software also grays out the Composite check box.
5. Repeat Steps 2 and 3 to create a subset from the second line segment.
6. Click **OK**. The constructed line is displayed in the Model as dashed lines and the subsets are no longer highlighted.





10. Running the Current Routine

10.1 Overview

To run the current routine, click  in the toolbox or select **File** / **Run** from the main menu. The Run Routine window displays instructions to do the part setup before you run the routine and the current run number, which is incremented each time the routine is run.

If you are running the routine the first time after you load it, the system may display the setup instructions automatically. This occurs only if the Display_Setup_Instructions parameter in INPUT.INI has a value of 1. The default is 0. To access this parameter, select System / Configuration / Editor from the main menu, and go to the Graphics section.

Before you run the routine, be sure you have the same part setup as when you created the routine.

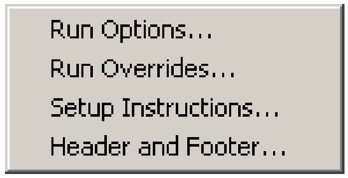
If you did the setup and you want to use the default options, click **OK**. The routine runs through every step displaying the measurements that you did.

If the setup is not correct for the current part, do it now before clicking OK.

If you want to change the run number, specify the desired number before clicking OK. For example, you may want to do this if the last run was stopped before it was completed.

You can access the run options, run override options, setup instructions and report header/footer information in the Part Setup menu.

10.2 Part Setup Menu

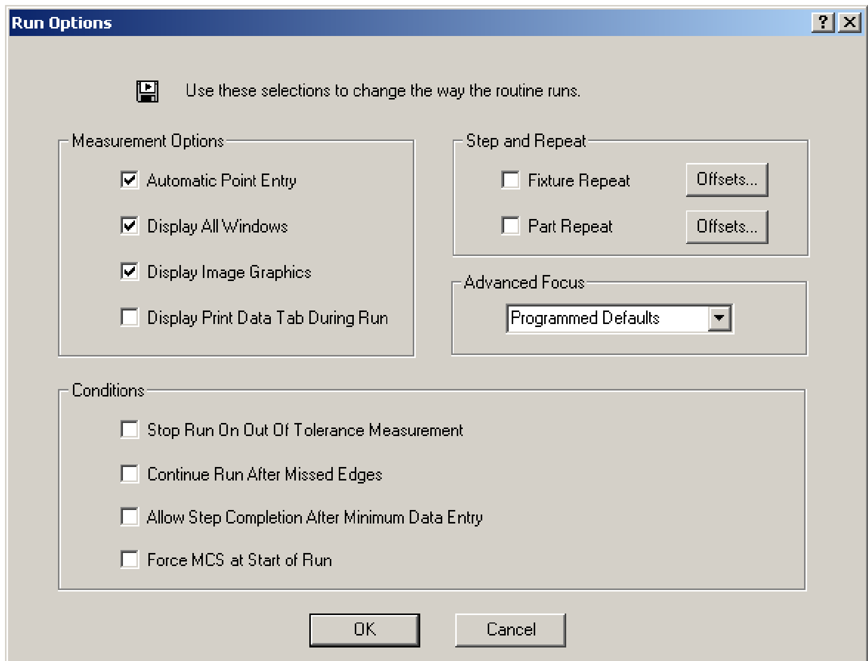


You can perform the following functions from the Part Setup menu:

- Select the desired run options and run override options before you run the routine.
- Enter setup instructions when you do the part setup.
- Type the header and footer that appear at the beginning of any report when you print routine.

10.3 Routine Run Options

Select **Part Setup** / **Run Options** from the main menu if you want to specify certain settings when you run the routine. Then select the check box next to each option that you want to select. Clear the check box to deselect it.



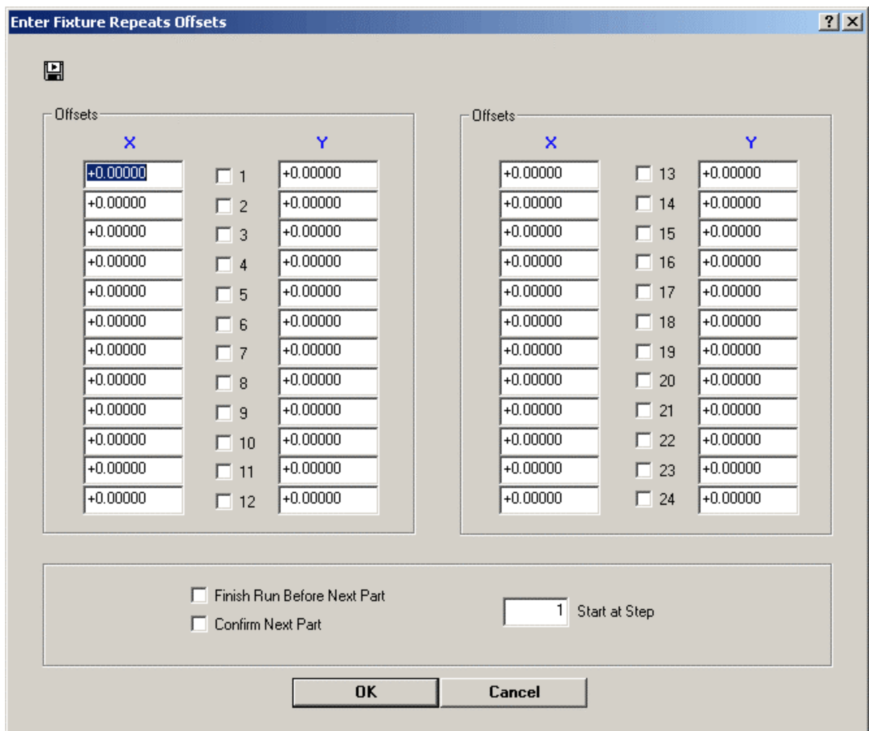
You can select any of the following options:

- Measurement options:
  - **Automatic Point Entry**. This activates the automatic entry of Strong Edge and Focus points as well as rotary indexer safe points.
  - **Display All Windows**. This allows you to display or not display the Measurement window during the routine run. The routine runs slightly faster if you do not display the window.
  - **Display All Image Graphics**. This allows you to display or not display the graphics that appear in the Image window, e.g., tools, during the routine run. The routine runs slightly faster if you do not display the graphics.
- Step and Repeat (These options are mutually exclusive. You can use one option or the other but not both at the same time.):
  - Fixture Repeat
  - Part Repeat
- Conditions:
  - Stop Run On Out Of Tolerance Measurement check box. If you select this option, the system stops the run if a feature is out of tolerance and puts up a confirming window. You may then click one of the following buttons:
    - **OK** to stop the run
    - **Cancel** to continue with the run
    - **Change to Step** to step edit the routine
  - Continue Run After Missed Edges
  - Allow Step Completion After Minimum Data Entry
  - Force MCS at Start of Run

10.3.1 Fixture Repeat

This routine run option is typically used when measuring multiple samples of the same part that are mounted in a fixture. The fixture may have empty cavities and uneven offsets.

In the Routine Run Options, select the **Fixture Repeat** check box and click the **Offsets** button to enter the fixture repeat offsets. The software displays the Fixture Repeat dialog box.



After creating the routine to measure the first part, you may enter the location offsets of up to 24 additional parts from the Run Options menu. The parts do not need to be spaced evenly; you can enter a unique offset for each additional part. The offsets are from the original part. The offsets are saved with the routine.

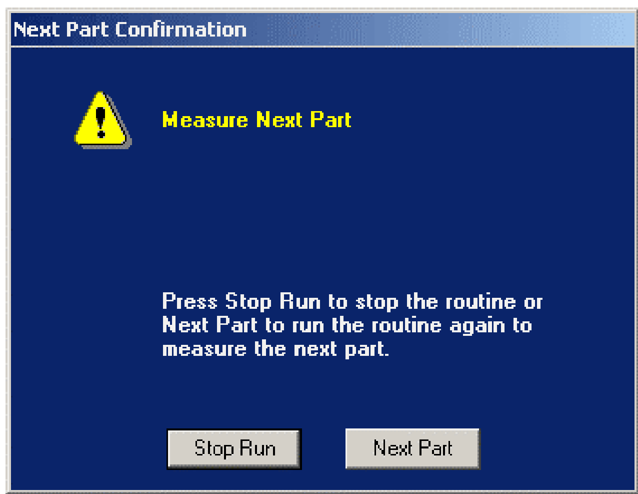
A check mark in the box next to the offset indicates that the offset is active. Fixture locations with offsets that are not active are not measured. That way, the fixture does not have to be fully loaded. However, the offset values are retained for future use.

To activate an offset, select the check box. If you want to turn off an offset, clear the check box.

If both the X and Y offsets are 0 and the box has a check mark in it, the software will measure the first part again.

Fixture Repeat has two additional check boxes and a Start At Step box:

- Finish Run Before Next Part. This option is used to modify the printer control and certain Stats templates. The default is to print the measurement results for all the parts in one report at the end of the measurement routine. When you check Finish Part Run, measurement results are printed after each part is measured, and the End of Run template in Stats is output after each part (rather than at the end of the entire routine), and the Begin of Run template precedes each part.
- Confirm Next Part. When this option is on, the routine stops after measuring a part. You can stop the run or continue to run to measure the next part.



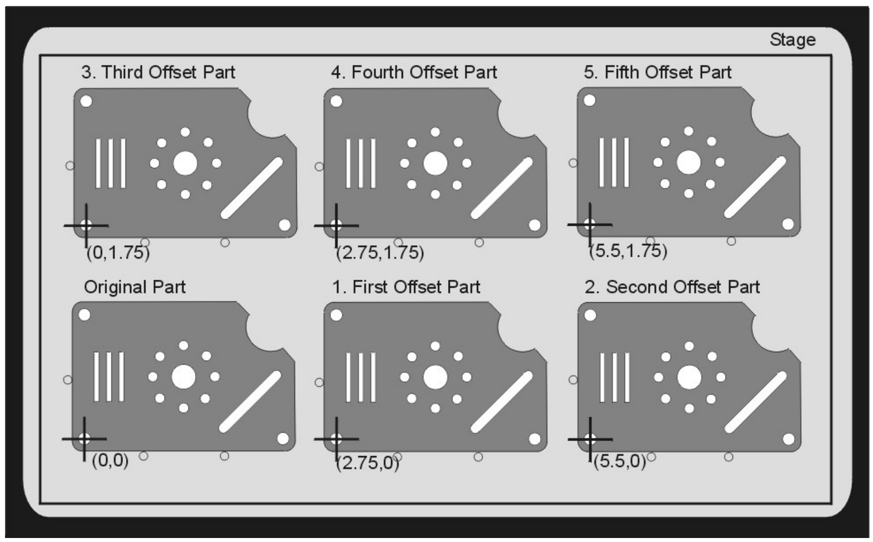
- Start At Step. The default is Step 1. If you want to start the repeat at another step, type the desired step number. Be careful that you do not skip any datum steps.

10.3.2 Part Repeat

This routine run option contains two offset entries to measure equally spaced parts in a fixture.

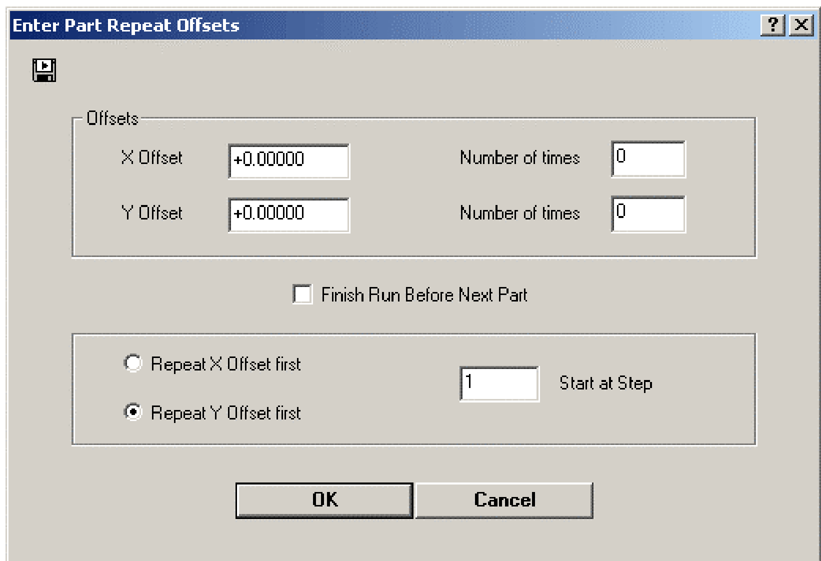
The parts can be measured in any orientation or direction. Up to 999 parts can be measured with Part Repeat.

After measuring the first part, you may enter the X and/or Y offsets. Each offset can be repeated 999 times. The offsets are from the original part. They are saved with the routine when you select File / Save.



To enter the part repeat offsets, follow the steps below.

1. In the routine run options, select the **Part Repeat** check box and click the **Offsets** button to enter the part repeat offsets. The software displays the Part Repeat dialog box.
2. Specify the X and/or Y offset value(s).
3. Specify the number of times each offset is to be repeated. If you want to re-measure the same part, leave the offset value 0 and simply specify the number of times.
4. Decide in which direction the parts are to be repeated first. The default is across the row first (the radio button next to "Repeat X offset first" is filled in). To repeat the parts in a column first, select the Repeat Y offset first radio button.



5. If you want to start the repeat at another step, type the desired step number in the Start At Step field. The default is Step 1.

Typically, the X offset is used to measure all the parts in one row. The Y offset indicates the number of additional rows to be measured. For example, to measure 18 parts (in 3 rows with 6 parts per row), measure the first part at the beginning of the bottom row. Then specify the X offset and 5 for the number of repeats. Then specify the Y offset and 2 for the number of repeats.

Part Repeat has an additional check box:

- Finish Run Before Next Part. This option is used to modify the printer control and certain Stats templates. The default is to print the measurement results for all the parts in one report at the end of the measurement routine. When you check Finish Run Before Next Part, measurement results are printed after each part is measured, and the End of Run template in Stats is output after each part (rather than at the end of the entire routine), and the Begin of Run template precedes each part.

10.3.3 Continue Run After Missed Edges

When the software encounters a missed strong edge or a focus fails during the run of a routine, it displays a warning message and waits for the user to select a valid edge or redo the focus to continue the run. For missed weak edge features or points, the system prompts the user to change the run to Step Edit.

This routine run option instructs the software not to display any message and not to wait for user input. The software continues with the next point in the feature. If this results in too few points to constitute a valid feature, the software reports the "bad" actual values based on the BAD_VALUE_DIGIT parameter in INPUT.INI. The default is 0. The "bad" actual values will be printed and/or sent to Stats. The software then continues with the next step. You can stop the run on missed tolerance features if you selected the Stop Run on Out of Tolerance Measurement run option.

10.3.4 Allow Step Completion After Minimum Data Entry

This run option is available for measurements that use manual alignment targets. It allows you to end the measurement during the run after the software has processed the minimum number of points for the feature (the OK button is enabled in the step), regardless of how many points were originally programmed.

With this option, the calculation is based only on the points actually measured and entered. The remaining points will be retained in the routine but they will not be used in the calculations.

This option works only if the Display Windows run option is ON.

10.3.5 MCS at Start of Run

This routine run option clears the current PCS or SCS and resets the system coordinates to MCS, as necessary, when you select Run or Step Edit. It is assumed that the routine was created in MCS.

When you start the measurement software, the check box defaults to an unchecked state.

If the check box has been checked, the software clears it whenever you select **File** / **New** or **System** / **Reset** from the main menu.

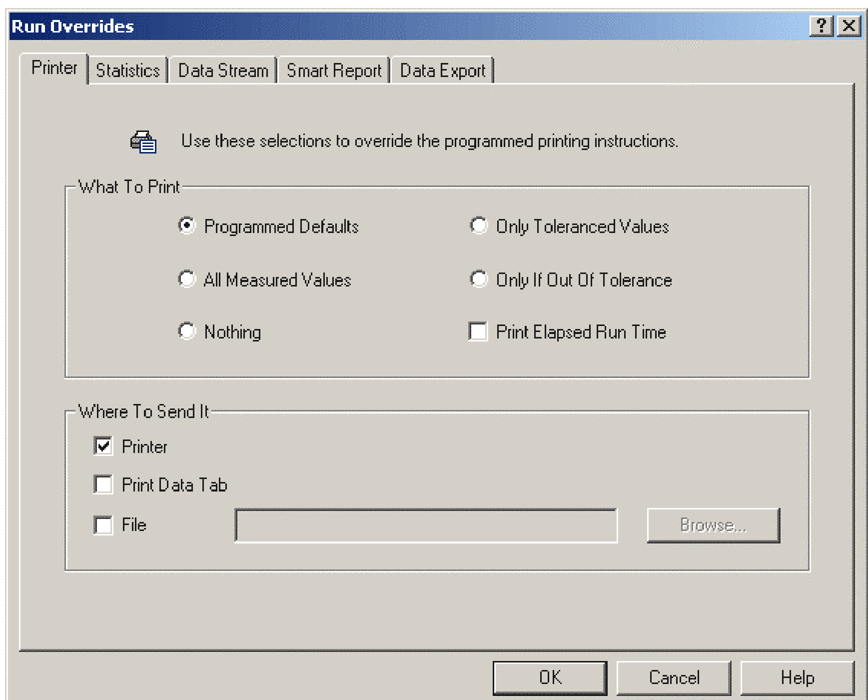
If you click the Cancel button in the Run window after the coordinates have been reset to MCS, the MCS will remain in effect, i.e., the software will not revert back to the previous coordinate system.

Be sure to create the routine using only machine coordinates in order to safely use this function.

10.4 Run Override Options

Select **Part Setup / Run Overrides** from the main menu to display a dialog box with the run override options that were used last.

- Printer Override Options
- Statistics Override Options
- Data Stream Override Options
- SmartReport Output Override Options
- Data Export Override Options

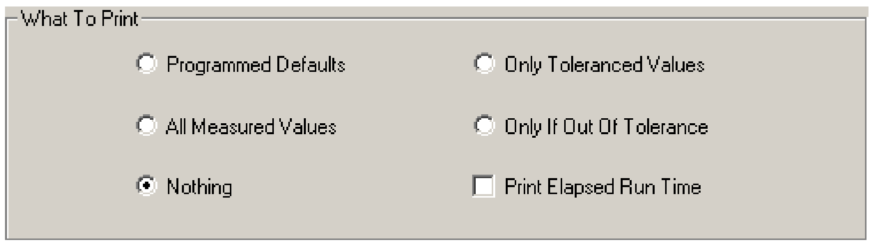


Click a tab to display the desired override options.

When the system displays the selected override options, each option has a radio button or a box next to it.

- If the option has a check box next to it, select the check box to select that option. Clear the check box to deselect it. You can select as many options as you want.
- If the option has a radio button next to it, you can select only one of the options in the group because they are mutually exclusive.

10.4.1 Printer Override Options



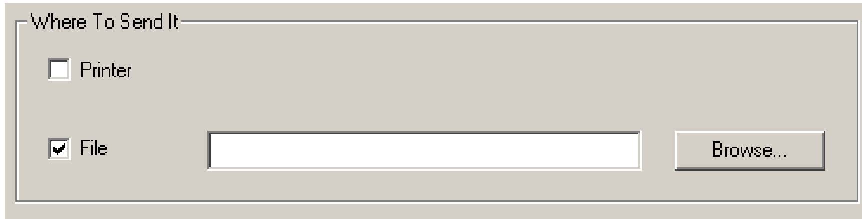
These options override any Printer settings for the dimensions or measurements in the individual steps.

You can select any of the following printer override options when you begin to run the routine.

- Programmed Defaults. This uses the Print settings in the steps.
- All Measured Values. All dimensions and measurements are sent to the printer.
- Nothing. No dimensions or measurements are sent to the printer.
- Only Toleranced Values. Only those dimensions and measurements that have nominal values and tolerances are sent to the printer.
- Only if Out of Tolerance. Only those dimensions and measurements that are out of tolerance are sent to the printer.
- Print Elapsed Run Time. If this option is checked, the software prints the elapsed time for the run of the routine.

The selected values will be output to the Print Data Destinations.

10.4.2 Print Data Destinations



In the Printer Override Options, you can specify up to two destinations for the print data (when you select dimensions to be printed in a step):

- Printer (this is the default setting).
  - When you are creating or editing a routine or the routine is in step edit mode, the checked dimensions are sent to the Print Data Output.
  - When you run the routine, the checked dimensions are sent directly to the printer and printed when the run is completed.
- Print to File. The checked dimensions are sent to a file.
  - When you check this box, you can type the name of a new or existing file. The default file type is PRT. If you click the Browse button, the software displays the standard "Save As" dialog box where you can change the path and file name.
  - The software overwrites the data if you use the same file each time you run the routine.
  - The override is turned off whenever you select System / Reset, File / New, or File / Open from the main menu.

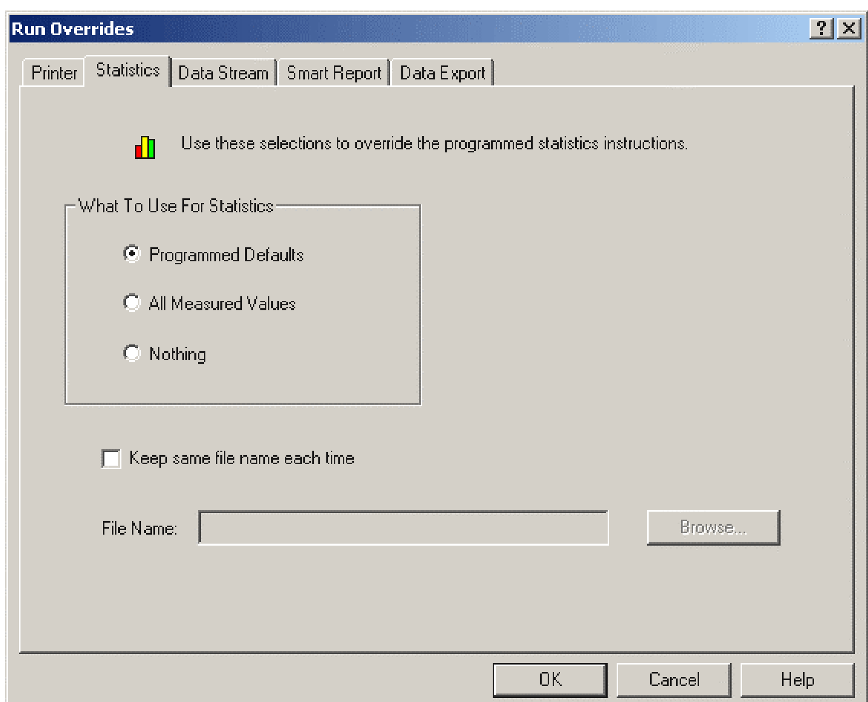
You can send the print data to any of these destinations during the creation and step edit modes, and when you run the routine.

10.4.3 Statistics Output Override Options

The options listed below override any statistics output settings for the dimensions or measurements in the individual steps. You can select any of the following statistics override options when you begin to run the routine.

- **Programmed Defaults**. This uses the statistics output settings in the steps.
- **All Measured Values**. All dimensions and measurements are sent to the statistics file.
- **Nothing**. No dimensions or measurements are sent to the statistics file.
- **Keep same filename each time**. When this option is checked, the software will not prompt you for a statistics file name when the routine is run. It will use the file name stored with the routine. The default file extension is TXT, which is specified in the statistics output configuration.

  If you want to change the file name before running the routine, check the box and type the desired file name in the File Name field. If you click the Browse button, the software displays the standard "Save As" dialog box where you can change the path and file name.



10.4.4 Data Stream Override Options

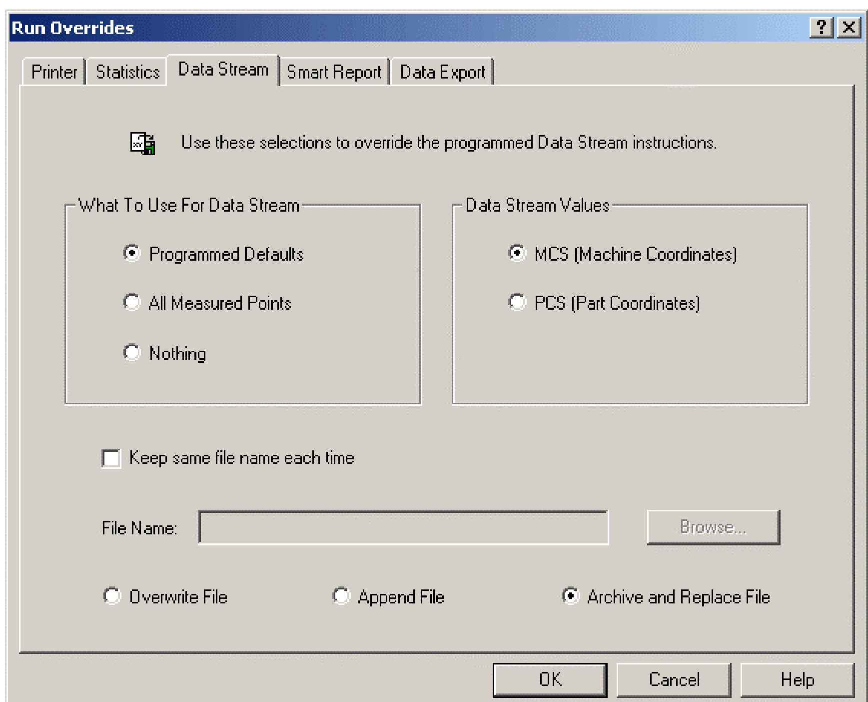
The options listed below override any data stream settings for the dimensions or measurements in the Measurement windows of the individual steps. You can select any of the following data stream override options when you begin to run the routine.

- Programmed Defaults. This sends the dimensions and measurements of the features whose data stream check boxes are checked in the Measurement windows.
- All Features. All valid dimensions and measurements are sent to the data stream file, whether or not the data stream check boxes are checked in the Measurement windows of selected features.
- Nothing. No dimensions or measurements are sent to the data stream file, even if the data stream check boxes are checked in the Measurement windows of selected features.
- Machine Coordinates (MCS). The measured points that are output are relative to stage home or to the part setup coordinates, i.e., the coordinates after the part setup and axis alignment have been done. This is the default for export to MeasureFit.
- Part Coordinates (PCS). The measured points that are output are relative to any datums that have been reset.
- Keep same filename each time. When you select this option, the software will not prompt you for a data stream file name when the routine is run. It will use the file name stored with the routine. The default file type is DAT.

  If you want to change the file name before running the routine, select the check box and type the desired file name in the File Name box. If you click the Browse button, the software displays the standard "Save As" dialog box where you can change the path and file name.

Three radio buttons allow you to select how the data stream file is saved:

- **Overwrite File** - If a DAT file of the same name already exists when running this routine, then replace or overwrite that existing file with the new data stream file and values.
- **Append File** - If a DAT file of the same name already exists when running this routine, and then add (append) the new data stream values to the existing file. This will result in a multiple series of data in a single file.
- **Archive and Replace File** - If a DAT file of the same name already exists when running this routine, the system renames the existing file with a numeric suffix or extension. For example, if SAMPLE.DAT already exists, the system renames it SAMPLE.001 and keeps it in the same directory. In subsequent runs, if *DAT and *001 exist, the system renames the existing *DAT to *002, and so on. The current run produces a new DAT file that remains in memory until the next run.

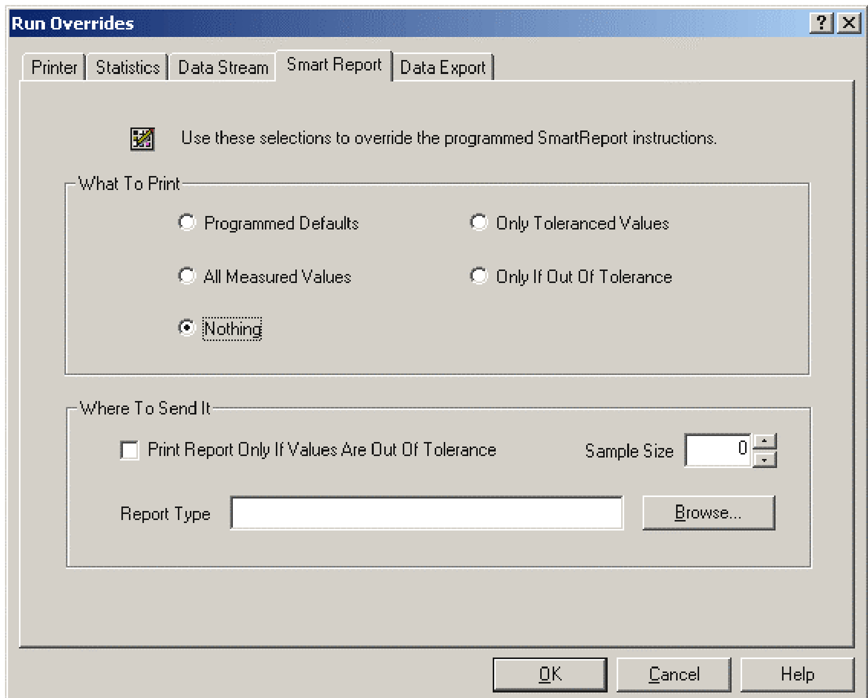


10.4.5 SmartReport Override Options

The options listed below override any SmartReport settings for the dimensions or measurements in the individual steps. You can select any of the following override options when you begin to run the routine.

- Programmed Defaults. This uses the SmartReport settings in the steps.
- All Measured Values. All dimensions and measurements are sent to the SmartReport file.
- Nothing. No dimensions or measurements are sent to the SmartReport REPORT.STA file.
- Only Toleranced Values. Only those dimensions and measurements that have nominal values and tolerances are sent to the SmartReport file.
- Only if Out of Tolerance. Only those dimensions and measurements that are out of tolerance are sent to the SmartReport file.
- Print Report Only if Values Are Out of Tolerance. If this option is checked, the software sends the data to the SmartReport file only if the resulting values are out of tolerance.
- Sample Size. This enables you to specify a different sample size.

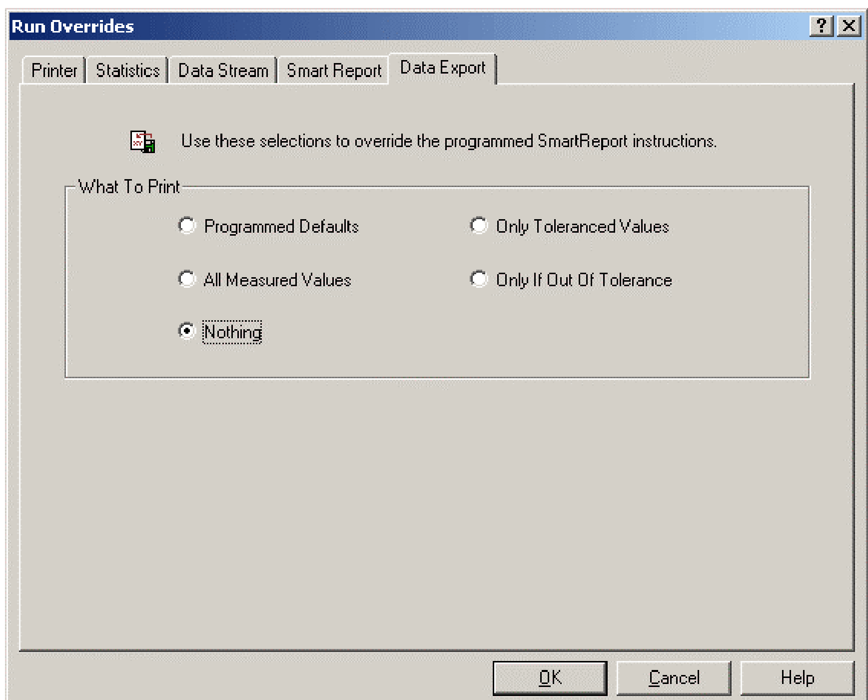
- **Report Template**. This enables you to specify a different file name and report type.
  - First article report (FST)
  - First article (five piece) report (FPF)
  - Raw data file (RAW)
  - Statistics summary (STT)



10.4.6 Data Export Override Options

The options listed below override any SmartReport Export settings for the dimensions or measurements in the individual steps. You can select any of the following override options when you begin to run the routine.

- Programmed Defaults. This uses the Export settings in the steps.
- All Measured Values. All dimensions and measurements are sent to the SmartReport file.
- Nothing. No dimensions or measurements are sent to the SmartReport EXPORT.STA file.
- Only Toleranced Values. Only those dimensions and measurements that have nominal values and tolerances are sent to the SmartReport file.
- Only if Out of Tolerance. Only those dimensions and measurements that are out of tolerance are sent to the SmartReport file.



10.5 Other Run Features

When a routine is run, the hourglass-shaped cursor changes to a boxed question mark whenever the routine is halted and the system is waiting for a point entry. The routine will continue when you enter a location.

The Model window displays all the features when a routine is running. The features that have not been measured yet are displayed with a dark gray color.

If the feature calculation fails while a routine is running, the system will switch to Step Edit to measure the feature unless you selected the Continue Run after Missed Edges run option. You can go back to the Run mode when you finish measuring the feature.

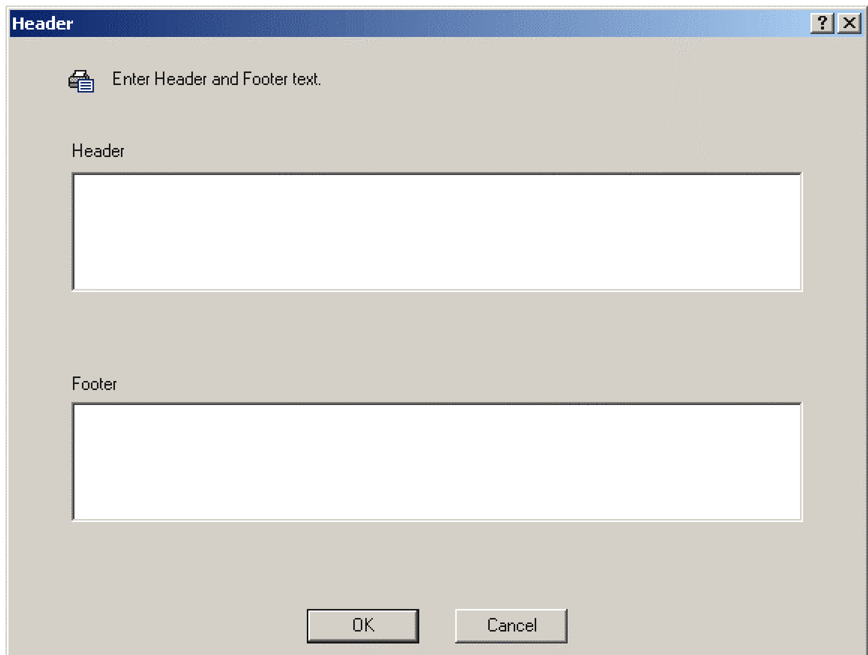
Programmed and actual point locations are separated when you run the routine. When creating a routine, the programmed and actual points are the same. When running a routine, the stage is driven to the programmed point location and the entered point is saved in the actual point location. Features are calculated from the actual point locations. The model is constructed from the calculated features. Step Edit will overwrite the programmed point locations with the actual point locations.

To edit the programmed locations click the Edit Points button in a measurement step and set the new locations. Remember that you need to resave the program when you are done editing.

If the statistics data "destination" is "to a file," the data is sent to a temporary file C:\QVSI\BASICX\TEMP.STA during the creation and run of a routine. The temporary file contains all the features sent during routine creation, not just the last feature. When the routine run is finished (before confirming the All Features Completed window), the statistics are appended to the specified statistics file or a new file is created.

10.6 Header and Footer on the Run Output

When you select **Part Setup** / **Header and Footer** from the main menu, a dialog box pops up where you can enter a header and footer that appears in the run output report. The cursor appears in the Header field.



The header appears at the beginning of the report and the footer appears at the end of the report.

To enter each line of text:

1. If you want to specify a footer, click in the Footer field.
2. Type the desired text. If you reach the end of the line, characters will wrap to the next line.
3. If you want to go to another line, press **Enter** on the keyboard and type the desired text.
4. Click **OK** or Cancel to close the dialog box.
    - OK saves the text and any changes/additions you made.
    - Cancel saves only your original text; any additions or changes are not saved

If you want to display the header when you click the **Run Routine** icon to run the routine, edit the configuration parameters and change the value of RUN_DISPLAY_HEADER to 1 in the Graphics section.

10.7 Save a Video Image During a Routine Run

This function allows you to save a video image as part of a step each time you run the routine.

The image is saved in a file in the Routines folder (default) or another folder that you may have specified. If you want to change the default path, modify the IMAGE_DIR parameter in the Paths section of the Configuration Editor.

The image can be saved in a TGA (default), BMP, JPG, or TIF format. If you want to change the default format, modify the OUTPUT_IMAGE_TYPE parameter in the Video section of the Configuration Editor.

An image is typically saved as a reference for comparisons with other identical features. For example, you can save an image of the master part or of a part with a known defect. You may also want to archive images of measured features for future use.

When the routine is run, the software saves the image at the point where it encounters a User Input step with Save Image instructions. It uses the file name that was specified or the routine name followed by the step number and index number, e.g., FASTART-12-1.TGA.

To have your program automatically save images during its run, select **System / Configuration / Editor** and go the **EMX General** section. There are three settings that control Auto Saving Images:

- **EMX_AutoSaveImage** must be set to 1 in order to enable AutoSave.
- **EMX_MaxNumImages** is set to any number between 1 and 1000; this is the maximum number of images you want to have AutoSaved.
- **EMX_ArchiveDirectory** is the directory to which the images are saved. This defaults to Archive when the software is installed; it must be changed to an existing directory before you run your program.

AutoSave will save your part image (without tools) each time you stop and take a measurement.

The size of each image file is approx. 900 KB. If the routine creates many files, this may quickly use up the available disk space resulting in a possible computer crash. It is extremely important that you track the available disk space. For example, you may want to store the image files on another device such as a networked drive or a writable CD-ROM. If you want to send the files directly to a networked drive, change the default path to point to the desired drive.

If the OUTPUT_IMAGE_DATA parameter in the Video section of the Configuration Editor is set to 1, a text file is written with each saved image that contains the XYZ stage position in current part coordinates, the rotation matrix, the XY pixel size in the image, and the XY center pixel location. The text file has the same name as the selected image file, but with a .TXT suffix.

10.8 How to Save an Image in a User Input Step

This function allows you to save a video image as part of a step each time you run the routine. The software saves the image at the point where it encounters a User Input step.

To save an image in a User Input step when creating a routine:

1. Select **Construct** / **User Input** from the main menu.

   The software displays the User Input step in the Measurement window.
2. Click the **Prompt** / **Text** button to display the Report/Prompt dialog box.
3. Type Save_Image in the User Prompt field. The "S" and "I" must be upper-case. You do not need to specify an extension; the system appends the selected output type extension (default is TGA) when the routine is saved.
4. If you want, add a space after Save_Image and type a filename; do not type the path. The software automatically appends a unique index number when you run the routine.
5. Click **OK** to close the dialog box.
6. Click **OK** to close the step.

⚠ The size of each image file is approx. 900 KB. If the routine creates many files, this may quickly use up the available disk space resulting in a possible computer crash. It is extremely important that you track the available disk space. For example, you may want to store the image files on another device such as a networked drive or a writable CD-ROM. If you want to send the files directly to a networked drive, change the default path to point to the desired drive.

- If you want to change the default path, modify the IMAGE_DIR parameter in the Paths section of the Configuration Editor.

- If you want to change the default image format, modify the OUTPUT_IMAGE_TYPE parameter in the Video section of the Configuration Editor.

- If the routine has not been saved yet, the root name of the file will be in the format YYYY-MM-DD-HR-MIN-SEC.

10.9 Autorun Function

The Autorun function enables you to load and run a routine automatically after starting the measurement software. This is typically used when an external application, such as MeasureMenu or an application created in Visual Basic, is controlling the measurement software. This may also be used to loop programs together when multiple parts of different types are placed in the same fixture. Two methods are available, depending on your fixturing and part setup method.

To use Autorun, a text file is placed in the Routines folder. The file must contain just one line of text with the full path and name of the routine to be run automatically. Example: file contains the text "C:\QVSI\BASICX\ROUTINES\routine2.bxi". This causes the software to automatically load the program routine2.bxi from the Routines folder.

- AUTORUN0.TXT. If a file of this name is found, the measurement software automatically loads the routine, but waits for the user to click OK before starting the inspection. This allows the user to perform a manual setup. The user is also required to click OK at the end of the run.
- AUTORUN1.TXT. If a file of this name is found, the measurement software automatically loads and runs the routine without any user intervention. The associated routine must use a part setup based on stage home.

The Autorun function is enabled by setting the AUTORUN_ENABLED parameter in the Autorun section of the Configuration.INI file to 1 (ON). The AUTORUN_TIME_CHECK parameter controls the frequency (in seconds) in which the software checks for the presence of the Autorun file. The default is 1 second. The AUTORUN_ERASE_AT_END parameter controls when the file is deleted - immediately after it is read, or at the end of the routine run. The default setting is 0 erase file immediately after reading.

Once the Autorun function is enabled, the software automatically looks for and loads the created Autorun text file. When it executes the routine specified in the Autorun file, the software deletes all the autorun text files. The controlling external application then rewrites the autorun text file to the Routines folder, and the sequence begins again.

10.10 Tips for Faster Measurements and Routine Runs

- Plan the order in which the features from the blueprint are measured and minimize stage movements between feature measurements.
- If you are using the Strong Edge tool, reduce the number of points to be measured for a given feature.
- Turn Display All Windows to off in the routine run options and turn the audio Pass signals to Off in System / Configuration / Sound.
- Avoid unnecessary magnification changes when creating a routine; change to another magnification only when it is necessary to do so.

10.11 Tips for More Repeatable and Accurate Measurements

- Always measure and set datums within the routine; do not depend on just the manual setup. Be sure to set the Z datum using a focus point.
- To ensure maximum repeatability when doing Z axis measurements with focus points, make sure that **all** the autofocus measurements in the routine use:
  - The **same** magnification (highest magnification using Maximum Contrast is recommended).
  - The **same** light source. This should be the light source that provides the highest contrast. For example, use surface illumination and do an autofocus on a surface.
  - The **same brightness** (light level).
- Edges and surfaces are located more accurately and repeatably at higher magnification settings. This is because the depth of field gets smaller as the magnification increases.
- Reset the Z axis to zero in a datum step (re-establish the Z datum) after you change the magnification and/or light source, if you plan to do Z axis measurements.
- Surface illumination and high magnification are recommended for:
  - Critical Z-axis measurements
  - Thick parts
  - Cylindrical parts
- The image may appear distorted or fuzzy (wall effect) when measuring thick, highly polished parts using profile illumination. It is recommended that you fixture the part above the stage glass to eliminate the wall effect.
- When using FeatureFinder, be precise in placing the selection points on the edge.
- When measuring difficult weak edges, use the weak edge weighting factors to "fine-tune" which points should be included in a measurement.

- Pay attention to the light level for each measurement. Light below 45% or above 70% can reduce accuracy and repeatability.
- Make sure that the parts to be measured are clean.

10.12 How to Stop the System

You can stop the system in the following ways:

- Click the **Stop / Start button** in the DRO window to stop the routine while it is running. The software completes the current step, stops the routine, and displays a confirmation prompt to end the routine. If you click Cancel, the software resumes the run.

  This is used most often when you specify a destination incorrectly in a routine. For example, if you forget to change the sign and the stage moves in a direction that is opposite to the desired destination, stop the stage and specify the destination correctly.

- Press the **Stop / Start button** on the joystick (if equipped) or control panel (if equipped) to:
  - Cut power to all motors, stop all the XYZ motion immediately, and interrupt the current measurement. The yellow Stop LED (visible through the optics cover) is turned on (steady glow). You **cannot operate** the XYZ, focus and zoom lens motors as long as the yellow Stop LED is on.
  - Take the system out of Stop mode when the yellow Stop LED is turned on (slow blinking) during the machine power up sequence.
  - Take the system out of Stop mode and resume (start) operation of the system again. The yellow Stop LED is turned off. When you do this, remember to press the Start/Stop button **before** you click OK in response to the displayed prompt.

- Press the **red E-Stop switch** to:
    - Cut power to all motors, cut the voltage to the surface and standard ring lights, stop all the XYZ motion immediately, and interrupt the current measurement. The yellow Stop LED (visible through the optics cover) is turned on (fast blinking). You **cannot operate** the XYZ, focus and zoom lens motors and you **cannot operate** the surface and standard ring lights as long as the yellow Stop LED is on. If it is necessary to view a feature, you may operate the programmable ring light or the back light.
    - To take the system out of E-Stop mode and resume operation, twist the red button clockwise to pull it out and press the Stop/Start button on the joystick.

⚠ Use the red E-Stop switch or the Stop/Start button on the joystick or control panel to prevent the accidental collision of the optics or accessories such as the touch probe, with an object (part or fixture) on the stage.

If a destination is beyond the maximum stage travel distance specified in the calibration parameters, the stage stops automatically.

11. Measurement Results & Output

11.1 Overview

Measurement results provide a way for you to evaluate the measurements and make the necessary decisions about the part being measured. Measurement results include actual measurements, specified nominal and tolerance values, and deviations.

After you measure a feature, the measurement results are displayed in the Measurement window. For example, this includes the number of points measured, the dimension of that feature (e.g., diameter for a circle) and the XYZ location of the feature.

The default setting is to display the measurements and not to output the measurement results. If you want to output a measurement, see:

- Measurement Output Options and Destinations
- Use the measurement buttons displayed at the bottom of the Measurement window to perform an action in the window.

🗎 Entering additional points recalculates the results. You can enter a maximum of 99,999 points.

11.2 Pass/Fail Indicators

All Measure and Construct windows have pass / fail indicators. If the feature is toleranced (see *Nominal Values and Tolerances*), one of the following indicators appears next to the measurement value:

- Green indicates in tolerance
- Yellow indicates in tolerance, but using 75% or more of the tolerance band
- Red indicates out of tolerance

If you want, you can change the 75% threshold value to any number below 100%. To do this, edit the INPUT.INI file (see System / Configuration / Editor) and specify your value in the line GRAPHIC_YELLOW_PERCENT. When you specify the value, do not add the % sign.

11.3 Measurement Output Options and Destinations

Measurement results are displayed in the Measurement window after the minimum number of points has been specified. You can output the measurement results to up to four destinations, which are shown on the measurement output icons. However, only two destinations are displayed at the same time.

The results can be output in the following ways after you select the destinations and the measurements to be output:

- Displayed in the Routine listing on the screen when the step is created and when you run the routine. This includes the actual measurements and specified nominal and tolerance values, but does not include the deviations.
- Sent to a data stream file when you run the routine

- Print data (when you click the Print Data tab in the Image window) can be sent to up to three destinations.
  - You can indicate whether data is to be printed during the creation of a routine or in step edit mode in the Print Data Configuration.
  - The data printed during run mode is based on the selected Printer Override Options
- Sent to a statistics file when the step is created and when you run the routine (depending on the Statistics Override Options)
- Sent to a file for SmartReport Print Output when you run the routine
- Sent to a file for SmartReport Data Export when you run the routine

If you want to override the output defaults when the routine is run, select the desired run override options before you run the routine.

The software "carries forward" the selected destinations to the next measurement step.

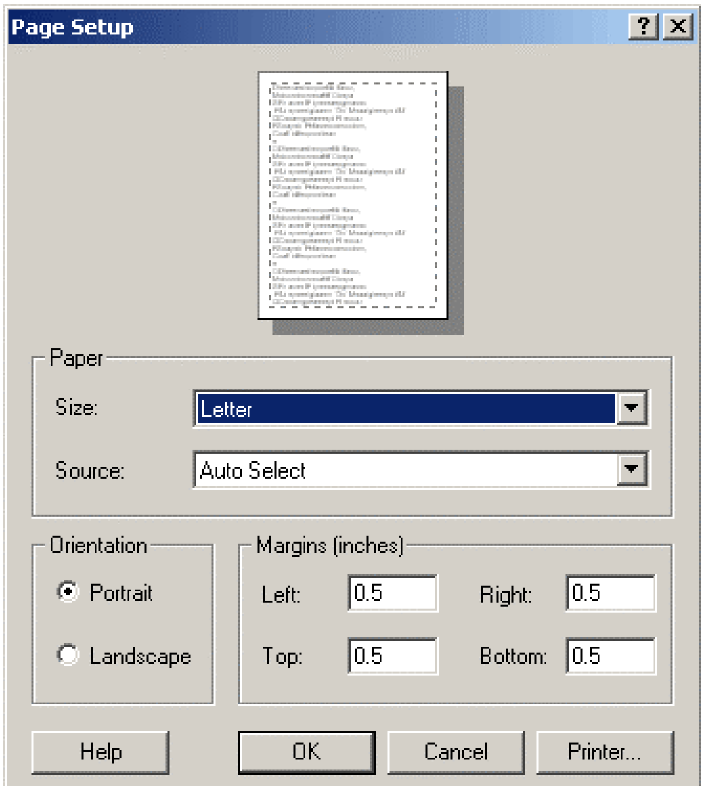
A setting of 1 (default) for the PARTIAL_OUTPUT parameter in the Default Preferences section of the configuration settings allows partial output to a file or printer if a routine run is not completed.

11.4 Page Setup

You can use the standard Windows Page Setup dialog box (accessed from the File menu) to:

- Change the paper size and source, and the printing orientation.
- Specify the left, right, top, and bottom page margins. The default is 0.5 inches on all sides.

To display the Page Setup dialog box, select **File** / **Page Setup** from the main menu.



To select the printer, click the **Printer** button. This displays the same drop-down list as in the Print Setup dialog box.

11.5 Print Setup

The default setting is to output measurement results to the printer that is attached to the system when you run the routine. eBx sends all print data to the printer through the Windows print manager. You must have a properly configured printer driver to print to the printer.

If you want to change the default settings, for example, to send the results to a different printer or to a print file during the creation of a routine, you must select it in the Print Setup dialog box.

To select the printer and the port, select **File** / **Print Setup** from the main menu. The software displays and uses the standard Windows Print Setup dialog box.

The default printer is indicated at the top. In most cases you can simply use the default printer and click OK; you do not need to make any changes. However, if you want, you can select a specific printer and port. You can also change the paper size and source, and the printing orientation.

11.6 Setup to Print Results to a File During Creation of a Routine

If you want to send measurement results to a file during the creation or step edit of the routine, select **Generic** / **Text Only** in the Print Setup before you start creating the routine.

The data is output to the file specified in "TEMP_PRINT_FILE" in the INPUT.INI file. The default file name is TEMPFILE.TXT and it is located in the QVSI\BASICX folder. To specify a different file name, select System / Configuration / Editor, locate the Printer section, and specify the name.

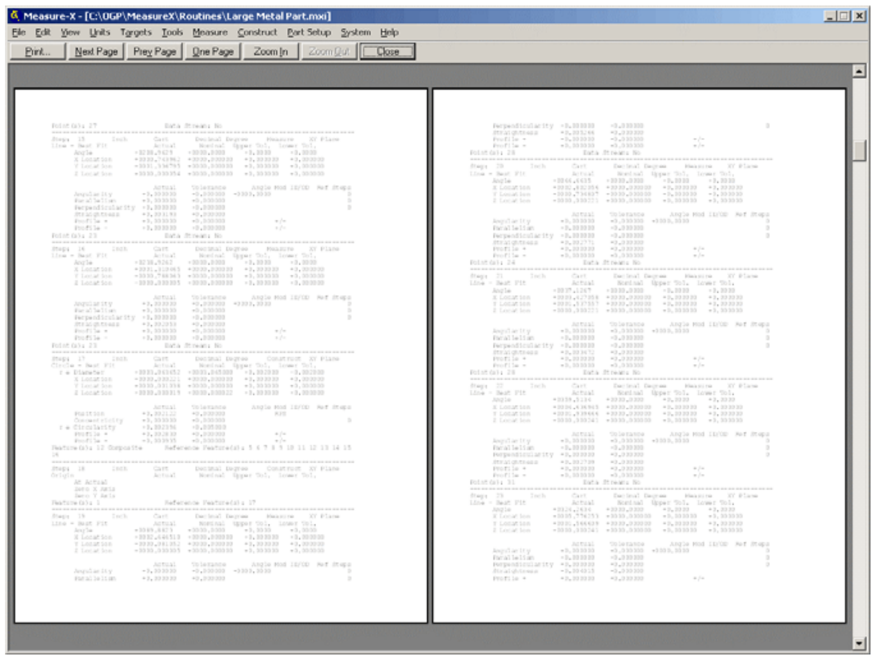
If Generic / Text Only does not appear as an option, you must first install the Generic / Text Only driver and select File as the destination.

> With Generic / Text Only connected to File, you can specify a different file name each time you run a routine. This enables output to go to multiple files. For example, this may be used to output non-linear calibration results to different files.

11.7 Print Preview

Select **File** / **Print Preview** to view what is displayed in the image area (image, model, routine listing) before it is printed. This function displays each page as it will look when you print it.

You can view one or two pages at a time and you can magnify or reduce the size of the page on the screen.



11.8 Print Data Output

The dimensions that were output to a printer in a measurement step appear in the Print Data window. This includes the actual measurements, specified nominal and tolerance values, and deviations.

> If the system is configured to display the Classic User Interface, click the **Print Data** tab to display the Print Data window.

This window acts as a temporary location for the print data output. You can output the data to the printer or to a file using the Print Data toolbar.

The dimensions are sent to this display area during the creation, editing, and step edit modes when Printer is selected in the Printer Override Options and Print During Create is On in Print Data Configuration.

When you load a routine the Print Data window appears grayed out and the Print Data toolbar is disabled. The window text will become darker and the toolbar is enabled once you run the routine.

When you run the routine:

- The software clears this area at the start of the run. It does not clear the area if the routine is run again with the Part Repeat or Fixture Repeat run options.
- The software sends the print data to this area.
- The software displays this window instead of the Image and Model windows if Print Data Tab is selected.
- At the end of the run, the software sends the data to the printer if Printer was selected in the Printer Override Options. Otherwise, if you want to print the data, you need to click the Printer icon in the Print Data toolbar.

11.8.1 Print Data Toolbar

When the Print Data Output is displayed, you can control the current data in the following ways:

- Click the **Save** icon to save the data to a text file.
- Click the **Printer** icon to immediately send the data to the printer.
- Click the **Font** icon to display a dialog box where you can select the desired font, font style, color, etc.

Clicking the Save or Printer icon does not clear the data.

These controls are not available until you run the loaded routine; they are not available while the routine is running.

The software sends all print data to the printer through the Windows print manager. You can also use the Windows print manager in the Windows taskbar to cancel or move the jobs that are in the print queue.

When the software is configured to display the Single Monitor User Interface on a widescreen monitor, you can temporarily increase the size of the Print Data window by double-clicking the title bar. To decrease the size of the window, double-click the title bar again.

11.8.2 How to Output Measurement Results to a Print Destination

To output measurement results in a measurement step to a specified destination:

1. Click next to the measurement output icon and click  from the drop-down list. This clears all the check boxes for the current destination but it does not clear the check boxes for any previously selected destinations.
2. Select the check box next to the desired measurement(s) under the Printer icon to select the measurement(s) for output. A check mark appears in the box.
3. Click the Printer icon only if you want to output the measurements to the print data output or to a print file during the routine creation mode (before it is run).
   - If you selected Printer or Print Data tab in the Printer Override Options, the software immediately places the selected measurements in the print data output. It is assumed that "Print during Create" is checked in the Print Data Configuration.
   - If you selected File in the Printer Override Options, the software immediately sends the measurements to the print file.
4. Click the **OK** or **Again** button at the bottom of the Measurement window to close the window.

The selected measurement results are output to the selected destination(s) when the routine is run.

> You can specify whether selected measurements are printed during create or step edit mode in the Print Data Configuration.

11.8.3 Understanding Measurement Results Sent to a Print Destination

All measurement printouts include the name of the routine, run number and data and time at the beginning of the report along with the page numbers at the top of every page. If you entered a report heading it also appears at the beginning of the report.

For each step, the report lists the step number, any comment (report text) that you entered, and the units, nominal values, tolerances, and deviations of each feature in the step.

The Exceeded column uses + and - signs to indicate where the actual value falls in relation to the nominal and upper and lower tolerances.

Sign	Meaning
blank	A deviation is not calculated -- no tolerances
+	Deviation is between the nominal (0) and 25% of the upper tolerance
++	Deviation is between the 26% and 50% of the upper tolerance
+++	Deviation is between the 51% and 75% of the upper tolerance
++++	Deviation is between the 76% and 100% of the upper tolerance
-	Deviation is between the nominal (0) and 25% of the lower tolerance
--	Deviation is between the 26% and 50% of the lower tolerance
---	Deviation is between the 51% and 75% of the lower tolerance
----	Deviation is between the 76% and 100% of the lower tolerance

A number in the Exceeded column indicates that the actual value has exceeded the tolerance. The number represents the deviation from the nominal, minus the associated tolerance. The number is calculated as follows:

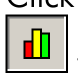
- Actual Value - Nominal = Deviation
- Deviation - Associated Tolerance = Number in the Exceeded Column

> In the output of a linear or non-linear grid run, an asterisk (*) next to the deviation value indicates that the deviation is equal to or exceeds the specified tolerance.

11.9 How to Output Measurement Results to a Statistics File

You can output measurement results to a statistics file in a format that is suitable for further statistical analysis. The format is based on the statistics output configuration and statistics templates.

To output measurement results in a measurement step to a statistics file when you run the routine, follow the steps below.

1. Click next to the measurement output icon and click  from the drop-down list. This clears all the check boxes for the current destination but it does not clear the check boxes for any previously selected destinations.
2. Select the check box next to the desired measurement(s) under the Statistics icon to select the measurement(s) for output. A check mark appears in the check box.
3. Click the **OK** or **Again** button at the bottom of the Measurement window to close the window.

Each time you run the routine, the software displays a prompt for a file name, drive and folder if the "Keep same file name each time" check box is not checked and a file is not specified in the Statistics Overrides options.

4. Select the destination drive/folder and type the desired file name. The default extension is STA. Then close the window.

The selected measurement results are output to the specified statistics file either during the run or at the end of the run, depending on the setting of the Partial Output configuration parameter. If you send data to the same statistics file, the system:

- **Appends** the data in the statistics file; it does not overwrite the file.
- **Overwrites** the data in the temporary file.

> If you want to send the selected data to the Statistics file while you are creating the routine, click the Statistics icon itself while the step is still open. When you close the first step that has output to a statistics file, the software displays a prompt for a file name, drive and folder. After you specify a file name, the software outputs the data to that file and then appends the data in the statistics file each time you click the Statistics icon in a step.

11.10 Output Measurement Results to a SmartReport Print File

You can output measurement results in a measurement step to a SmartReport file, REPORT.STA. The default location of the file is in the C:\QVSI\SmartReport folder.

The output data is used as input by SmartReport to design and generate standard or custom inspection reports. You can also print the contents of this file in a format specified in the SmartReport Output Configuration.

If SmartReport is open while you're running the routine, it will read the output measurement results immediately.

To output the measurement results, follow the steps below.

1. Click next to the measurement output icon and click  from the drop-down list. This clears all the check boxes for the current destination but it does not clear the checked boxes for any previously selected destinations.
2. Select the check box next to the desired measurement(s) under the SmartReport icon to select the measurement(s) for output. A check mark appears in each check box.
3. Click the SmartReport icon only if you want to specify custom labels for the selected dimensions (described on the next page). If you do not specify any custom labels, the software uses autolabeling that includes the step number, feature tag and a label tag. For example, the autolabel for a hole diameter in Step 1 would be 1: Circle: Dia.
4. Click the **OK** or **Again** button at the bottom of the Measurement window to close the window.

The selected measurement results are output to the SmartReport file or directly to SmartReport when you run the routine. The software appends the data in this file each time you run the routine.

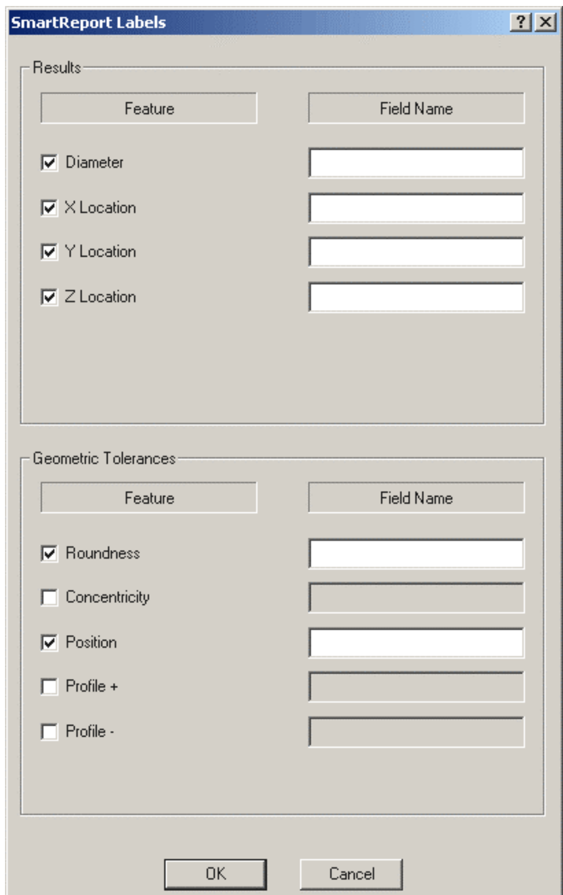
> If you want to change the location of the REPORT.STA file, select System / Configuration / Editor from the main menu, go to the Paths section and the Smart_Report_Path setting, and specify the desired location.

11.11 Custom SmartReport Print Labels

Click the SmartReport icon to display a dialog box where you can type custom labels for the selected dimensions.

The dimensions that you selected in the Measurement window have check marks next to them in the SmartReport Labels dialog box and the label field is activated for each selected dimension.

The software uses autolabeling if you do not type anything in the label field. The software uses the custom label only if you type the label.



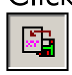
11.12 Output Measurement Results to a SmartReport Data Export File

You can output measurement results in a measurement step:

- To a SmartReport data export file (EXPORT.STA in the C:\QVSI\SmartReport folder).
- Directly to a database program, e.g., Microsoft Excel, if SmartReport and the database program are open

The output data is used as input by SmartReport to export inspection results to a variety of databases for further evaluation. You can specify the format of this file in the SmartReport Output Configuration.

To output the measurement results, follow the steps below.

1. Click next to the measurement output icon and click  from the drop-down list. This clears all the check boxes for the current destination but it does not clear the checked boxes for any previously selected destinations.
2. Select the check box next to the desired measurement(s) under the Data Export icon to select the measurement(s) for output. A check mark appears in each check box.
3. Click the **Data Export** icon only if you want to specify custom labels for the selected dimensions (described on the next page). If you do not specify any custom labels, the software will automatically insert appropriate field names when you export the data.
4. Click the **OK** or **Again** button at the bottom of the Measurement window to close the window.

The selected measurement results are output to the SmartReport file or directly to the database program when you run the routine. The software appends the data in this file each time you run the routine.

- If you have a pre-existing database, browse for that database in the Data Export configuration dialog box. If you want SmartReport to create the database automatically, be sure to specify a new database file name.

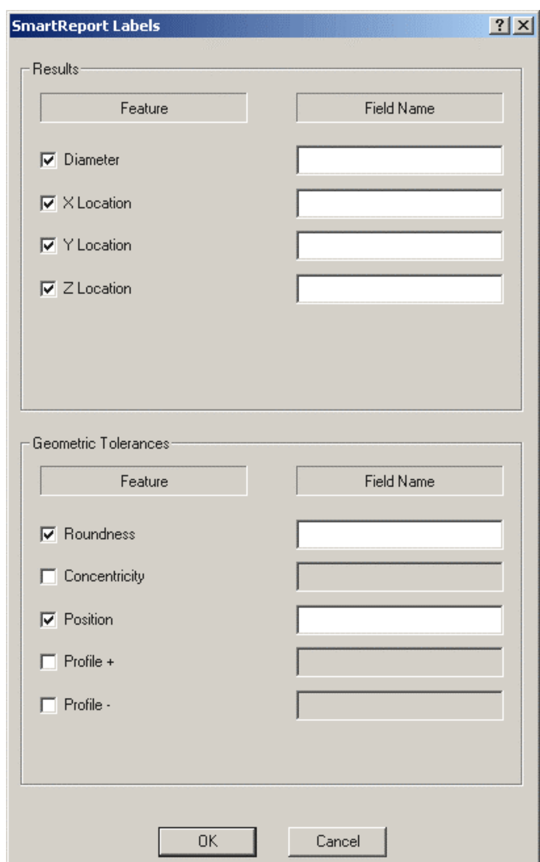
- If you want to change the location of the EXPORT.STA file, select System / Configuration / Editor, display the Paths section and the Smart_Report_Path setting, and specify the desired location.

11.13 Custom SmartReport Data Export Labels

Click the Data Export icon to display a dialog box where you can type custom labels for the selected dimensions.

The dimensions that you selected in the Measurement window have check marks next to them in the Data Export Labels dialog box and the label field is activated for each checked dimension.

- For an export to a new Excel or Access database, leave the fields blank. SmartReport will automatically insert appropriate field names when you export the data.
- For an export to any other database or for re-export to an existing database, you must enter the pre-existing database names into the appropriate fields. Otherwise, SmartReport will not be able to correlate the two sets of field names and the data transfer will not be completed correctly. In this case, enter only alphanumeric characters and spaces into the fields. In particular, do not enter commas (,) or periods (.) into the fields.



11.14 Output Measurement Results to a Data Stream File

This function enables you to output the "raw" measurements (that is, XYZ edge coordinates instead of calculated geometry values) of selected features to a data stream file. The data that is sent is based on the data stream templates.

To send the measurements, when the routine is run:

1. Select the Data Stream check box in the Measurement Results window of the measured feature.
2. Click **OK** or **Again** in the measurement step.

The software prompts you for a data stream file name (the default extension is DAT), drive, and folder:

- Only the first time that you run the routine, if "Keep Same Filename Each Time" is checked in the Data Stream Override Options. The software appends the data in this file each time you run the routine.
- Each time you run the routine, if "Keep Same Filename Each Time" is not checked in the Data Stream Override Options. The software appends the data if you select the same file each time you run the routine.

If you want, you can change the file name in the Data Stream Override Options when you run the routine.

🗎 No data is sent if the feature is missed during the run of the routine.

🗎 The name of the temporary file for data stream output is DSTREAM.TMP in the QVSI\BASICX folder. This file is created when the CREATE_TEMP_FILE parameter in the Data Stream section is set to 1. This file contains only the default data, even if you select the Nothing or All Features in the Data Stream Override Options when you run the routine.

This page was left blank intentionally.

12. Editing a Routine

12.1 Overview

Use the Edit icons in the toolbox or the Edit menu to make the following kinds of changes to the current routine:

- Delete last step
- Insert a step
- Delete a step
- Change a step
- Copy one or more steps a specified number of times with an offset
- Run the routine in Step Edit mode (selected from the Edit menu only)

When you finish editing a routine in any way, it is strongly recommended that you run the routine to verify that the changes were made correctly.

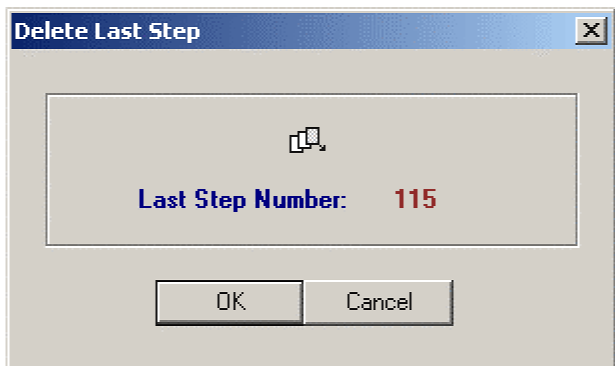
The last step of the routine is displayed in all the Edit fields that appear in the Measurement window.

12.2 Delete the Last Step

This Edit function deletes (undoes) the last step that was completed in the routine.

To delete the last step, click or select **Edit** / **Delete Last Step** from the main menu.

The system displays a confirmation prompt and verifies the step number. Click **OK** if you are sure that you want to delete the last step.



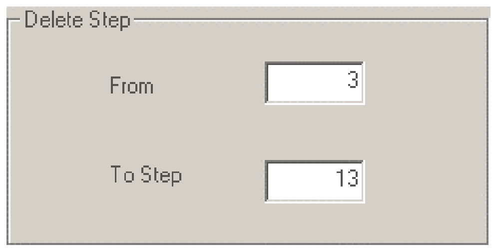
You can use this function more than once. To delete the last three steps in the routine, use it three times in a row.

12.3 Delete Step(s)

You can delete one or more steps anywhere in the routine. The software decrements the step numbers that follow.

To delete a step:

1. Click  or select **Edit** / **Delete** from the main menu. A prompt for the step number(s) appears in the Measurement window and the first field becomes active.



2. Specify the beginning step number in the first field in one of the following ways:
   - Type the desired number.
   - Click the desired feature in the Model. The step number for that feature appears in the field. If you select the wrong feature, you can continue to select features until the desired step number appears. If multiple features appear in the same location, the software displays a menu where you can select the desired step.
3. If you want to delete only one step, leave the second field empty and go to Step 5 below.
4. If you want to delete two or more steps, repeat Steps 2 and 3 for the ending step number in the second field.
5. Click **OK**. The software displays a confirmation message showing the steps to be deleted.
6. Click **OK** to delete the feature(s) or click Cancel to quit the deletion.

- Be careful about deleting a feature that was used as part of a constructed measurement. If the deleted measurement is a component of a constructed feature, and the constructed feature becomes invalid (for example, it no longer has the minimum required number of steps to calculate a geometry), the software displays a message indicating the other constructed steps that may be deleted or modified.

- Be very careful about deleting a step that contains a datum. This could make other measurements become invalid.

12.4 Change a Step

You can change a step in many different ways. For example, you can:

- Add, change or remove points or features that were used to measure or construct a feature
- Remeasure or change the feature using different tools, targets, units, magnification, or light settings
- Add, change or remove nominal and tolerance values for a feature by clicking the Tolerances tab
- Change the output choices and destinations in the Measurement window. For example:
  - For Print, you may change or add to the nominal and actual values sent to the printer by placing a check mark in front of at least one value.
  - For Statistics, you may change or add to the actual values sent to the designated port or file by placing a check mark in front of at least one value. If you specify a file, a window displays allowing you to create a new file or access an existing file.

12.5 Enhanced Editor

The Enhanced Editor allows you to edit your program steps directly in the Listing window without having to use the Edit Step function.

To activate the Enhanced Editor, click at the top of the Listing window. To return to the standard Listing window, click the icon again.

> If the Enhanced Editor does not activate when you click the icon, you may need to change a system parameter. Under System / Configuration / Editor, in the EMX General section, look at the EMX_ListingEditor setting. This must be set to 1 in order for the Enhanced Editor to be available.

The Enhanced Editor allows you edit the Reference Plane, Best Fit/Min/Max, Data Destination, Nominals, Tolerances, Modifiers, and Lights directly for every step in your program. Any changes made in the Enhanced Editor are immediately reflected in the Model window and the Print Data window. A single click in the Enhanced Editor will highlight your data and give you the ability to make changes. Double-clicking the step in the Enhanced Editor opens the standard Step Editor. Steps may also be deleted directly from the Enhanced Editor condensed, standard, or expanded modes. Simply highlight the step in the Editor and press Delete on the keyboard.

When Constructing a feature, features to be used in the construction can be identified in the Enhanced Editor by pressing and holding the Ctrl key and clicking individual features, or by selecting a beginning step, pressing and holding the Shift key, and clicking the last step that is needed for the construction. This is the same behavior that is used in many other Windows programs when identifying individual or groups of items.

12.6 How to Change a Step

1. Start by doing one of the following:
   - Click the feature in the Model window. Go to the end of Step 3.
   - Click  or select **Edit** / **Change** from the main menu.

     A prompt for the step number appears and the model is displayed.

   - Click the step you want to change in the Enhanced Editor.
2. Specify the step number in one of the following ways:
   - Type the desired step number in the field.
   - Click a feature in the Model window. The step number for that feature appears in the field.

3. Click **OK**. If you want to quit the change mode, click Cancel.

When you select the feature (as in Step 1) or click OK (as in Step 3), the system does the following depending on how the feature was measured:

- If the feature was measured optically, the system:
  - Displays the measurement results in the Measurement window and the video image in the Image window.
  - Adjusts the magnification and illumination used to measure the feature.
  - May display a prompt to move the stage. The stage motion is based on the setting of the MODEL_MOVE_STAGE parameter in the HARDWARE.INI file (0 = move stage without any prompts, 1 = display a prompt with location, 2 = don't move the stage at all).
  - If a prompt appears and you respond Yes, or the above parameter is 0, the system moves the optics and the stage to the location where the feature was measured.
  - If the measured feature is a plane or sphere, the system does not move the stage.
  - If a touch probe is attached but disabled, the system does not move the stage.
- If the feature was constructed or measured with a touch probe, the system displays the Model window and the measurement results in the Measurement window. It does not move the stage.

4. Change the feature as desired, just as if you were creating the step.
5. Click **OK**.

If you are editing the points of a step, it is recommended that you install the correct sensor that was used to create the feature in the step being edited. This prevents unexpected stage motion, which may cause the optics or touch probe to crash into the part. For example, if the feature was measured with a touch probe, you must make sure the touch probe is enabled.

If you change a step that was measured with a rotary indexer, make sure that the part is rotated to the same position where the feature was measured during the creation of the routine.

12.7 Insert a Step

You can insert a step anywhere in the routine. The software increments the step numbers that follow.

Inserting a step consists of two tasks:

- Measure or construct the feature to be inserted
- Choose the position number where you would like your step to be inserted, and click the Insert button

To measure or construct the feature:

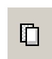
1. Select the desired tool or target. You can also specify different units or save the routine.
2. Select a geometry using the Measure or Construct menus or toolbox icons.
3. Measure or construct the feature.
4. Look for the **Insert** button and the following **at position** box. Fill in the position number you would like your new step to take, then press **Insert**.

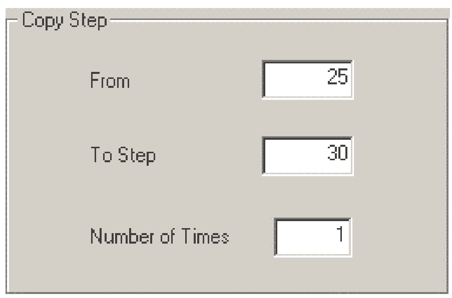
   The software adds the new step at the specified position and all the steps that follow the inserted step are renumbered. For example, if you insert a new step at position 4, the new step becomes Step 4, the old Step 4 becomes Step 5, the old Step 5 becomes Step 6, etc.

12.8 Copy Step(s)

If a part has many features with the same characteristics (geometry, size, tolerance, output, etc.) you can use Copy to quickly add the features to your routine. You can copy one or more steps at a time.

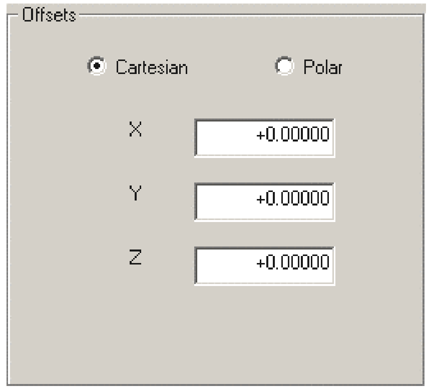
To enter the step numbers and copy the feature(s) one or more times:

1. Click  or select **Edit** / **Copy** from the main menu. Copy fields appear in the Measurement window and the model is displayed.

2. Specify the Start and Stop step numbers in one of the following ways:
   - Type the desired step numbers in the applicable fields.
   - Click a feature in the model. The step number of that feature appears in the Start field. Put the cursor in the Stop field and repeat this for the Stop step number.
3. Position the cursor in the Number of Times field and type the number of times the feature is to be copied. They must be copied at least once.

4. Select the desired radio button to select Cartesian (default) or Polar coordinates (for example, if you will specify an angular offset, select Polar).



5. Enter an offset (XYZ or RAZ coordinates). If you do not specify an offset, zeros are assumed. Offsets are added to the nominal coordinates of each copied step. All offsets are relative to the current datum setup.
6. Click **OK**. The software displays a confirmation message showing the steps to be copied.
7. Click **Yes** to copy the feature(s). The software displays the copied feature(s) in the model.

The copied features are not measured at the completion of the copy operation. This will happen when you run the routine. The copied features will produce output when they are encountered in File / Run or Edit / Step.

If you copy step(s) that contain rotary indexer move(s), it is recommended that you run the routine immediately after copying the steps so that the rotary indexer is at the appropriate position for the steps that are measured afterward.

When copying steps that contain datums and the steps are copied in multiple rows and columns (e.g., an integrated circuit), it is recommended that you copy rows or columns of features one row or column at a time. If you copy more than one row or column at a time, small changes may occur in the nominal location values of the copied steps.

🗎 If the system displays an error message "Copy Created Invalid Features", check the XYZ offsets and their values, Cartesian/Polar coordinates, and the number of features that were copied. For example, if you copied data extraction steps, you may also need to copy the reference features of the data extraction steps.

⚠ Do not use Copy to measure several samples of the same part at the same time. Instead, use the Part Repeat option located in File / Run to measure multiple parts.

⚠ You may copy features from a prior datum coordinate system but they will be added in the current system.

12.9 Step Edit

This option allows you to do interactive editing while running the routine in step edit mode. In this mode, the system steps through the routine in the way it was created. The routine pauses at every step or at each point in a step so that you can view the measurement results and make the necessary changes. You can:

- Change the current step, including individual points in a touch probe step
- Insert a feature after the current step
- Delete the current step

To prevent unexpected stage motion or contact between the sensor and the part, check the following **before you begin** the step edit of the routine:

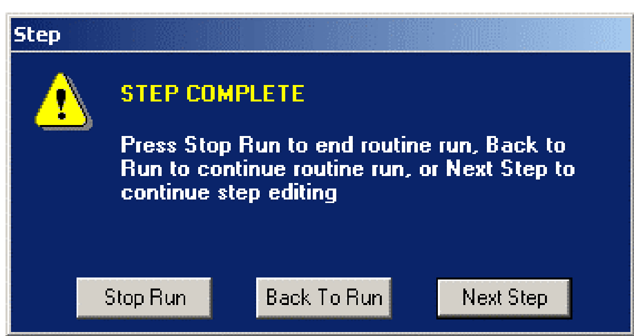
- Make sure that the path is clear to the first feature where the step edit is to begin or to the port (if equipped). This is necessary if that feature was measured with a touch probe and the probe is not installed, or if the feature was measured optically and a touch probe is currently installed.
- Be sure you have the same part setup as when you created the routine.
- If a rotary indexer was used to measure a feature, make sure that the part is rotated to the same position where the feature was measured during the creation of the routine.

Do not use Step Edit when using the Search function. Step Edit allows the stage to move, producing an offset change. To edit steps while using the Search function, select Edit / Change from the main menu, or click the **Change a step** icon in the toolbox.

12.10 How to Run a Routine in Step Edit Mode

To interactively edit the routine while running it in step edit mode:

1. Select **Edit** / **Step Edit** from the main menu. The software displays a Step Edit Run window with the starting and ending step numbers where you can specify the steps you want to edit. The default is all the steps in the routine.
2. To specify the steps, place the cursor in each field and type the step number or select the desired feature from the model window.
   - If the feature in the starting step was measured optically, the system displays the measurement results in the Measurement window and the actual image, along with the tool or target that was used, in the Image window. The stage moves to the first point of the selected feature or to the center of the feature, depending on which tool or target was used.
   - If the feature was measured with a touch probe, the system displays the scaled feature in the Model window and the measurement results in the Measurement window. It moves the stage to the first safe point.
   - If the feature was constructed, the system displays the scaled feature in the Model window and the measurement results in the Measurement window. It does not move the stage.
3. Make any desired change(s) in any of the ways described under Change Step.
4. Click **OK**. The software displays a Step Complete dialog box with Stop Run and Next Step buttons.



5. Click **Next Step** to continue with the step edit or on Stop Run to stop the run.

A Back to Run button is available in this mode. When a measurement fails while running a routine, a confirmation message allows you to go to the Step Edit mode to make the corrections. When you are done, select Back to Run to resume running the routine.

12.11 Insert a Step During Step Edit

The Insert function in Step Edit works the same as Edit / Insert with only one exception: the feature is inserted **after** the current step.

To insert a step during step edit, when the current step is displayed:

1. Click the **Insert** button. The software displays a confirmation prompt.
2. Click **OK**. The software saves the feature that was just measured and displays the same type of feature again.
3. Measure or construct the same type of feature or immediately select another type of feature from the Measure or Construct menus.

After you insert a step, all the step numbers that follow are incremented.

12.12 Delete the Current Step in Step Edit

When the routine displays the current step during Step Edit, you may remove it by clicking **Delete** at the bottom of the current step. A confirmation prompt appears so you can verify that you really want to delete the step.

When you delete the current step, all associated measurement data is also deleted and all the step numbers that follow are decremented.

If you want to delete more that one step, use the Delete function.

12.13 Global Editor

Global Editor is an editing tool that enables you to change selected parameters in a specific step and then apply the changes to multiple steps at one time. This tool is available only when you are editing/changing a step.

You can use Global Editor to make changes to weak edge settings, lights, output, nominals, tolerances, units, and display settings. The software follows the basic rule that *items that can be changed will be changed*.

WARNING: **Save the routine** before making any Global Editor changes. Once you apply the changes, they cannot be undone.

To use Global Editor, follow the steps below.

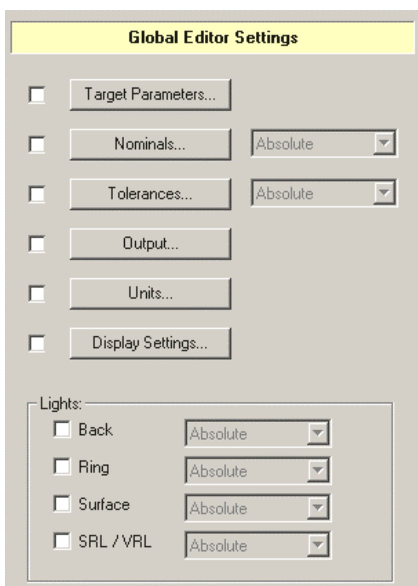
1. Edit/change a step. The Global Edit button appears in the same place as the Again button when you created the step.
2. Click the **Global Edit** button.
   - The software records all changes made to the current step and stops recording changes after you click this button. In order for changes to be saved for lights and target parameters, you must click Remeasure or delete and re-measure the points (or last point) before clicking on this button.
   - The software displays the Global Editor Settings window.
3. Make the desired changes in the Global Editor Settings window.
4. Click the **Next** button. The software displays the Apply to Steps window.
5. Specify the steps to which the changes will be applied (selected steps or all steps). You can also restrict the steps to which the setting changes are applied.
6. Click the **Preview Changes** button to preview the changes that will be made before you apply them.
7. Click the **Apply** button to apply the selected changes to the specified steps. Please note that *once you click this button, you cannot undo the changes*.

8. Click the **OK** button to close the "Changes have been applied" message.
9. Click the **Close** button to close the window and measurement step.

> 🗎 It is recommended that you check the routine listing when deciding on the steps to which the changes will be applied.
>
> 🗎 Routine run overrides take precedence over Global Editor changes.
>
> 🗎 The Global Edit button is not available for the following features: datums, Digital I/O, Branch, Rotary (feature).

12.13.1 Global Editor Settings

The software displays the Global Editor Settings window when you click the Global Edit button in a step that is being edited. The software places a check mark next to any setting that was changed in the step being edited. For example, if you changed target settings and the backlight intensity in the step, the software selects the Target Parameters check box and the Back Light check box.



You can globally change the following settings in this window: Target settings, Nominal settings, Tolerance settings, Output settings, Unit settings, Display settings, and Light settings.

You can indicate which settings will be changed globally in the following ways:

- If a settings category was changed in the step and you do not want to apply the changes, remove the check mark next to that category. In this case, none of the changes made within that category will be applied to other steps. For example, if you changed print and data stream settings in the step but you do not want to apply those changes to any other steps, remove the check mark from the Output check box.
- If a settings category was not changed in the step but you want to apply the changes to other steps, place a check mark next to that category. In this case, all the settings for that category will be applied to other steps. For example, if the current print and data stream settings are acceptable and you want to apply them to other steps, select the Output check box.
- Click a settings category button to change specific settings (listed above) to further control the settings to be applied. For example, to indicate that the change to the target color should not be applied, click the **Target Parameters** button and clear the **Color** check box in the displayed window.
- For Nominal, Tolerance, and Light settings, you can select how the changes will be applied in the drop-down box next to the applicable category.
- Each check box next to a settings button can have one of the following states:
  - If it is checked but partially grayed out, this indicates that only *some* of the settings are checked in the applicable window.
  - If it is checked but not grayed out, this indicates that *all* of the settings are checked in the applicable window.
  - If it is unchecked, this indicates that *none* of the settings are checked in the applicable window.

Click one of the following control buttons after selecting the desired settings:

- Click the **Reset** button to undo the accepted changes and return the settings in this window and in **all** the specific settings windows back to the default state, i.e., the state when this window was first displayed.
- Click the **Back** button to close this window and return back to the edit step's Measure window. This does **not** apply any of the selected changes to any steps in the routine.
- Click the **Next** button to display the Apply To Steps window. This also accepts any changes that were made in this window.

If you edit/change a step and a category does not apply, that category is grayed out. For example, since a Point does not have any Display Settings or a constructed feature does not have a Target, you cannot click the specific button. For Nominals and Tolerances, the check box is available if nominal values are entered for the current edit step.

If the step is a multi-point feature measured with single point targets, the last point will determine the global edit change.

12.13.2 Global Editor Target Settings

The software displays the Target Settings window when you click the **Target Parameters** button in the Global Editor Settings window. This enables you to further control the tool/target settings to be applied in other steps.

The software places a check mark next to any setting that was changed in the step being edited. For example, if you changed the tool color and the rough/smooth slider in the step, the software selects the check boxes next to these two settings.

You can change the following weak edge tool settings in this window:

- Tool and target color
- Search area (see *FeatureFinder Search Area*, *Weak Edge Point Search Area*)
    - For Line and Point, the software applies both the size and extent of the search area, up to the maximum allowed.
    - For Circle or Arc, the software applies only the size of the search area, up to the maximum allowed.
- Weak/Strong Edge and Rough/Smooth (see *Weak Edge Settings*)
- Weak edge weighting factors

You can indicate which specific settings will be changed globally in the following ways:

- If a setting was changed in the step and you do not want to apply the change to other steps, remove the check mark next to that setting. For example, if you changed the target color in the step but you do not want to apply the change to any other steps, remove the check mark from the Color check box.
- If a setting was not changed in the step but you want to apply the current setting to other steps, place a check mark next to that setting. For example, if you prefer the current target color and you want to apply it to other steps, check the Color check box.

Click **OK** to accept any changes that were made in this window.

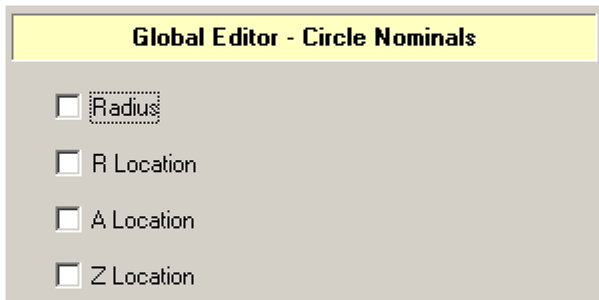
- If you want to undo the accepted changes, you must click the **Reset** button in the Global Editor Settings window; however, this will undo changes that were made in other windows.

Click **Cancel** to close this window without applying any changes that were made in it. This resets the window back to the same settings when it was opened and returns you to the Global Editor Settings window.

> If you edit/change a step and a setting does not apply, it is grayed out. For example, the Rough/Smooth setting is grayed out if a weak edge tool other than FeatureFinder is used in the step.

12.13.3 Global Editor Nominal Settings

The software displays the Nominal Settings window when you click the **Nominals** button in the Global Editor Settings window. This enables you to further control the nominal settings to be applied in other steps.



The software places a check mark next to any setting that was changed in the step being edited. For example, if you changed the nominal values for the radius in the step, the software places a check mark in the check box next to the Radius setting.

This window displays only the nominal items that apply to the current step being edited. For example, you can change the following nominal settings for a circle: Radius/Diameter and RAZ location.

You can indicate which specific settings will be changed globally in the following ways:

- If a setting was changed in the step and you do not want to apply the change to other steps, remove the check mark next to that setting. For example, if you changed the Z value in the step but you do not want to apply the change to any other steps, clear the Z Location check box.
- If a setting was not changed in the step but you want to apply the current setting to other steps, place a check mark next to that setting. For example, if you want to apply the angle to other steps, select the Angle check box.
- The Nominal and Tolerance Setting Changes chart shows what is affected when nominal settings are applied to all features.

Click **OK** to accept any changes that were made in this dialog box.

If you want to undo the accepted changes, you must click the **Reset** button in the Global Editor Settings window; however, this will undo changes that were made in other windows.

Click **Cancel** to close this window without applying any changes that were made in it. This resets the window back to the same settings when it was opened and returns you to the Global Editor Settings window.

For information about how the nominal setting changes will be applied, see *How to Apply Nominal, Tolerance, and Light Settings*.

> The first time you view existing nominal and tolerance values in the Tolerances tab during edit/change, the Global Editor will not note a change (even though all the nominals and tolerances have changed from zero to actuals) until you modify the nominal or tolerance values. However, the software can truncate the values internally when you click the Global Edit button. As a result, a check mark may appear next to the Nominals or Tolerance buttons in the Global Editor Settings window, even if you did not make any changes (the software interprets the truncations as a change). In this case, you need to select/clear the desired settings in this window.

12.13.4 Nominal and Tolerance Setting Changes

The following chart shows what is affected when applying global editor nominal settings and tolerance settings to different feature types. For example, changing the True Position tolerance setting in a Circle step and selecting All features will result in changes to the True Position tolerance setting for points, circles, spheres, cones, cylinders, contours, centroids, widths, intersections, midpoints, and gage ball/diameter.

	X	Y	Z	True Position*
Point	x	x	x	x
Line	x	x	x	

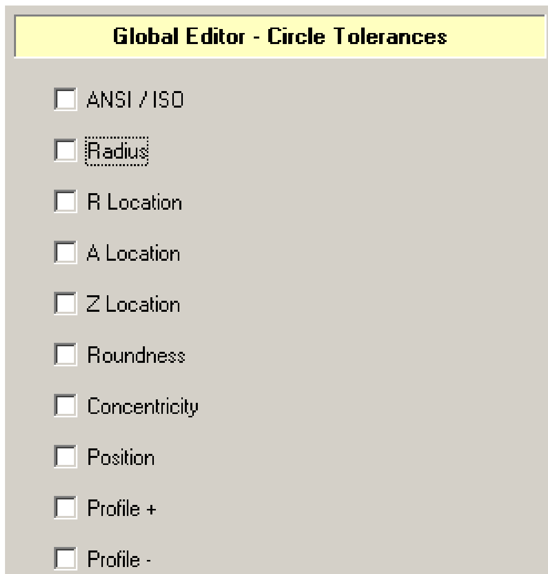
	X	Y	Z	True Position*
Circle	x	x	x	x
Sphere	x	x	x	x
Cone	x	x	x	
Cylinder	x	x	x	
Contour	x	x	x	x
Centroid	x	x	x	x
Width	x	x	x	x
Intersection	x	x	x	x
Midpoint	x	x	x	x
Gage Ball	x	x	x	x
Gage Diameter	x	x	x	x

*True position tolerance settings also include the material (LMC/RFS/MMC), and dimension (ID/OD) settings. These will also be applied along with the true position. Since they cannot be turned off or on individually, they will be applied along with the true position value as an absolute change. These settings will only be applied to Circle, Sphere, Width, and Midpoint features.

All other settings will apply only to like features and are not shown on the chart.

12.13.5 Global Editor Tolerance Settings

The software displays the Tolerance Settings window when you click the **Tolerances** button in the Global Editor Settings window. This enables you to further control the upper and lower tolerance settings (position, form, geometric, profile) to be applied in other steps.



The software places a check mark next to any setting that was changed in the step being edited. For example, if you changed the tolerances for the X and Y locations in the step, the software selects the check boxes next to these two settings.

This window displays only the tolerance items that apply to the current step being edited. For example, you can change the following tolerance settings for a circle: ANSI/ISO, radius, XYZ (or RAZ) location, roundness, concentricity, position, and profile +/-.

You can indicate which specific settings will be changed globally in the following ways:

- If a setting was changed in the step and you do not want to apply the change to other steps, remove the check mark next to that setting. For example, if you changed the upper and lower tolerances for the Z value in the step but you do not want to apply the change to any other steps, remove the check mark from the Z Location check box.
- If a setting was not changed in the step but you want to apply the current setting to other steps, place a check mark next to that setting. For example, if you want to apply the upper and lower tolerances for the angle to other steps, check the Angle check box.
- The Nominal and Tolerance Setting Changes chart shows what is affected when tolerance settings are applied to all features.

Click **OK** to accept any changes that were made in this window.

If you want to undo the accepted changes, you must click the **Reset** button in the Global Editor Settings window; however, this will undo changes that were made in other windows.

Click **Cancel** to close this window without applying any changes that were made in it. This resets the window back to the same settings when it was opened and returns you to the Global Editor Settings window.

For information about how the tolerance setting changes will be applied, see How to Apply Nominal, Tolerance, and Light Settings.

The upper and lower tolerances are *always applied together*, e.g., you cannot apply just an upper X location tolerance change.

The ANSI/ISO, True Position tolerance, material condition, and dimension settings are applied as an absolute change regardless of the drop-down setting associated with the tolerance check box.

> The True Position settings, material condition (RFS, LMC, MMC), and dimension (ID, OD) will always apply with the True Position tolerance value, i.e., there is no option to turn this “off.” A change to the True Position settings or a change to the True Position value will result in both being applied, if selected.

12.13.6 Global Editor Output Settings

The software displays the Output Settings window when you click the **Output** button in the Global Editor Settings window. This enables you to further control the output settings (print, report, export, statistics, data stream) to be applied in other steps.

The software places a check mark next to any setting that was changed in the step being edited. For example, if you selected the X and Y dimensions to be printed and sent to the statistics file, the software places a check mark in the check boxes next to the Printer and Statistics settings.

You can indicate which specific settings will be changed globally in the following ways:

- If a setting was changed in the step and you do not want to apply the change to other steps, remove the check mark next to that setting. For example, if you selected the X and Y dimensions to be printed and sent to the statistics file but you do not want to send these dimensions to the statistics file in other steps, remove the check mark from the Statistics check box.

- If a setting was not changed in the step but you want to apply the current setting to other steps, place a check mark next to that setting. For example, if the X and Y dimensions are checked to be printed and you did not make any changes to that, but you also want to apply the printing of the X and Y dimensions to other steps, check the Printer check box.
- The Output Setting Changes chart shows what is affected when output settings are applied to different feature types.

Click **OK** to accept any changes that were made in this dialog box.

If you want to undo the accepted changes, you must click the **Reset** button in the Global Editor Settings window; however, this will undo changes that were made in other windows.

Click **Cancel** to close this window without applying any changes that were made in it. This resets the window back to the same settings when it was opened and returns you to the Global Editor Settings window.

🗎 The settings for each output type for a given feature are *always applied together*, i.e., whatever print settings are checked or unchecked in the current step are always applied together to other steps. This means that the print settings in the current step will override the print settings in the other steps.

🗎 Data stream is only available for measured features.

12.13.7 Output Setting Changes

The following chart shows what is affected when applying global editor output settings to different feature types. For example, changing the Straightness output setting in a Line step and selecting All features will result in changes to the geometric tolerance output setting for lines, circles, planes, spheres, cones, and cylinders.

	X	Y	Z	Size	Size 2	Size 3	Size 4	Geo-Tol	True Pos
	‘X’	‘Y’	‘Z’	‘S’	‘O’	‘A’	‘E’	‘G’	‘T’
Point	x	x	x						x
Line	x	x	x	x (angle 1/2)	x (angularity)	x (perpen-dicularity)	x (parallelism)	x (straightness)	
Circle	x	x	x	x (diameter/ radius)	x (concentricity)			x (circularity)	x
Plane				x (inclination 1/2)				x (flatness)	
Sphere	x	x	x	x (diameter/ radius)				x (sphericity)	x
Cone	x	x	x	x (diameter/ radius)	x (taper angle)			x (conicity)	

	X	Y	Z	Size	Size 2	Size 3	Size 4	Geo-Tol	True Pos
	'X'	'Y'	'Z'	'S'	'O'	'A'	'E'	'G'	'T'
Cylinder	x	x	x	x (diameter/ radius)				x (cylindricity)	
Contour	x	x	x	x (length)	x (area)				x
Centroid	x	x	x	x (perimeter)	x (area)	x (min radial dist)	x (max radial dist)		x
Distance	x (Dist)	x (Dist)	x (Dist)	x (straight-line)	x (angle between)				
Width	x	x	x	x (angle)	x (width)				x
Intersect	x	x	x	x (angle)					x
Midpoint	x	x	x	x (3D-dist)					x
Gage Ball	x	x	x						x
Gage Diameter	x	x	x						x
Math				x (result)					

	X	Y	Z	Size	Size 2	Size 3	Size 4	Geo-Tol	True Pos
	'X'	'Y'	'Z'	'S'	'O'	'A'	'E'	'G'	'T'
User Input				x (input)					
Min/Max/ Avg	x (Min)	x (Max)	x (Avg)						

12.13.8 Global Editor Units Settings

The software displays the Units Settings window when you click the **Units** button in the Global Editor Settings window. This enables you to further control the units and coordinates settings to be applied in other steps.

The software places a check mark next to any setting that was changed in the step being edited. For example, if you changed inches to millimeters, the software places a check mark in the check box next to the Inch/Millimeter setting.

You can indicate which specific settings will be changed globally in the following ways:

- If a setting was changed in the step and you do not want to apply the change to other steps, remove the check mark next to that setting. For example, if you selected polar coordinates but you do not want to apply the change to any other steps, clear the Cartesian/Polar check box.

- If a setting was not changed in the step but you want to apply the current setting to other steps, select the check box next to that setting. For example, if the units in the current step are in inches and you want to have inches in other steps, select the Inch/Millimeter check box.

Click **OK** to accept any changes that were made in this window.

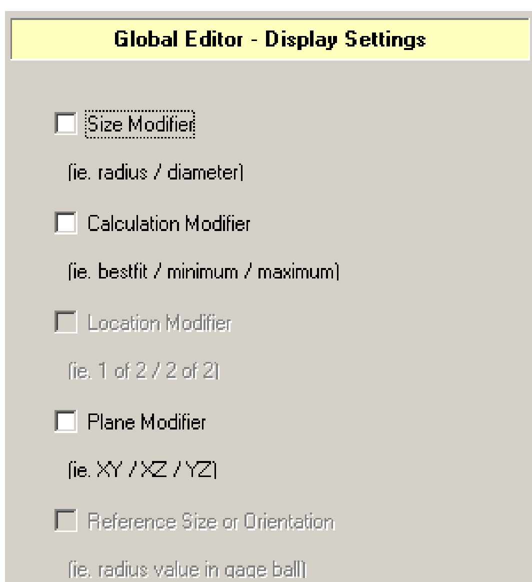
If you want to undo the accepted changes, you must click the **Reset** button in the Global Editor Settings window; however, this will undo changes that were made in other windows.

Click **Cancel** to close this window without applying any changes that were made in it. This resets the window back to the same settings when it was opened and returns you to the Global Editor Settings window.

> Resolution is grayed out if the Global_Precision parameter in the Graphics section is set to 1.

12.13.9 Global Editor Display Settings

The Display Settings window appears when you click the **Display Settings** button in the Global Editor Settings window. This enables you to further control the display settings to be applied to other steps.



The software places a check mark next to any setting that was changed in the step being edited. For example, if you changed the Radius to a Diameter in a Circle step, the software places a check mark in the check box next to the Size Modifier setting.

You can change the following types of display settings: Size Modifier (e.g., radius/diameter, angle1/angle2), Calculation Modifier (e.g., best fit), Location Modifier (e.g., 1 of 2 / 2 of 2), Plane Modifier (i.e., XY, XZ, YZ), and Reference Size or Orientation (e.g., radius value of a gage ball). Any modifier that does not apply to the feature in the current step is grayed out.

You can indicate which specific settings will be changed globally in the following ways:

- If a setting was changed in the step and you do not want to apply the change to other steps, remove the check mark next to that setting. For example, if you changed the Diameter to a Radius in the step but you do not want to apply the change to any other steps, remove the check mark from the Size Modifier check box.
- If a setting was not changed in the step but you want to apply the current setting to other steps, place a check mark next to that setting. For example, if you want to apply the Diameter to other steps, check the Size Modifier check box.
- The Display Setting Changes chart shows what is affected when display settings are applied to different feature types.

Click **OK** to accept any changes that were made in this window.

- If you want to undo the accepted changes, you must click the **Reset** button in the Global Editor Settings window; however, this will undo changes that were made in other windows.

Click **Cancel** to close this window without applying any changes that were made in it. This resets the window back to the same settings when it was opened and returns you to the Global Editor Settings window.

> The following features do not have display settings: Point, Midpoint, Distance, Centroid, Math, and User Input.

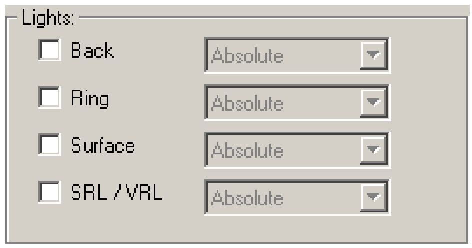
12.13.10 Display Setting Changes

The following chart shows what is affected when applying global editor display settings to different feature types. Any display settings not listed in this table are applied only to like features.

	Size	Calc. Modifier	Feature Plane	1 of 2
Line		x	x	
Circle	x	x	x	
Plane		x		
Sphere	x			
Cone	x			
Cylinder	x			
Contour			x	
Gage Ball	x			x
Gage Diameter	x			x
Width		x		
Intersection				x

12.13.11 Global Editor Light Settings

The software displays the Light Settings in the Global Editor Settings window. This enables you to further control the light settings to be applied to other steps.



The software places a check mark next to any setting that was changed in the step being edited. For example, if you changed the light level of the Back light, the software selects the check box next to the Back Light setting.

You can indicate which specific settings will be changed globally in the following ways:

- If a setting was changed in the step and you do not want to apply the change to other steps, remove the check mark next to that setting. For example, if you changed the Ring light but you do not want to apply the change to any other steps, clear the Ring Light check box.
- If a setting was not changed in the step but you want to apply the current setting to other steps, select the check box next to that setting. For example, if the Backlight is on in the current step and you want to turn on the Back light at the current light level in other steps, select the Back Light check box.

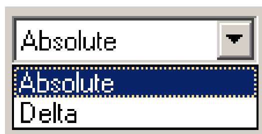
For information about how the light setting changes will be applied, see *How to Apply Nominal, Tolerance, and Light Settings*.

🗎 Light settings are available only if you are editing a measured step; they are grayed out otherwise.

🗎 If you change the Color On or Power On setting in the VectorLight Control window, the software selects the SRL/VRL check box even if the light settings were not changed. These setting changes are *always* applied with the VectorLight changes as absolute changes.

12.13.12 How to Apply Nominal, Tolerance, & Light Settings

For Nominal, Tolerance, and Light settings, you can select how the changes will be applied in the drop-down box next to the applicable category in the Global Editor Settings window.



For Nominal and Tolerance settings, select either:

- **Absolute** applies the changed values as they are in the step. This is the default setting.
- **Delta** calculates the delta change (positive or negative) between the original value and the new value, and applies the delta change to all specified steps.

For Light settings, select how the changes will be applied.

- **Absolute** applies all current light settings as they are in the step. Thus, lights that are currently off in other steps may be turned on and lights that are currently on may be turned off. This is the default setting.
- **Delta** calculates the delta change (positive or negative) between the original value and the new value, and applies it to all specified steps. In this case, lights that are currently off in other steps may be turned on and lights that are currently on may be turned off.
- **Percentage** calculates the percentage of change and applies it to all specified steps. In this case, it will not be possible to turn the lights off. However, certain cases, such as decreasing the light by 100%, may turn the light off. A light is considered off if the intensity is less than 5.

- If the change calculated is such that the light setting is greater than the maximum value, then the light will be set to the maximum value (255). If the change calculated is such that the light setting is less than the minimum value, then the light will be set to the minimum value (0, or off).

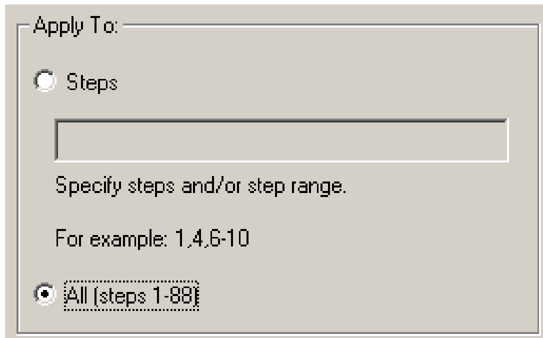
🗎 The SRL/VRL Percentage changes are based on the 8 or 6 ring intensities.

🗎 For Absolute changes, the VectorLight changes are an exact copy of the edit/change step. Thus, sectors and rings will be turned on and off to the settings in the step.

🗎 For Delta and Percentage changes related to VectorLight settings, only the sectors and rings that are currently turned on will be changed by a delta or percentage value. The delta and percentage value will be calculated by a change in ring intensity. Only the ring intensities that have changed will be applied.

12.13.13 Global Editor Step Selection

When you click the **Next** button in the Global Editor Settings window, the software displays the Apply To Steps window. This window allows you to specify the steps to which the global edit settings will be applied.



You must specify the steps to which the setting changes are applied, in one of the following ways:

- All. This applies the settings to *all* the steps. For example, if you increased the light level of the Back light and selected millimeters in the current step, the Back light will be turned on to the current level in all other measured steps and all the dimensions will be specified in millimeters in all other steps.

  The following charts show what is affected when applying output settings, nominal and tolerance settings, and display settings to different feature types.

- Steps (default). This enables you to select specific steps by specifying individual steps and/or ranges of steps in the edit box. Valid entries are numbers, commas, and dashes.
  - Use a comma to separate each step or range of steps. Use a dash to indicate a range.
  - You may specify steps by clicking on a feature in the Model window. This automatically enters that feature's step number in the edit box. Only one feature may be selected at a time and the feature will not be highlighted.

You may also refine or further restrict the steps to which the setting changes are applied.

Click the applicable step selection buttons in this window after specifying and restricting the steps.

12.13.14 Global Editor Step Selection Buttons

Use these buttons after specifying and restricting the steps in the Global Editor Step Selection window.

Click **Preview Changes** to preview the changes that will be made based on the current Global Editor settings.

Click **Apply** to apply all applicable, selected changes to all the specified steps in the routine. The software displays a confirmation message, which you need to acknowledge.

- Click this button only once. Clicking on it multiple times in a row may result in incorrect results if any Delta or Percentage changes are specified.
- This does **not** commit the changes made to the current edit/change step.
- This creates a log file, GEDLog.txt, in the default Temp folder in case you need to view the changes that were made. This file is overwritten the next time you click the Preview or Apply button.

Click **Close** to close this window **after** the changes have been applied with the Apply button. This also closes the current measurement step and commits the changes made in the step (this is the same as clicking on the OK button in the step).

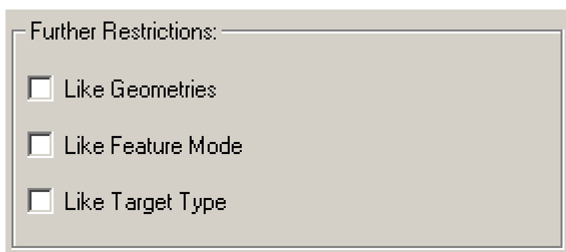
Click **Back** to close this window and return to the Global Editor Settings window. This does **not** apply any of the selected changes to any steps in the routine; also, it does not clear any changes made in the step selection window.

> If any light changes have been determined **and** the feature is a point feature with multiple points measured with single point targets, clicking the **Close** button applies the light changes to all points up to the last point number, but not including the last point number, based on the changes and selections made to the last point. This happens automatically with no indication to the user.

12.13.15 Global Editor Step Restrictions

You may refine or further restrict the steps to which the setting changes are applied in the Global Editor Step Selection window, in the following ways:

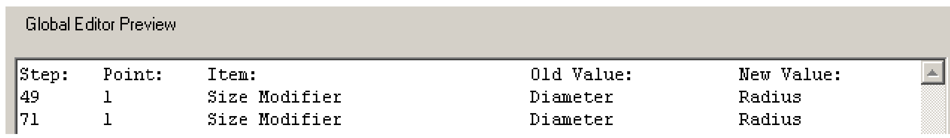
- Select **Like Geometries** to indicate steps that share the same feature type. For example, if you changed the units from inches to millimeters in a Measure Circle step, the changes will be applied only to other specified Circle steps.
- Select **Like Feature Mode** to indicate steps that share the same feature mode – measured or constructed. For example, if you changed the units from inches to millimeters in a Measure Circle step, the changes will be applied only to other specified Measure Circle steps.
- Select **Like Target Type** to indicate steps that share the same tool type. For example, if the feature is measured with the FeatureFinder tool, the changes will be applied only to other features that are measured with FeatureFinder. If you are changing a constructed feature, this option is grayed out.



12.13.16 Global Editor Preview Changes

Click **Preview Changes** in the Global Editor Step Selection window to view all the steps and the changes that will be made based on the current Global Editor settings. This enables you to see the effect of your changes without applying them.

The software displays the current step being edited first, followed by the steps that will be changed. For each change, it lists the step number, point number, the item that will be changed, and the old and new values.



If there is no change in the Global Editor settings, it will **not** be noted, even if the step number and parameter is explicitly specified in this window.

Global Editor applies changes in the following order: Lights, Units, Tool Parameters, Output, Display Settings, Nominals, and Tolerances.

The items that are listed in this window depend on the settings that are selected. For details about the items associated with a selected setting, see the topics listed below.

- Output Settings: Output Setting Changes
- Nominal and Tolerance Settings: Nominal and Tolerance Setting Changes
- Display Settings: Display Setting Changes
- Target Settings
- Units Settings
- Light Settings

This page has been left blank intentionally.

13. Units & Coordinates

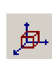
13.1 Overview

All measurements are calculated from one or more coordinate points.

To measure objects, it is important to understand:

- How points are entered, calculated and reported: Cartesian (XYZ) coordinates and polar (RAZ) coordinates
- How the distance between the points is specified: linear units of measurement and angular units of measurement
- The number of significant digits that can be used: Resolution

13.2 Cartesian (XYZ) Coordinates

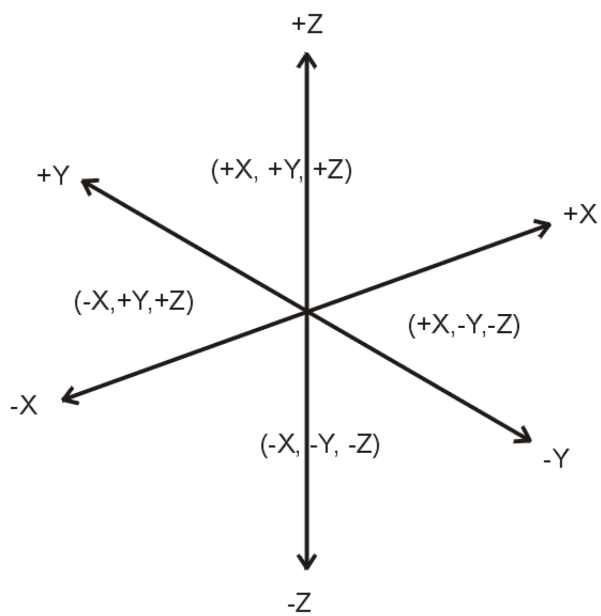
Click  in the toolbox or select **Units** / **Cartesian** from the main menu to display Cartesian coordinates.

A Cartesian (sometimes called rectangular) coordinate system uses three axes to indicate the location of a point. The axes intersect at a single point of reference called the Origin, which is assigned the value (0,0,0). The axes are at right angles to each other.

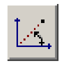
The direction of movement along the axes from the origin is either positive or negative. This direction of movement is called polarity.

- Coordinates along the X axis to the right of the origin are positive; coordinates to the left of the origin are negative.
- Coordinates along the Y axis above the origin are positive; coordinates below the origin are negative.
- Coordinates along the Z axis above the XY plane are positive; coordinates below the XY plane are negative.

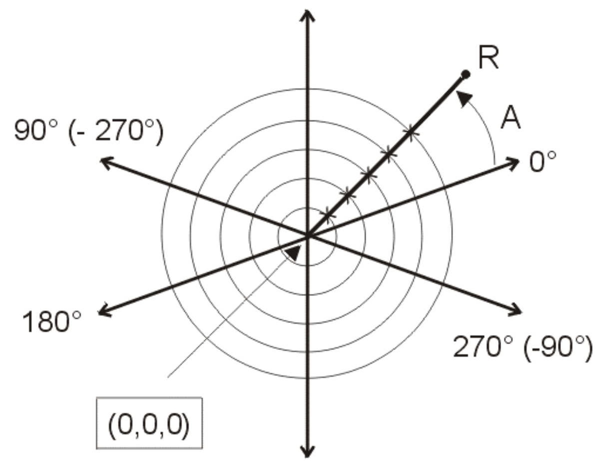
Cartesian coordinates are often referred to as XYZ locations. Each point has three coordinates describing the distance and direction from the origin. The software always displays the coordinate in the same order, X, Y and Z.



13.3 Polar (RAZ) Coordinates

Click  in the toolbox or select **Units** / **Polar** from the main menu to display polar coordinates in the DRO window and all feature measurements.

Polar coordinates use three axes that intersect at a single point called the Origin. The origin is assigned a value of (0,0,0). Polar locations are defined using circles rather than a grid.



Polar coordinates are often referred to as RAZ (radius, angle and Z) locations. Each coordinate has three values:

- The first value (R) is the straight line distance from the origin to the point. This distance is the radius of the circle that passes through the point and whose center lies at the origin. The radius can be displayed in inches or millimeters.
- The second value (A) is the angle between the horizontal axis and the point. Angles are indicated in degrees (a circle has 360°). A counterclockwise direction around the circle constitutes a positive angle. A clockwise direction constitutes a negative angle.
- The third value (Z) is along the Z axis above and below the RA plane.

13.4 Linear Units of Measurement

The measurement software displays coordinate locations in the XYZ (Cartesian) or RAZ (Polar) fields in the DRO window and all feature measurements.

Linear dimensions can be displayed in either English units (inches) or metric units (millimeters).

When the system is turned on, the defaults are the coordinates and units that were used the last time it was on. This is shown with check marks next to them in the Units menu.

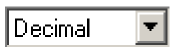
To display metric dimensions, click  in the toolbox or select **Units** / **Millimeter** from the main menu. A check mark appears next to this selection in the menu. The coordinates are converted to millimeters; for example, the inch values are multiplied by 25.4.

To display English dimensions, click  in the toolbox or select **Units** / **Inch** from the main menu. A check mark appears next to this selection in the menu.

13.5 Angular Unit Format

When polar coordinates are displayed in the DRO window and the measurements, the angle is displayed in the A field. It can be displayed either in decimal degrees or degrees, minutes, and seconds.

- Decimal degrees have four significant digits. Example: 45.0000
- Degrees, minutes, seconds are displayed as 45:00:00



You can select the angular unit format from the drop-down list in the toolbox or by selecting Decimal Degrees or Degrees:Minutes:Seconds in the Units menu.

The software reports angular measurements from 0° to 360°. Angles above the X axis range from 000.00° up to +180.00°. Angles below the X axis range from +180.01° to +360.00°.

For angles that are negative or in a clockwise direction (e.g., -45°), the software displays the equivalent positive angle in a counterclockwise direction. Therefore, -45° is displayed as +315°.

13.6 Resolution

Resolution indicates the number of significant digits (to the right of the decimal point) that are used in all displays, reports and inputs. Calculated and entered values are rounded to the number of significant digits selected. However, the calculation itself is not rounded or truncated; all the significant digits are used in all calculations.

The resolution can be up to six significant digits. The default is five significant digits for inch linear dimensions and four for metric or angle dimensions. For example:

- Inches: + 02.01559 (five significant digits)
- Millimeters: + 0051.1960 (four significant digits)

The current resolution is indicated by the number in the drop-down list in the toolbox or by a check mark next to the number in the Units / Resolution menu.

To display another number, select the desired number from the drop-down list in the toolbox or select **Units** / **Resolution** from the main menu, and then the desired number from the menu.

When you run the routine, the software displays all the measurements at the currently selected resolution. If you want to display each measurement at the resolution that is used when you create the step, edit the configuration parameters and change the value of GRAPHICS_GLOBAL_RESOLUTION to 0 in the Graphics section before you begin the create the routine.

14. Configuration Settings

14.1 Overview

This section explains the following software configuration settings:

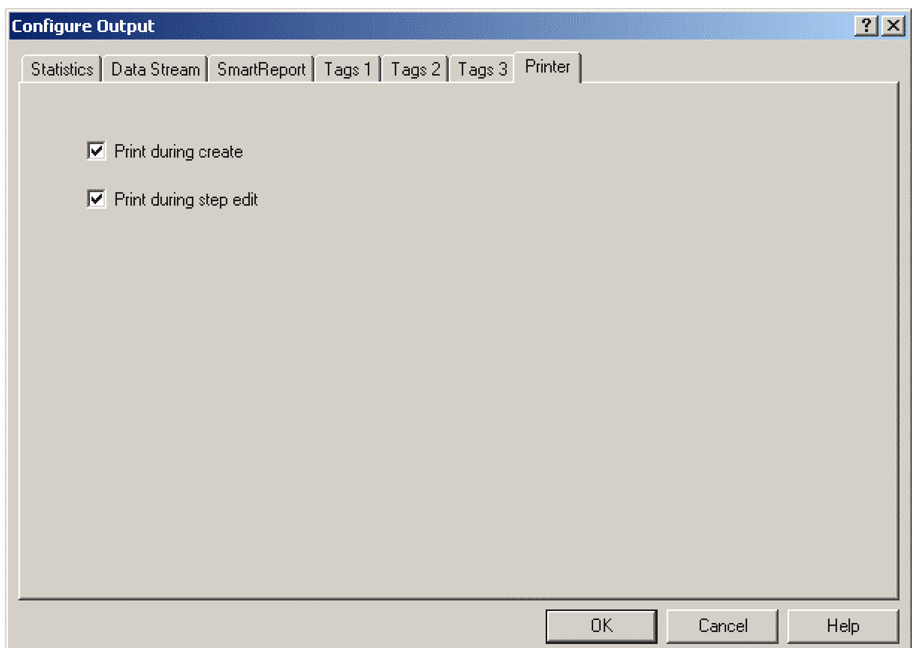
- Output configuration
- Rotary configuration
- Optics configuration
- Sound configuration
- Language configuration
- Menu configuration

14.2 Output Configuration

14.2.1 Print Data Configuration

This option enables you to specify when measurements are sent to the Print Data Output whenever you print data in a measurement step, i.e., when you select dimensions to be printed, click the Print icon, and click the OK button in a measurement step.

To display or change the print data output configuration, click the **Printer** tab in the **System** / **Configuration** / **Output** dialog box. The dialog box contains two check boxes: Print During Create Mode and Print During Step Edit Mode.

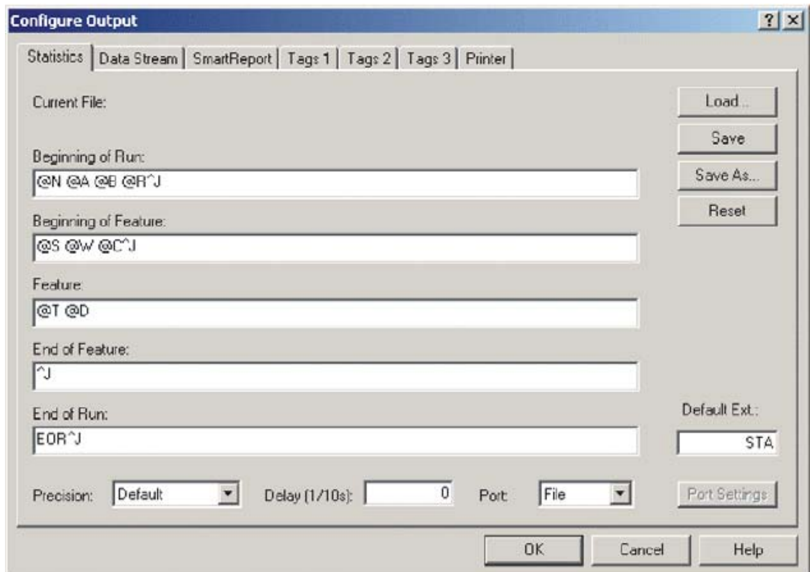


You can indicate whether data is to be printed during the creation of a routine, in step edit mode, in both modes (both boxes are checked), or not at all (neither box is checked). The data printed during run mode is based on the selected printer override options.

See *Page Setup* for information about selecting the printer, paper size and source, paper orientation, and page margins.

14.2.2 Statistics Output Configuration

This option allows you to configure the destination and format of the statistics output. This includes the templates, tags, destination (port), delay, and filename extension for the file containing the statistics output.



To display or change the statistics output configuration, click the **Statistics** tab in the **System** / **Configuration** / **Output** dialog box. The name of the file from which the configuration is loaded is shown at the top. This can be either:

- The default configuration is defined in the Statistics section of the CONFIGURATION.INI file. You can use the default configuration with any routine that you create. The software loads the defaults into temporary memory when you start creating a new routine.
- The configuration that you have defined and saved in a separate file. Each routine can have its own statistics output configuration file. Different routines can also share the same statistics configuration data file.

You can also configure the following:

- Precision of the values to be from 2 to 8 decimal places. The default is 0, which indicates the current system resolution (selected in the Units menu). The selected precision is saved with the part routine.
- The default file name extension for the statistical data, which is TXT. If you want to change it, highlight the extension and type the desired extension.
- Delay, which is used to pause the system after the Stats value has been transmitted to allow additional processing time. The value entered is in tenths of seconds. For example, to add a 30 second delay, enter 300.
- Destination, which can be a file or output device connected to a serial port.

You can save/load the statistics output configuration in the following ways:

- Click the **Save** button to save the configuration data currently in memory and close the dialog box. The data is saved in the file whose name is displayed at the top of the dialog box (or the default file if a name is not displayed).
- Click the **Save As** button to save the configuration data in a new file and close the dialog box. The default file extension is INI and the default folder is CONFIG.
- Click the **Load** button to open a different statistics configuration file and load the data into temporary memory. The system displays an appropriate message if you try to load an invalid file.
- Click the **Reset** button to reload the default data that are in the CONFIGURATION.INI file (in the QVSI\BasicX\Config folder).

When you click the **OK** button, the software will use the data that are currently in memory. If you changed any of the data but you did not save it, the software displays a confirmation prompt asking if you want to continue without saving the data.

If you click the **Cancel** button, the software closes the dialog box and continues to use the statistics data that was in memory before you made any changes or loaded another file.

14.2.3 Statistics Output Options and Destinations

You can output the statistics measurements in the following ways, based on the statistics output configuration and statistics templates:

- Send the data to a printer, data collector or other output device connected to the serial (COM) port. If you select a serial port, the software displays a **Port Settings** button. You can click this button to configure the port. Any changed COM port settings are saved in the Com Ports section of the configuration editor so that they are restored the next time eBx is started.
- Save the data in a file. If you send data to a file, it will not go to the output device. The system prompts you for a statistics file name as described in the topic How to Output Measurement Results to a Statistics File.

When you run a routine, the data is first sent to a temporary file, C:\QVSI\BASICX\TEMP.STA. This file contains all the features sent during the most recent run of the routine.

- If the Partial Output parameter is 0 (Off), the software copies the data to the specified statistics file only **after** the run is completed. If the run is not completed for any reason, you can access the statistics data of the completed steps in the TEMP.STA file.
- If the Partial Output parameter is 1 (On; default), the software copies the data to the specified statistics file as each step is measured during the run.

If you want, you can change the path and name of the TEMP.STA file. To do so:

1. Select **System** / **Configuration** / **Editor** from the main menu.
2. If you want to change the name, find the **Statistics** section and specify the new name.
3. If you want to change the path, find the **Paths** section and specify the new path.

14.2.4 Templates for Statistics Output

eBx outputs measurement results in ASCII form. Five templates control the form and content of the statistics and data stream output. Each template has a maximum length of 80 characters.

To display the templates, click the **Statistics** tab in the **System** / **Configuration** / **Output** dialog box.

The templates format the data as described below. The default templates are shown below each description. The meaning of each letter is described under the Output Characters topic.

The Beginning of Run template formats the header for the statistics report.

@R @A @B @N^J

The Beginning of Feature template indicates the data to appear at the beginning of the feature. The default is a blank template. An example is shown below.

@W @S^J

The Output template formats the output for each measured feature.

@T @D^J

The End of Feature template indicates the end of a specific feature.

EOM^J

The End of Run template indicates the end of a specific run.

EOT^J

The two characters, @ and ^, are used together with letters to specify action:

- @ followed by a letter is a command to output a specific piece of information
- ^ followed by a letter outputs a control character such as carriage return and line feed

For example, the Output Template default string @T @D^J has the following meaning:

Output Tag - Output Data - Carriage Return&Line Feed

- The ^J character (carriage return and line feed) in the Output Template default string indicates that the tag and data values will be output in a column format. If this character is omitted, the statistics values will be output in row format.

- To specify a space as the first or last character of an output template or tag, substitute the ^` (caret, left tick) characters for each space. Your output template should read @D^` for the "Data Space" that you are generating.

EOM (End of Measurement) and EOT (End of Transmission) are default text messages that appear in the output data. You can enter your own message if you want.

The EOM and EOT text messages are followed by the control characters that send a carriage return and line feed.

14.2.5 Tags for Statistics Output

Tags are labels or identifiers that may be output with each type of measured value. To output a tag, the @T (Output Tag) character must appear in the statistics output template.

Whenever the general measurement value or location described by the tag is selected for statistics output, the tag for that value is transmitted or recorded with the actual measured value. For example, the tag for diameter is DIA. The output for a diameter measurement would look like this:

DIA +00.00000

To display the tags, click the **Tags 1** and **Tags 2** tabs in the System / Configuration / Output dialog box.

The tags associated with the @T output character are listed below. You can edit these tags using the mouse and keyboard. Each tag can consist of up to six characters. The default is 3 (two spaces are added for single-character tags: X, Y, Z, R, A).

Tag	Meaning
X	X location
Y	Y location
Z	Z location
R	R location
A	A location
ANG	Angle
ANG	Angle 1
ANG	Angle 2
ANG	Angle 3
ANG	Angle 4
ANG	Angle between
ANG	Supplementary angle
AVG	Average
CND	Cone diameter
CNR	Cone radius
CON	Concentricity
CYD	Cylinder diameter
CYL	Cylindricity
CYR	Cylinder radius
DIA	Diameter
DIA	Spherical diameter
DIS	Straight line distance

DIS Perpendicular distance

DIS 3D distance

DST Angularity

FLT Flatness

FTA Full taper angle

HTA Half taper angle

INC Inclination angle 1

INC Inclination angle 2

INP User input

MID Minimum diameter

MIN Minimum

MIR Minimum radius

MAX Maximum

MXD Maximum diameter

MXR Maximum radius

PAR Parallelism

PHD Height / depth

POS Position tolerance

PRM Profile minus

PRP Profile plus

PRP Perpendicularity

RAD Radius

RAD Spherical radius

RES Result

RND Roundness (circularity)

STR Straightness

WCA Centerline angle

WID Width

WID Minimum width

WID Maximum width

XD X distance

YD Y distance

ZD Z distance

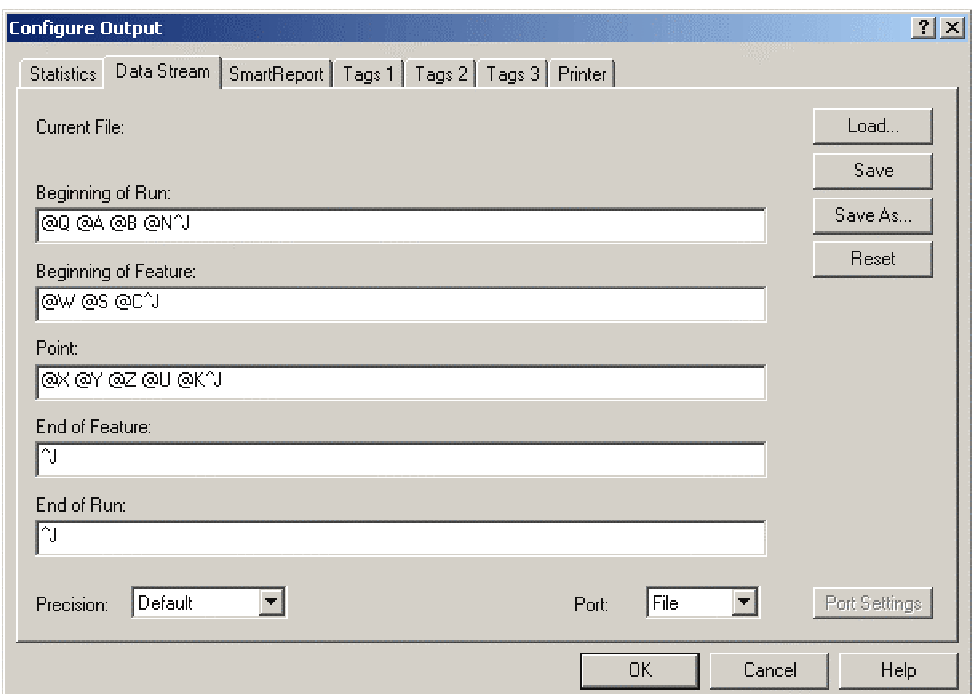
You can output collected stats data in a format that can be imported easily by spreadsheets such as Lotus 1-2-3 and Microsoft Excel. The format includes a space between each measurement result.

Each row is imported as a sample part and each column is a dimension. To specify a space as the first or last character of an output template or tag, substitute the ^` (caret, left tick) characters for each space. Your output template should read @D^` for the "Data Space" that you are generating.

14.2.6 Data Stream Output Configuration

This option allows you to configure the destination and format of the measurements being sent via data stream. When you click the **Data Stream** tab in the **System** / **Configuration** / **Output** menu, a dialog box appears where you can specify the:

- Templates to format the data so that you can read the results, and the data is compatible with your data collector
- Precision of the data stream values, which can be from 2 to 8 decimal places. The default is 0, which indicates the current system resolution (selected in the Units menu).
- Destination to indicate where the data can be sent: to a file or to a data collector or other output device connected to the serial (COM) port. If you select a serial port, the software displays a Port Settings button. You can click this button to configure the port. Any changed COM port settings are saved in the Com Ports section of the configuration editor so that they are restored the next time the measurement software is started.



You can save/load the data stream output configuration in the following ways:

- Click the **Save** button to save the configuration data currently in memory and close the dialog box. The data is saved in the file whose name is displayed at the top of the dialog box (or the default file if a name is not displayed).
- Click the **Save As** button to save the configuration data in a new file and close the dialog box. The default file extension is INI and the default folder is CONFIG.
- Click the **Load** button to open a different data stream configuration file and load the data into temporary memory. The system displays an appropriate message if you try to load an invalid file.
- Click the **Reset** button to reload the default data that are in the CONFIGURATION.INI file (in the QVSI\BASICX\CONFIG folder).

When you click the **OK** button, the software will use the data that are currently in memory. If you changed any of the data but you did not save it, the software displays a confirmation prompt asking if you want to continue without saving the data.

If you click the **Cancel** button, the software closes the dialog box and continues to use the data stream data that was in memory before you made any changes or loaded another file.

Regarding the output characters used for data stream output:

- W enables you to specify the type of feature being output and the X, Y and Z output characters enable you to output the XYZ values for each point in the feature.
- K is used to output the diameter of the current tip in the current units of the step. The @K character must be at the end of the POINT TEMPLATE line in the Configuration / Output / Data Stream dialog box (e.g., @X @Y @Z @U @K ^J). The line specifies that the XYZ values, units and probe tip diameter be output to the data stream for each point in a touch probe geometry step (line, arc, circle, sphere, plane). This character is valid for data stream output only. For example, a line output to the data stream would appear as shown below:

  +0.91811107 -0.06292034 -0.18337782 in
  +0.11792338
- Q is used to output the complete path and routine name.

🗉 The TP_POINTS_UNCORRECTED parameter in the INPUT.INI file controls whether or not the tip radius is included in the calculation of the touch probe data stream values. The parameter has one of two values:

0 = Output touch probe XYZ point values corrected by the tip radius. In this case the actual point of contact is output. This is the default.

1 = Output raw, uncorrected touch probe XYZ point values. In this case, the tip radius is not included in the calculation and the center of the touch probe tip is output.

🗉 Data stream and statistics output can be sent to the same file or printer.

🗉 When you load user-defined data stream templates, the software does not load in any of the user-defined tags.

14.2.7 Templates for Data Stream Output

The software outputs measurement results in ASCII form. Five templates control the form and content of the statistics and data stream output. Each template has a maximum length of 80 characters.

To display the templates, click the **Data Stream** tab in the System / Configuration / Output dialog box.

The templates format the data as described below. The default templates are shown below each description. The meaning of each letter is described under the Output Characters topic.

The Beginning of Run template formats the header for the statistics report.

@Q @A @B @N ^J

The Beginning of Feature template indicates the data to appear at the beginning of the feature. The default is a blank template. An example is shown below.

@W @S @C ^J

The Point template formats the output for each measured point.

@X @Y @Z @U @K ^J

The End of Feature template indicates the end of a specific feature.

^J

The End of Run template indicates the end of a specific run.

EOR^J

The two characters, @ and ^, are used together with letters to specify action:

- @ followed by a letter is a command to output a specific piece of information
- ^ followed by a letter outputs a control character such as carriage return and line feed

🗎 The ^J character (carriage return and line feed) in the Output Template default string indicates that the tag and data values will be output in a column format. If this character is omitted, the statistics values will be output in row format.

🗎 To specify a space as the first or last character of an output template or tag, substitute the ^` (caret, left tick) characters for each space. Your output template should read @D^` for the "Data Space" that you are generating.

EOR (End of Run) is the default text message that appears in the output data. You can enter your own message if you want. The EOR text message is followed by the control character that sends a carriage return and line feed.

14.2.8 Output Characters for Statistics & Data Stream Output

The output characters that can be used in the statistics or data stream templates are listed below. All output characters must be preceded by the @ sign.

A

Date. The current system date will be output.

B

Time. The current system time is output in the format specified on the computer.

C

Feature Report Text. If the selected feature has a comment, it is output.

D

Actual Data. The selected measurement values are output.

(not valid for data stream output)

E

Nominal. The nominal value for the dimension is output.

(not valid for data stream output)

F

Upper Tolerance. The upper tolerance for the dimension is output.

(not valid for data stream output)

G

Lower Tolerance. The lower tolerance for the dimension is output.

(not valid for data stream output)

Hx

Header Line, where "x" is a value from 1 to 5. Up to five lines of the routine header can be output.

(not valid for data stream output)

In

Footer Line, where "n" is a value from 1 to 5. Up to five lines of the routine footer can be output.

(not valid for data stream output)

Ix

Two-letter output character, where the first letter is I and the second letter, "x," is a letter from A to Z.

IE is the end point of a laser scan.

IS is the start point of a laser scan.

J

Deviation. The deviation between actual and nominal is output.

(not valid for data stream output)

K

Touch Probe Diameter. The diameter of the current tip (in the current units) is output (data stream only).

Laser Spacing. For laser features, the laser spacing in the "begin of feature" area is output (data stream only).

N

Run Number. The sequential run number of the routine is output.

P

Feature Prompt. If the selected feature has a prompt, it is output.

Q

Path, Routine Name. The complete path and routine name is output (data stream only).

R

Routine Name. The name of the routine is output.

S

Step Number. The step number in the routine is output.

T

Tag. The label is output (up to six characters).

(not valid for data stream output)

U

Units. The units associated with the value are output.

V

Date. The current system date is output in the format specified on the computer.

W

Feature Title. The type of feature (circle, line, etc.) is output.

X

X value. The X value is output. (data stream only)

Y

Y value. The Y value is output. (data stream only)

Z

Z value. The Z value is output. (data stream only)

The @Hx output character is very useful for logging your own tracking information in the statistics output. For example, to add your own heading such as "Operator" on the first line, along with the routine name and run number, specify the following in the Beginning of Run template:

OPERATOR @H1 @R @N^J

When you create or run the routine you can enter the actual "header" text and fill in the appropriate information such as the name of the operator.

- By default, all five header lines are labeled "Header Line." To include your own header lines (for example, "Operator" and "Batch") edit strings STR_158 through STR_162 in the language file.

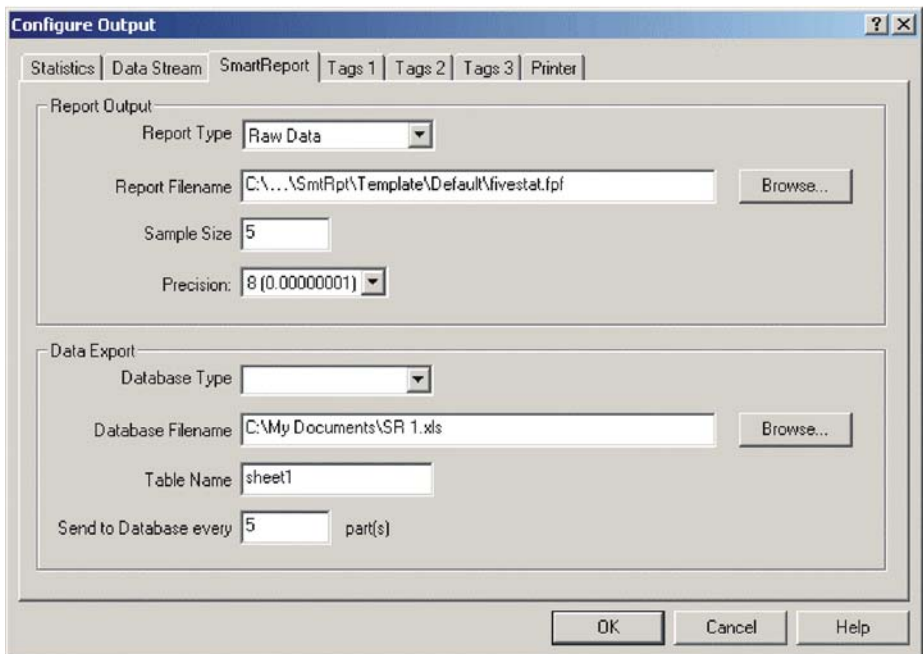
- A value of 2 can be specified for the STATS_DATE_FORMAT parameter in the INPUT.INI file to enable the date (@A or @V) to be output in a year-month-day format.

14.2.9 SmartReport Output Configuration

This option enables you to configure the:

- SmartReport print output going to the REPORT.STA file. This file is used as input by SmartReport to design and generate standard or custom inspection reports.
- SmartReport data export output going to the EXPORT.STA file. This file is used by SmartReport to export inspection results to a variety of databases and spreadsheets for further evaluation.

To display or change the SmartReport output configurations, click the **SmartReport** tab in the System / Configuration / Output dialog box.



In the top part of the dialog box (SmartReport print output) you can configure:

- **Report type**. You can select First article, First Article (Five Piece), Raw data, or Stat summary.
- **Report file** which indicates the path and name of the *template* file (C:\SMARTRPT\TEMPLATE\DEFAULT) and the SmartReport file that contains the default pre-designed format of the report data (can be customized with SmartReport). You can select one of the following files:
  - ONEPIECE.FST (First Article Report), which summarizes the record for a single inspected part. This includes general information about the part and a table containing rows for each part feature. Table columns contain information about observed values, nominal values, control limits, etc.
  - First Article (Five Piece): FIVEPORT.FPF (portrait), FIVELAND.FPF (landscape), or FIVESTAT.FPF (statistical), which summarizes the inspected data for up to five parts.
  - DATAFACT.RAW (default raw data) or DATASTAT.RAW (raw data with statistical calculations), which prints raw data from the routine run in report form.
  - STATSUM.STT (stats summary), which prints raw statistical calculations in table form.
- **Sample size.** This instructs SmartReport to output a report after a specified number of parts have been inspected. The value is set to 1 for First Article and it is usually set to 5 for First Article (Five Piece).
- **Precision** of the report output values, which can be from 2 to 8 decimal places. The default is 0, which indicates the current system resolution (selected in the Units menu).

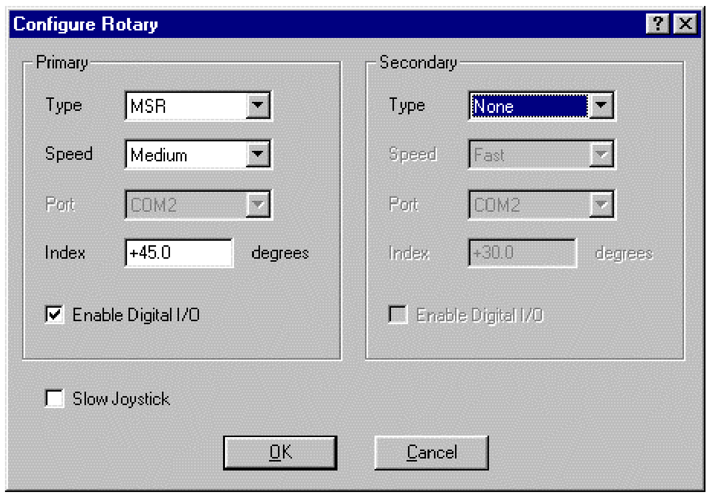
In the bottom part of the dialog box (SmartReport data export) you can configure:

- **Type** of the database, for example, Excel 5.0
- **Location and name** of the database file
- **Table name**, for example, Sheet1 for an Excel XLS file
- **How often** the data is to be sent to the database

All the information specified in this dialog box is saved with the part routine.

14.3 Rotary Configuration

If your system is equipped with a rotary indexers, you can indicate the type, speed and index step size in **System / Configuration / Rotary**.



For each rotary you must:

- Select the type of rotary indexer that is attached to the system
- Select the rotary indexer speed
- Specify the default index step size in degrees. This is the number of degrees that the indexer will move when you click the Index button in the Measurement window when Rotary Move is active.

You can also:

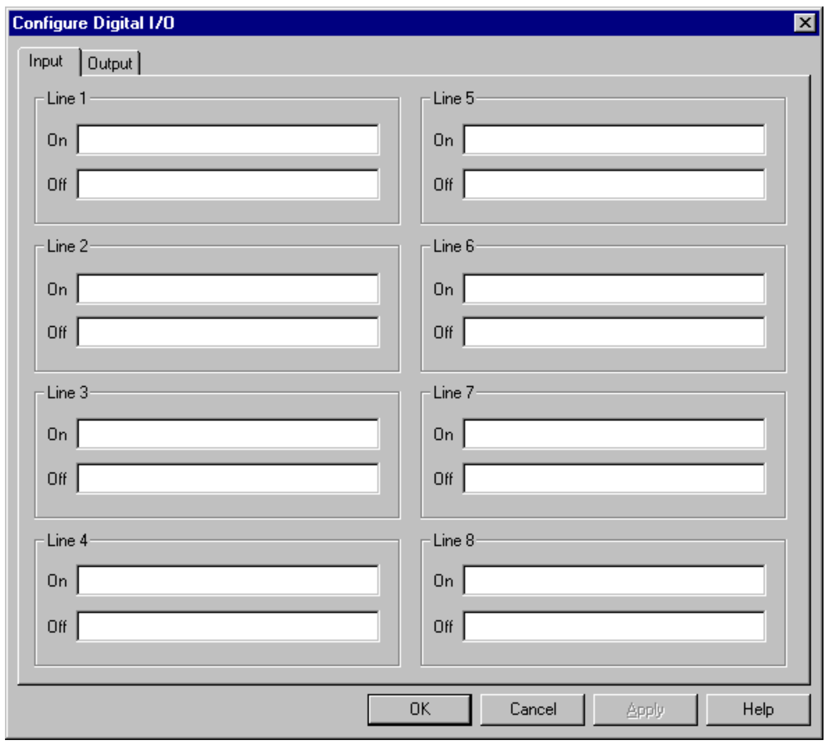
- Select the Digital I/O check box next to enable the digital I/O function with the selected rotary.
- Select the Slow Joystick check box next to reduce the maximum velocity of the joystick motion when a rotary indexer is selected.

14.4 Digital I/O Text

The Digital I/O option (in the System / Configuration menu) allows you to enter descriptive text for the on and off status of each line. This text explains the meaning of each bit.

The descriptive text is displayed in the Image window when you select
Measure / Digital I/O.

You can enter text both for the Input and Output steps. The default type of step is Output. To enter text for the Input lines, click the **Input** tab.



You can enter up to 60 characters for each on and off condition on Lines 1 through 8. To enter each line of text:

1. Place the cursor after the colon and press the left mouse button once.
2. Type the desired text. Characters will not wrap to the next line. Do not press Enter on the keyboard; it has the same function as OK.
3. Repeat Steps 1 and 2 for each line.

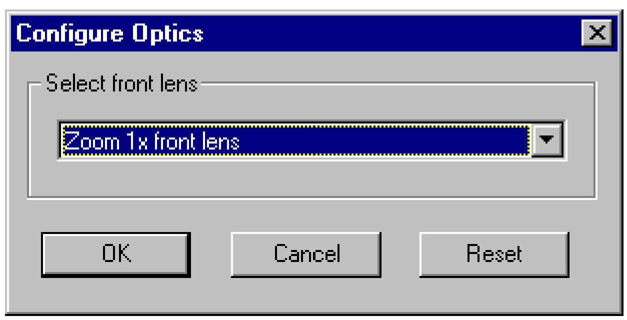
4. Click **OK** to accept the entries or changes, and to close the window.

Click Cancel to close the window without saving any new entries or changes. However, it saves any of your original text.

14.5 Optics Configuration

The base system is configured with no additional lens, i.e., only a 1X lens is built into the optics. The Optics option allows you to configure the lens attachment. This controls the magnification.

You must configure the additional lens each time you attach or remove an add-on lens. To do so, select **System** / **Configuration** / **Optics** from the main menu and select the lens attachment from the drop-down list.



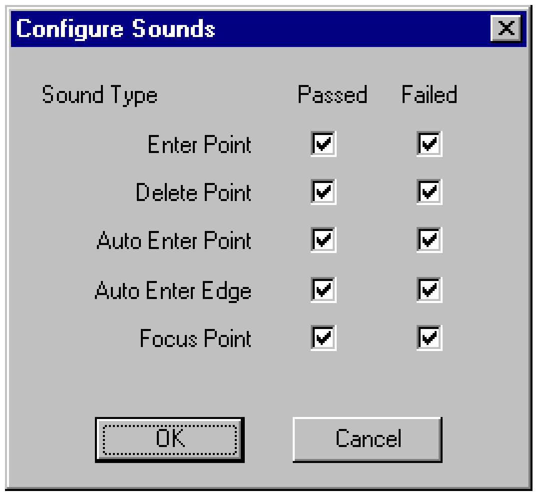
The Reset button clears all the field of view and zoom lens calibration values for all lenses. **If you click Reset, you must perform the field of view and zoom lens calibration for all lenses before you can use the system.**

The field of view calibration must be done anytime the optics configuration is changed.

14.6 Sound Configuration

This function configures the audio signals that serve as a guide or warning during the measuring process.

To configure the sound, select **System** / **Configuration** / **Sound** from the main menu. Then turn the sound on or off for any listed action by clicking in the appropriate box to specify when the sound should occur.



- Enter Point is activated when you use a Measure function during the creation of a routine. You hear a high-tone beep when you press Enter to accept the point. You hear a low-tone beep if the entry of the point is not successful.
- Delete Point is activated when you remove a point from a measurement. You hear a low-tone beep when you remove a measured point successfully or when you try to remove a non-existent point.
- Auto Enter Point is activated when you use the Weak Edge tool while running a routine. You hear a high-tone beep each time a valid (pass) edge is encountered or a low-tone beep each time an invalid (fail) edge is encountered.
- Auto Enter is activated if you used a Strong Edge tool while running a routine. You hear a high-tone beep each time a valid (pass) edge is encountered or a low-tone beep each time an invalid (fail) edge is encountered.

- **Focus** is activated when you use the Focus tool or you perform an autofocus. You hear a high-tone beep for each pass condition or a low-tone beep for each fail condition. The same beep is used for either edge or surface focus.

🗎 Even if all the audio signals are off, warning sounds are given if you use incorrect options or out of sequence operations, e.g., when you try to measure another feature before completing the current measurement.

14.7 How to Select a Different Language

The language file (the file suffix is .LNG) contains all the text that appears in the menus and windows. This includes prompts and messages.

The language file is specified by the Language File parameter in the INPUT.INI file. The default language file is ENGLISH.LNG.

To display a different language, select **System** / **Configuration** / **Language** from the main menu and then the desired language from the submenu. The software immediately displays all text in the selected language.

You can also copy the current eBx text into another language file. To do so, select **System** / **Language** / **Export Language** from the main menu and specify the desired file name in the displayed dialog box. The suffix is .LNG and the file is stored in the QVSI\BasicX\Language folder. Any new entries are added at the end of each section of the file.

This may be useful to create a customized version of a language file or to create a version of text in another language that is not listed in the submenu.

For example, you can use this method to update your custom language file after a software update. To do so:

1. Start the updated eBx software using your own custom language file.
2. Copy the current eBx text to a new file. This file contains your custom text and any new text; you do not need to re-enter your customized text.
3. Edit the new language file using a text editor and change any text you desire. You need to enter customized text only for new strings.
4. Exit from eBx and restart it. The menus and prompts are now in the new language.

The language file also contains the information that causes the underline to appear under selected letters in the eBx menus. You can select a menu item letter from the keyboard by pressing [**Alt**] and the underlined letter.

You can add the underline to a menu item letter by inserting an & (ampersand) character before the character to be underlined when editing the language file with a text editor. To insert the & character, put the cursor in the "insert" mode, place the cursor on the character to be underlined and key **&**. The underlined character then becomes the key character for selecting that menu item with the [**Alt**] key.

14.8 Menu Configuration

eBx supports a user-definable menu configuration, which allows or restricts access to entire menus or one or more items in a menu, including the associated functions in the toolbox and Model window. For example, this enables you to set one level of access for the computer system administrator or programmer, and another level for an operator.

The menu configuration information is stored in the MENU.INI file located in the Config folder. The default settings allow all menus and menu items to be displayed on the screen.

To set up and use a menu file with different access settings, follow the steps below.

1. Select **System** / **Write Menu File** from the main menu. The software displays a dialog box for the file name.
2. Specify the path (C:\QVSI\BASICX\CONFIG) and file name and click **OK**. The software writes the current menu information to the file.
3. Edit the file using a text editor. To restrict access to a menu item, change its value to **0**.
4. Save the changes in the file.
5. To use the changed file (e.g., operator1.ini), create another eBx shortcut on the desktop and change the command line to:

   C:\QVSI\BASICX\BX.exe -update operator1.ini

If the file name has spaces in it, the entire name must be encompassed in double quotes.

The changed menu configuration will be active when you run eBx using the changed shortcut.

If you want to read in any changes that you have made since the software was started, select **System** / **Load Menu File** from the main menu. The software displays a dialog box to select the .ini. file. When you select the file and click OK, the software loads the file and enables or disables the applicable menu items.

- The system administrator is responsible for securing the menu configuration files to prevent unauthorized access or use. For example, the administrator may want to create separate files such as admin.ini with full access settings and operator.ini with restricted access settings.

- If a menu configuration file is not specified on the command line, the software loads the MENU.INI file to determine the menu configuration. If this file has been changed, delete the file before starting eBx to restore the system to full access.

14.9 Configuration Settings

Configuration parameters control the general operation of the system. Any user can change these parameters.

You can change the following configuration settings when you select **System** / **Configuration**:

- Statistics, data stream, and SmartReport output configuration options
- Rotary
- Digital I/O
- Optics
- Sound
- Touch probe tips
- Select a different language
- Load and save a menu configuration file
- Edit the configuration file parameters

Once you have entered the configuration settings, they remain in effect every time the system is started. The only time you may need to change them is when you update the software or change peripheral devices.

🗎 The current settings (which tools/targets are displayed, default units, etc.) are saved in the Default Preferences section in the Preferences.INI file (located in the QVSI\BasicX\Config folder) when you exit from the measurement software. These become the default settings when you power up again.

14.9.1 Configuration Files

The system configuration parameters are in the following configuration files, which are located in the Config folder:

- Calibration.INI contains all the calibration parameters.
- Configuration.INI contains all the configuration parameters.
- Diagnostics.INI contains the advanced diagnostics parameters.
- Focus.INI includes autofocus parameters such as backoff factor and number or retries.
- Hardware.INI contains the parameters related to the system hardware such as camera, joystick, stage, etc.
- Input.INI includes file names, paths, graphics and analysis tools.
- Menu.INI contains all the settings that allow or restrict access to menus.
- Preferences.INI contains all the default setting used for system startup, e.g., the tool/target displayed and their colors.
- Touchprobe.INI contains all the touch probe calibration parameters.

To edit any parameter in any file, select **System** / **Configuration** / **Editor**.

14.9.2 How to Edit Configuration Parameters

To access and view or change any of the configuration parameters in any of the configuration files:

1. Select **System** / **Configuration** / **Editor** from the main menu. The system displays a dialog box where you can search for and display the parameters.

   The drop-down list in the dialog box allows you to access all the parameters in all the configuration files by section.

2. Find the desired section in the drop-down list and click it to select it.
3. Double-click the parameter value that you want to change.
4. Make the desired change.
5. Press **Enter** on the keyboard to accept the change or press **Esc** to cancel the change.
6. If you want, change other parameters in the section and repeat Steps 3 to 5. If you need to restore the parameters to their previous settings, click the **Restore Section** button. However, you must do this before clicking on Save Section; once you have clicked Save Section, you will not be able to restore the parameters.
7. Click the **Save Section** button after changing the parameters in the section.
8. Repeat Steps 2 through 7 for parameters in other sections.
9. Click the **Save All** button after you complete changing all the desired parameters.

If you need to restore the parameters to their previous settings, click the **Restore All** button. You must do this before clicking Save Section or Save All; once you have clicked Save Section or Save All, you will not be able to restore the parameters.

Click the **Defaults** button to save listings of all the parameters and their default values. The listings are saved in the respective configuration files in the selected folder.

Click **Find** to search for a parameter.

Click **Comments** to display the comments for the current section.

Click **Close** to exit from the configuration parameter editor.

The configuration files should be edited only by an authorized field engineer or service representative. Do not make any unnecessary changes to the .INI files yourself. Inappropriate changes may cause physical damage to the system. If you have questions about the .INI files, call your Customer Service representative.

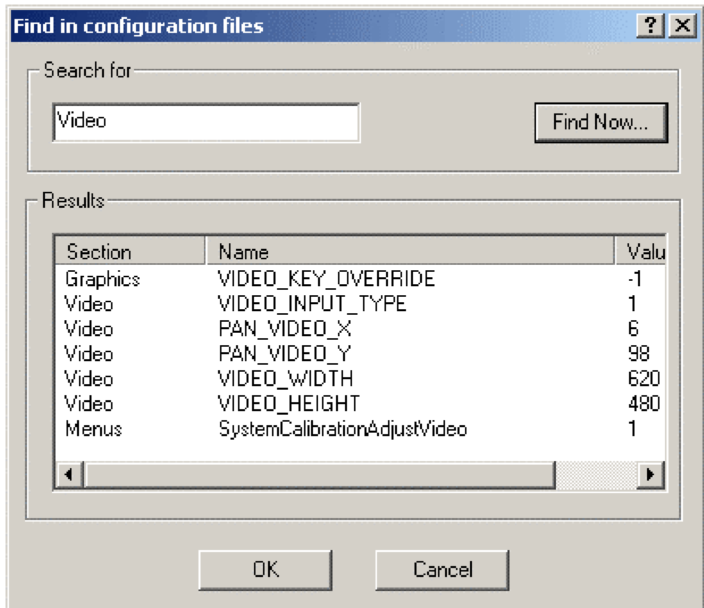
If the LED_INSTALLED or LENS_POSITIONS parameters in the AccuCentric section are changed, the software displays a message stating that all lenses will need to be re-calibrated and it resets the optics.

14.9.3 How to Search for a Configuration Parameter

To search for a configuration parameter while in the Editor window:

1. Click the **Find** button.

   The software displays a Search dialog box.



2. Type the desired parameter name or value in **one** string.
   - The software searches for all occurrences of that string or value.
   - The software does not search for section names and it does not perform the search if more than one string is entered.
3. Click **Find Now**.

   The found names are displayed in the Results area.
4. Highlight the desired name and click **OK**, or double-click a name.

   This displays the section in which the parameter is located. This also closes the Search dialog box.

15. Calibration & Alignment

15.1 Overview

This section describes the various options in the System / Calibration menu.

15.2 Calibration Parameters

Calibration parameters affect video, measurement and stage positioning accuracy. Only authorized users can change these parameters.

The calibration mode must be enabled to access the calibration options and perform the system calibration. The procedures to enable the calibration mode are described in the *Calibration and Alignment Manual* for your system.

To access the calibration options summarized below, select System / Calibration and then the desired option. Detailed information about the options and the calibration procedures are described in the *Calibration and Alignment Manual* for your system.

- Stage calibration optimizes stage positioning accuracy over the entire measuring area. The procedure measures a known standard (certified grid reticle) and applies correction factors to remove any residual errors.
  - With linear calibration, the same correction factor is applied over the entire measuring area.
  - With non-linear calibration, the correction factors vary depending on the combined X-Y position of the stage.

- Video - This procedure adjusts the video image in the Image window. It affects the field of view calibration.
- Lights - This procedure is used when you have more than one system and you want to run the same routines on all machines, without modification. After designating one machine as the "master," use this procedure to set the light levels on the other machines so that they match.

  This procedure can also be used to set light offsets for each individual zoom position. To do so, select the desired magnification and then select the desired light level in the Lights window. If you want to apply the selected light levels to all the zoom positions, make sure the All Mags check box is selected.
- Optics - This procedure performs the following calibrations:
  - Video to VGA to align the fixed graphic on the video board with the movable cross in the live video image
  - Field of View so that an edge can be detected and measured accurately anywhere in the field of view.
  - Zoom calibration, which calibrates the magnification throughout the entire zoom lens range.
- Autofocus - This procedure is used to correct astigmatism errors that are present in some lens systems.
- Dock Station - This procedure calibrates the location of each port and detachable stylus module (DSM) if your system is equipped with the off-axis touch probe. See the *Touch Probe User's Guide* (PN 790424) for more information.
- Probe Tip - This procedure calibrates the stylus diameter if your system is equipped with a touch probe.
- Sensor Align - These procedures establish the following offsets:
  - Sphere to Optics - This procedure calibrates the position of the reference sphere optically if your system is equipped with a touch probe.
  - Laser to Optics - This procedure establishes an approximate centerline offset from the laser to the optics if your system is equipped with a laser sensor.

16. Diagnostics

16.1 Overview

Select **System / Diagnostics** to access the following diagnostic tools:

- Basic diagnostic tools, which can be used for a more detailed analysis of measurements with the selected software tools.
- Advanced diagnostic tools, which are used for troubleshooting and diagnostic tests by authorized service technicians.
- Diagnostic corrections, which are used for troubleshooting and diagnostic tests by authorized service technicians.

> Using the diagnostics slows down the image processing because all the points need to be plotted. You may want to use it only if you encounter problems with the measurement tools.
>
> The diagnostics mode needs to be enabled for the advanced diagnostics. For example, you can use the DIAG.ENG file and place it in the Config folder for this purpose.

16.2 Basic Diagnostics

The basic diagnostic tools are used to enable graphics for selected software tools to do a more detailed analysis of measurements with those tools.

Select **System** / **Diagnostics** / **Basic** from the main menu to display the following diagnostic tools, which can be enabled by clicking in the appropriate check box in the Measurement window:

- Focus graphics, which are displayed when you use a Focus tool.
  - Points plotted in a bell-shaped contrast curve in a box in the lower-right part of the Image window to indicate the contrast level. The top of the curve displays the calculated Z axis position with the maximum contrast (sharpness). If the curve is oddly shaped, this may indicate that an autofocus calibration may be necessary.
  - Measured and calculated focus results in the upper-left portion of the Image window.
- Weak Edge graphics, which are displayed when you use a Weak Edge Point tool. The weak edge measurement displays two sets of weak edge points:
  - The actual (selected) weak edge points, which are larger and have the same color as the tool
  - All the other possible points that were used to calculate the actual weak edge, which are smaller and are shown with another color
- Strong Edge graphics, which displays a contrast curve when you use the Strong Edge tool.
- Laser graphics, which are displayed when you use the Laser tool.
  - Evaluated points are shown in a box in the Image window. The width of the box is a graphical representation of the laser sensor's range. The height of the box indicates what the laser finds. A vertical line indicates the center of the laser capture range; a horizontal line indicates the current threshold level, which is set in the Advanced Laser Settings window.

- The laser data are displayed in white and the graph is shown in the color of the current tool.
- Points in the middle (spikes) are typically the best points. Points below the threshold bar indicate reflections and noise. A point above the threshold bar displays the calculated Z axis position with the maximum contrast.

- **Engineering** check box. This turns on the display of more advanced weak edge graphics. This checkbox does not appear unless there is a DIAG.ENG file in the Config folder.

When you move the Accucentric light level slider, the system displays an image of the LED reticle and changes the light intensity in the Image window. This is used for the alignment of the LED reticle (described in the *Calibration and Alignment Manual* for your system).

The Grid Light check box enables you to toggle the grid light (if equipped) on and off. This is not related to any diagnostic function or dialog box.

16.3 Advanced Diagnostics

The advanced diagnostic tools are used for troubleshooting and diagnostic tests by authorized service technicians.

Select **System** / **Diagnostics** / **Advanced** from the main menu to display the following diagnostics, which can be selected by clicking the appropriate tab in the Measurement window:

- Focus
- Servos
- Video
- Stage
- Laser
- SmartRing

The diagnostics mode needs to be enabled for the advanced diagnostics. For example, you can use the DIAG.ENG file and place it in the Config folder for this purpose.

16.3.1 Advanced Diagnostics - Focus

Select **System** / **Diagnostics** / **Advanced** from the main menu and click the **Focus** tab to troubleshoot the Focus tool.

The Measurement window displays the step size and S-curve correction factors for all 25 calibrated zoom positions (1 = lowest mag, 25 = highest mag).

- The step size is the distance (in microns) traveled in Z between video frames during an autofocus.
- The matching S-curve correction (also in microns) compensates for variations in timing when the focus data is collected.
- If you need to change the step size or correction value, double-click the value and change it. The new value is updated automatically.

Click the **Find Step Size for Current Position** button to find and update the step size for the current zoom position. The software changes to an Advanced Focus tool (if it is not displayed already), performs a Reset focus, and updates the step size and S-Curve correction value in the table.

16.3.2 Advanced Diagnostics - Servos

Select **System** / **Diagnostics** / **Advanced** from the main menu and click the **Servos** tab to troubleshoot the servos.

The Measurement window displays servo parameters for the different types of servos.

- Select the servo type from the drop-down list.
- Double-click any parameter that you want to change. The new value is updated automatically.

16.3.3 Advanced Diagnostics - Video

The advanced diagnostic tools are used for troubleshooting and diagnostic tests by authorized service technicians.

Select **System** / **Diagnostics** / **Advanced** from the main menu and click the **Video** tab for troubleshooting data in the Image window.

In the Measurement window you can perform standard RGB, light intensity and noise tests.

- For the standard RGB test, select the desired size (width and height, in pixels) of the test box, which gets attached to the cursor, click **Start** and move the cursor in the Image window.
- For the Noise test, click the **Start** button. The signal to noise ratio should be above 40 dB. The focus pixel noise should be between 3.0 and 4.0.
- For the intensity histograms, select the desired radio button and click **Start**. Then check that the signal image is straight and even as it follows an edge.
- For the Screen Intensity test, click **Start** to display an array that shows the distributions of the light intensities in the Image window.
- For the Dust test, click **Start** to display a small box attached to the cursor. When you move the cursor in the Image window, the software calculates the contrast inside the box and provides a numerical value of camera dust contamination on the CCD or IR cut filter.

Click the appropriate **Stop** button to stop the test. Remember to do this before going to another tab.

16.3.4 Advanced Diagnostics - Stage

Select **System** / **Diagnostics** / **Advanced** from the main menu and click the **Stage** tab to troubleshoot stage motion.

The Measurement window displays the following diagnostics, which allow you to "torture" the system by constant motion of its Zoom, X, Y, and Z drives.

- Select the check boxes to select the desired drives (axes).
- Select the **SmartTorture** check box to enable random motions.
- Select the **Output Results** check box to store the test results in the RANDOM.TXT file in the Config folder.
- Click **Start** to begin the motion. Remember to Stop before clicking a tab to go to another window.

16.3.5 Advanced Diagnostics - SmartRing

Select **System** / **Diagnostics** / **Advanced** from the main menu and click the **SmartRing** tab to troubleshoot the programmable ring light illumination.

In the Measurement window you can perform the following diagnostic tests:

- Specify the interval (duration) of the test in seconds.
- Click the **Browse** button to send the test results to a specified file.
- Display the current temperature during the test.

Click **Start** to begin the test. Remember to Stop before clicking a tab to go to another window.

16.3.6 Advanced Diagnostics - Laser

Select **System** / **Diagnostics** / **Advanced** from the main menu and click the **Laser** tab to troubleshoot the Laser tool.

In the Measurement window you can perform laser diagnostic tests.

- Click the **Detect** button to check if the laser is attached to the system.
- Click the **Turn On** button to turn and leave the laser on and click the **Turn Off** button to turn it off. When the laser is turned on, the blue laser LED (visible through the optics cover) is also turned on.
- Change the threshold level, e.g., lower it, to gather data about the laser performance. Valid values are 0 to 255.

16.4 Diagnostic Corrections

The correction factor diagnostic tools are used for troubleshooting and diagnostic tests by authorized service technicians.

Select **System** / **Diagnostics** / **Corrections** from the main menu to display the diagnostic correction factors.

The dialog box indicates which correction factors are enabled (check mark in the box).

You can do the following in this dialog box:

- Enable or disable specific correction factors
- Click Center Corrections to view the FOV graphical representation
- Select a non-linear correction factor file, display its contents, and reload it if changes were made
- Specify desired XYZ location values and apply all the checked correction factors to those values

The diagnostics mode needs to be enabled for the advanced diagnostics. For example, you can use the DIAG.ENG file and place it in the Config folder for this purpose.

17. Manual Rotary Motion

17.1 Overview

Use the System / Rotary Move function to manually rotate (position) the rotary indexer, e.g., when setting up the part. This option is available only if a rotary indexer is installed and the rotary has been configured in System / Configuration / Rotary.

The system displays the current rotation angle of the rotary and the default incremental rotation amount (shown in degrees) specified in the System / Configuration / Rotary window.

Use the following controls for incremental moves, setting the zero position, and fine adjustment.

- Click the **Home** icon to set the current position of the indexer to 0,0. This is disabled if Enable Manual Positioning or Foot Switch is active.
- If you want, you can change the incremental rotation amount. This overrides the default number of degrees specified in System / Configuration / Rotary.
- Click the **Index** icon to move the rotary indexer by the specified number of degrees.
- Click the **Go To Home** icon to go back to the 0,0 position.
- Click the **left arrow buttons** to move the rotary indexer in a counter-clockwise (CCW) direction in increments of one degree or a tenth (0.1) of a degree.
- Click the **right arrow buttons** to move the rotary indexer in a clockwise (CW) direction in increments of one degree or a tenth (0.1) of a degree.

- Select the **Enable Foot Switch** check box to enable the foot switch (if installed). When you press the foot switch itself, the rotary indexer moves in the increments that are set with the switches in the rotary power supply I/O box. This is disabled if the MSR rotary is selected.
- Select the **Enable Manual Positioning** check box to unlock the motor to allow manual positioning. This is disabled if the MSR rotary is selected.

17.2 How to Enable Digital I/O

The Digital I/O function allows the software to communicate with other factory automation systems and control external devices. For example, this function is used to send and receive commands from external devices such as indexers, robots and automated fixtures.

To enable the Digital I/O function and communication port:

1. Select **System** / **Configuration** / **Rotary** from the main menu.
2. Select a rotary indexer.
    - If you select LPR, the Digital I/O communication is done through a COM port. In this case you can send data out but not receive data.
    - If you select MSR, the Digital I/O communication is done via the Digital I/O connector (on the side of the machine) that is connected to the DSP board. In this case you can send and receive data.
3. Select the **Enable Digital I/O** check box.

18. Using the CAD Navigator

18.1 Overview

The CAD Navigator allows you to import a CAD file and generate measurement steps automatically. Nominal values for the measured features are automatically imported from the CAD file. If the system is configured to display the Single Monitor User Interface, click the **Show CAD** icon in the Model window toolbar or click the **CAD Navigator** tab to display the CAD Navigator.

Supported two-dimensional CAD file types:

- DXF (standard)
- IGES (optional)
- Gerber (optional)
- Excellon (optional)
- HPGL (optional)

The CAD Navigator is an optional software tool. Contact your local authorized RAM Representative for more information.

CAD files contain lines, arcs, and circles. Additional feature relationships (for example, widths and lengths) required for complete inspection need to be constructed after the feature steps are automatically generated.

The CAD Navigator is only available if the system is configured to display the Single Monitor User Interface or Dual Monitor User Interface.

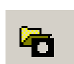
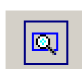
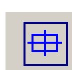
This section describes how to:

- Import a CAD file
- Generate measurement steps from a CAD file

Both procedures in this section use the QVI Training Part. It is assumed the part is securely mounted on the stage and that the manual part setup (described in Section 4) has been done.

18.2 CAD Navigator Toolbar

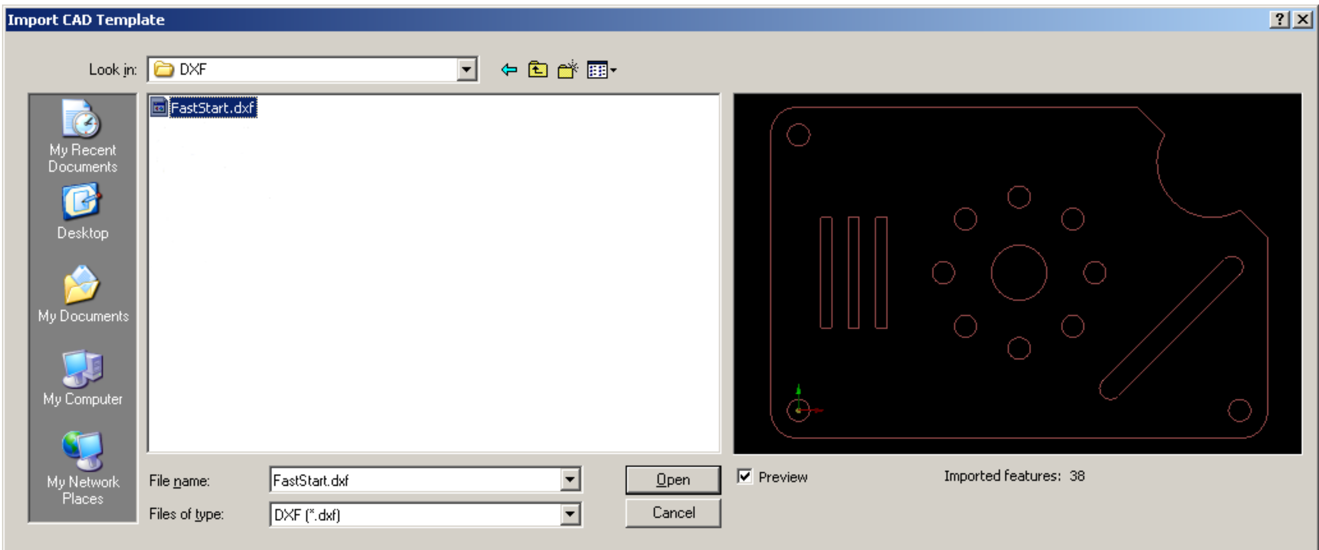
The CAD Navigator toolbar appears at the top of the CAD Navigator and contains several icons that invoke different CAD-related functions, as defined in the table below.

Function	Click this icon …	To do this…
Import CAD		Import a CAD file
Zoom to FOV		Zoom the Model window to match what is displayed in the Image window (current field of view)
Lock FOV		Link the Model window to the Image window – when you drive the worktable, the Model window displays the same area on the part that is currently displayed in the Image window (field of view)
Overlay Image		Display a mask of the CAD file over the video image in the Image window
Automatic Step Generation		Generate measurement steps from the CAD file automatically

18.3 Importing a CAD File

1. Select **File** / **Import CAD** from the main menu.

   The Import CAD Template window appears.

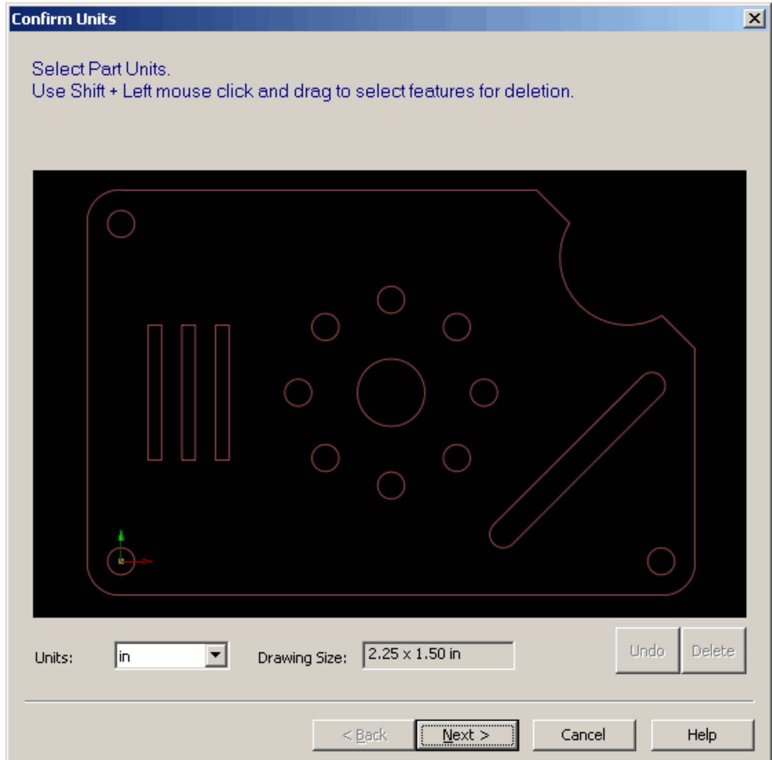


2. Browse for and select the desired CAD file (for example, FastStart.dxf), which can be in any folder on your computer or network.

   - To view the CAD file before actually opening it, select the **Preview** check box.
   - Use the **Files of type** drop-down list to specify the type of CAD file you want to import.

3. After selecting the CAD file you want to import, click **Open**.

   The Confirm Units window appears.



4. In the Units drop-down list, select the measurement units (inches or millimeters) used in the CAD file.
5. [Optional] Press and hold the **Shift** key on the keyboard, and then click the feature(s) you want to delete. When finished, release the Shift key and click **Delete**.

🗎 To zoom in/out, press the left mouse button while dragging the mouse up/down.

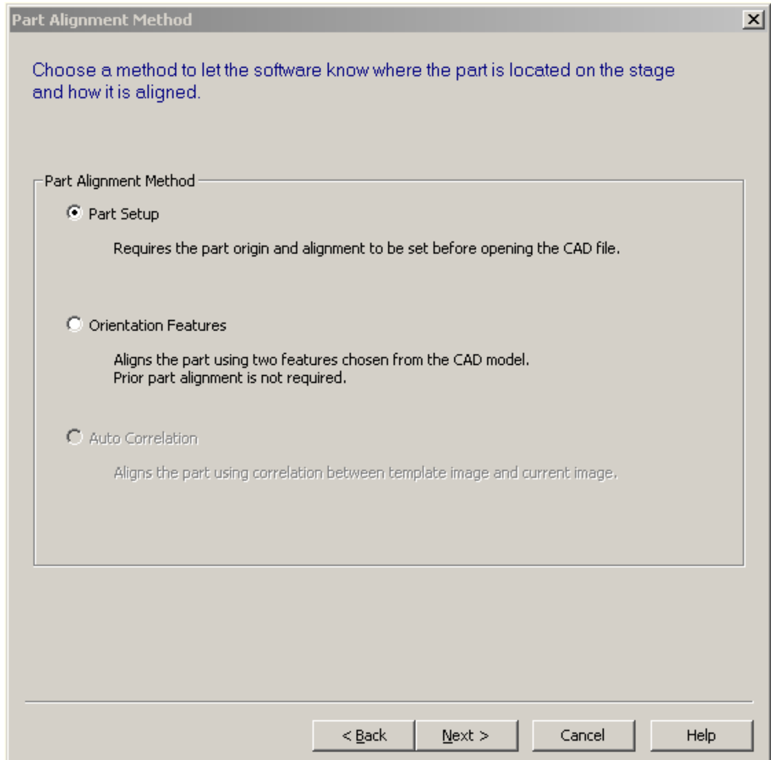
🗎 To pan the CAD model, press the middle mouse button while dragging the mouse in the desired direction.

🗎 To rotate the CAD model, press the right mouse button while dragging the mouse in the desired direction.

🗎 If you select a feature by mistake, press and hold the Shift key on the keyboard, and then click the feature(s) you want to unselect.

6. After selecting the measurement units and deleting any unwanted features, click **Next**.

   The Part Alignment Method window appears.



7. Specify how you want to indicate where the part is located on the stage and how it is aligned by selecting the desired part alignment method.

   - Part Setup – Specify the features that were used to manually align the actual part on the stage. This method assumes you manually aligned the part before starting the CAD import process.

   - Orientation Features – Specify two features that you want to use to orient the part on the stage to the CAD model. This method assumes you did not manually align the part before starting the CAD import process.

8. Click **Next**.

   Depending on the selected part alignment method, either the Part Setup or Orientation Features window appears.

9. If you selected the **Part Setup** part alignment method, do the following:

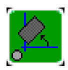
   1. Position the mouse cursor over the feature you want to use to set the XYZ origin (for example, use the lower-left hole in the QVI training part).

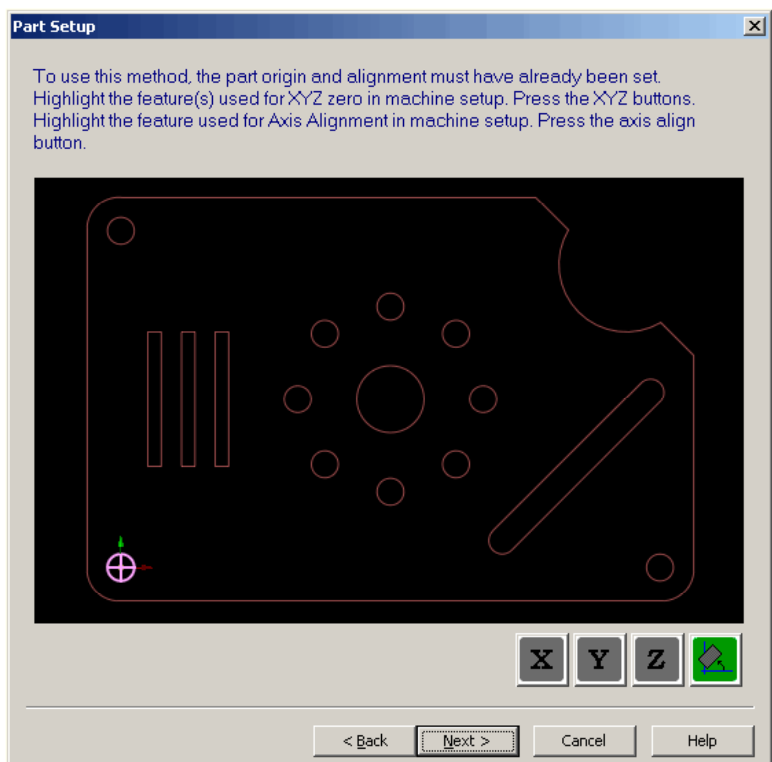
   2. Move the mouse cursor until a white cross appears. Then press the left mouse button to select the feature. The cross changes color.

   3. Click , , and  to set the XYZ origin at the selected feature. The feature color changes to yellow.

   4. Position the mouse cursor over the feature you want to use to set the axis alignment (for example, use the lower-right hole in the QVI training part).

   5. Move the mouse cursor until a white cross appears. Then press the left mouse button to select the feature. The cross changes color.

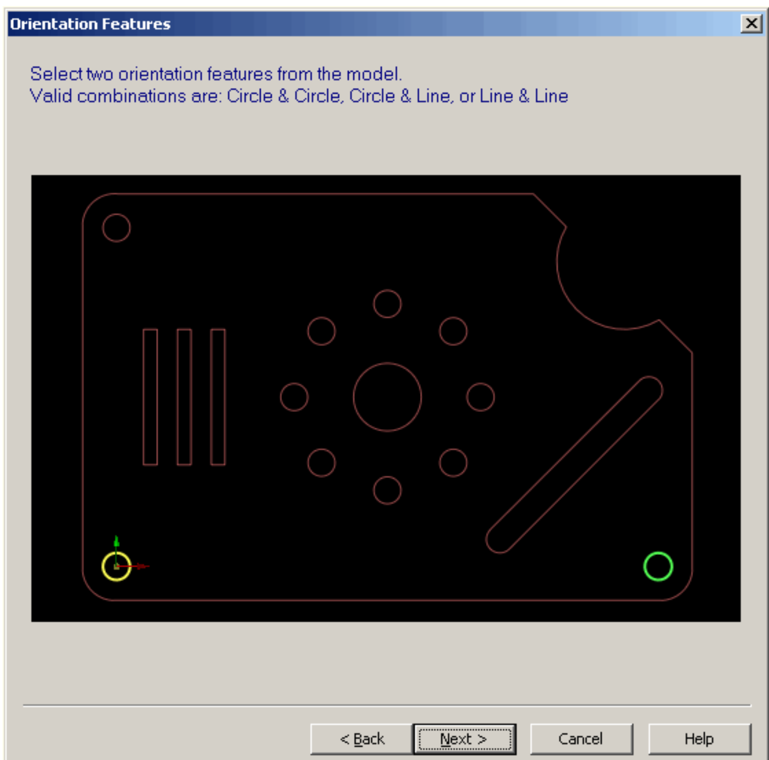
   6. Click  to set the axis alignment at the selected feature. The feature color changes to green.



10. If you selected the **Orientation Features** part alignment method, do the following:

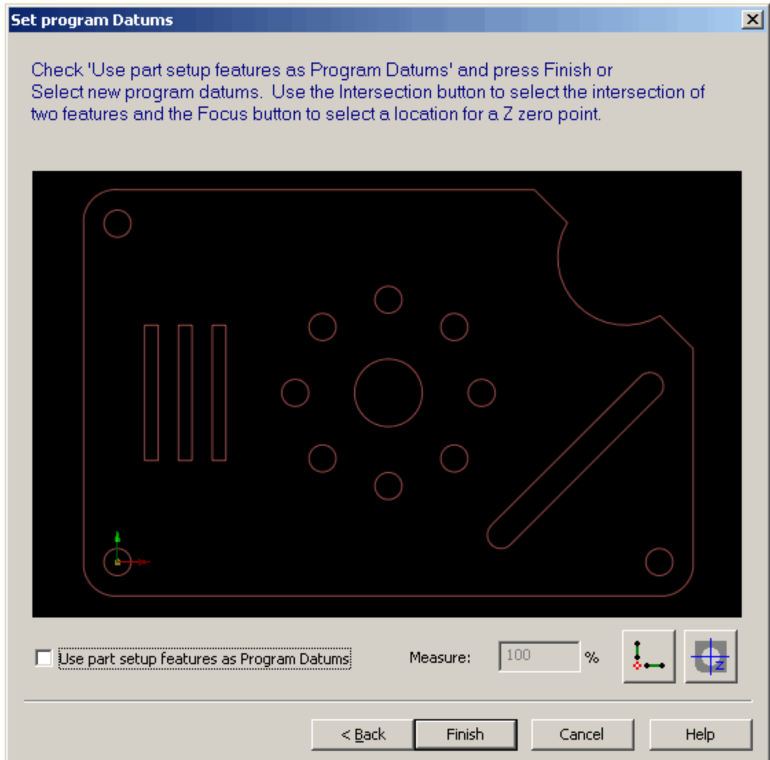
    1. Position the mouse cursor over the feature you want to use to set the XYZ origin (for example, use the lower-left hole in the QVI training part).
    2. Press the left mouse button to select the feature. Feature color changes to yellow.
    3. Position the mouse cursor over the feature you want to use to set the axis alignment (for example, use the lower-right hole in the QVI training part).
    4. Press the left mouse button to select the feature. Feature color changes to green.

Valid orientation features include circles, arcs, and lines. Any combination of the three can be used.



11. Click **Next**.

The Set Program Datums window appears.



12. [Optional] Click the **Focus** icon and then click any point on the CAD model to create a Z datum origin at that location.

13. [Optional] Specify the features you want to use to create XY datum origin and datum alignment steps by doing one of the following:

    - Select the **Use part setup features as Program Datums** check box to create datum origin and datum alignment steps automatically using the features selected for part alignment. Use the percent box to indicate what percentage of each datum feature should be measured if the feature is larger than the field of view (the default is 100%).
    - Select the feature you want to use for the XY datum origin, and then select the feature you want to use for the datum alignment.
        - To select a circle, arc, or line, verify the Intersection icon is not "pushed in" and then click the desired feature(s) in the CAD model. Use the percent box to indicate what percentage of each datum feature should be measured if the feature is larger than the field of view (the default is 100%).
        - To select an intersection, click the **Intersection** icon and then click the two features that make up the intersection. When finished, click the Intersection icon again.

14. Click **Finish** to import the CAD file.

    - If you selected Part Setup as the part alignment method and you selected datum features in Step 12 and/or Step 13, the system measures the features used to define the datum features, and then constructs the datum features from those features. This completes the CAD import process.
    - If you selected Orientation Features as the part alignment method, the first orientation feature blinks in the Model window. Perform the next step to continue the CAD import process.

15. If the first orientation feature is a circle or line that fits within the field of view, use the joystick to drive the stage so the feature appears in the Image window, within the measurement area of the displayed tool. Then click Measure in the Setup orientation features window.

    If the first orientation feature is a circle or arc that is larger than the field of the view, use the joystick to drive the stage so the feature appears in the center of the Image window. Then select three points on the circle or arc by clicking them in the Image window.

16. Click **Next** in the Setup orientation features window.

    The second orientation feature blinks in the Model window.

17. If the second orientation feature is a circle or line that fits within the field of view, use the joystick to drive the stage so the feature appears in the Image window, within the measurement area of the displayed tool. Then click Measure in the Setup orientation features window.

    If the second orientation feature is a circle or arc that is larger than the field of the view, use the joystick to drive the stage so the feature appears in the center of the Image window. Then select three points on the circle or arc by clicking them in the Image window.

18. Click **Finish**.

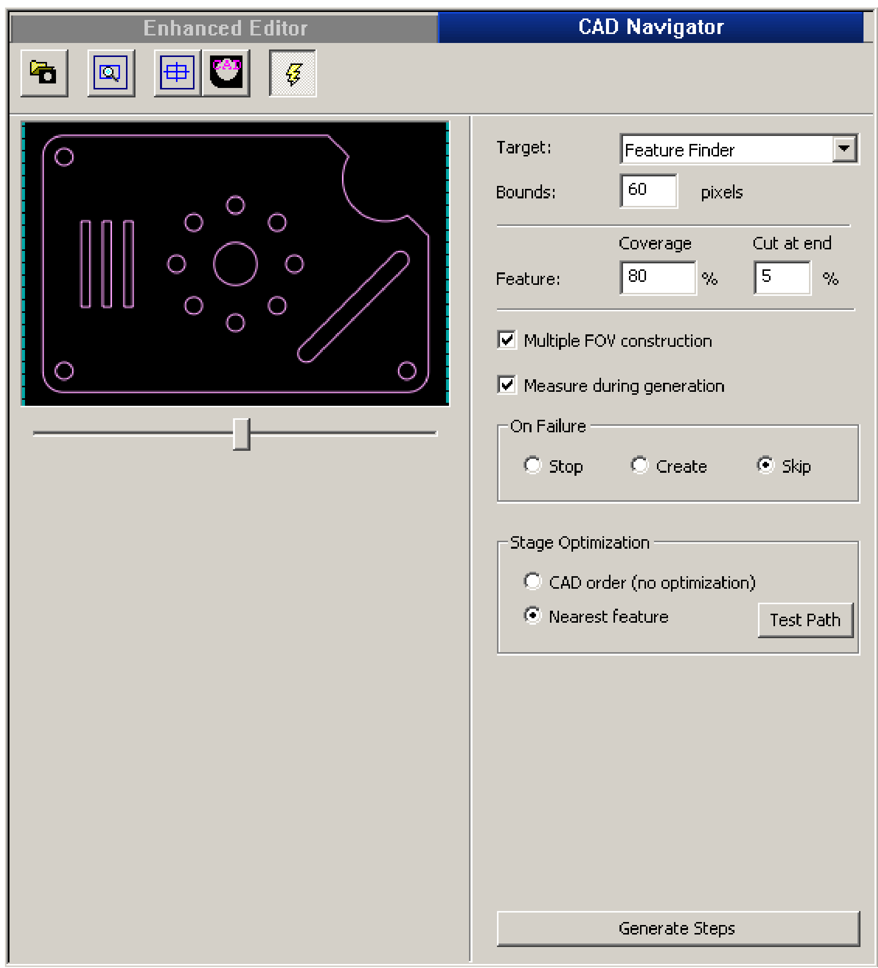
    If you selected datum features in Step 12 and/or Step 13, the system measures the features used to define the datum features, and then constructs the datum features from those features. This completes the CAD import process.

18.4 Automatic Step Generation Window

Click  in the CAD Navigator toolbar to display the Automatic Step Generation window, which includes various options for generating measurement steps from a CAD file.

- Use the **Target** drop-down list to select the tool (FeatureFinder or Strong Edge Finder) you want the system to use to measure the selected feature(s).
- Use the **Bounds** box to specify the size of the target search area.
- Use the **Feature Coverage** box to specify how much of the feature will be measured if the feature is larger than the field of view.
- Use the **Feature Cut at end** box to specify how much of the feature will not be measured at the end. This is helpful when you have adjoining features and you want to make sure the system measures the correct feature.
- Select the **Multiple FOV construction** check box to measure sections of the feature (if the feature is larger than the field of view), and then construct a complete a feature from the measured features. If left unchecked, the system measures the sections of feature, but does not construct the complete feature.
- Always select the **Measure during generation** check box to ensure the system measures the selected features when generating the measurement steps.
- Specify what you want the system to do if a measurement fails.
  - Select the **Stop** radio button to stop generating steps.
  - Select the **Create** radio button to create the step even though the measurement failed. You can edit the step later to correct the problem.
  - Select the **Skip** radio button to skip the step. You can manually insert the step later.

- Specify how you want the system to measure the selected features.
  - Select the **CAD order** radio button to measure the selected features in the order they were created in the CAD model, regardless of their location on the part.
  - Select the **Nearest feature** radio button to measure the feature closest to the current field of the view first, and then measure the remaining features in order from nearest to farthest.



18.5 Generating Measurement Steps from a CAD File

1. Set the magnification and illumination to the desired levels.
2. Click  in the CAD Navigator toolbar.
3. Change the desired settings in the Automatic Step Generation window.

> Click **Test Path** to view the path the system will take to measure the selected features.

4. Select the feature(s) you want to measure (by default, all features are selected).
   - To select a single feature, click the desired feature in the Model window.
   - To select multiple features, do either of the following:
     - Press and hold the **Ctrl** or **Shift** key on the keyboard while clicking the desired features in the Model window. When finished, release the Ctrl or Shift key (as applicable).
     - Drag the mouse to “draw” a box around the desired features in the Model window.
5. After changing the desired settings and selecting the desired feature(s), click **Generate Steps** to measure the selected features using the specified tool, and current magnification and illumination. The software adds the measurement steps to the measurement routine automatically.
6. [Single Monitor User Interface] Click the **Show CAD** icon in the Model window toolbar.

> The CAD model is saved with the measurement routine, so you can create additional steps whenever the routine is open.

This page was left blank intentionally.

19. MeasureFit Analysis Software

19.1 Overview

MeasureFit is an optional, powerful 32-bit software package that enables the analysis of two-dimensional contours. Analysis results show both position and form errors.

MeasureFit is designed to integrate seamlessly with eBx. It automatically analyzes eBx routines using a variety of best-fit algorithms to obtain the best analysis for each routine. You can even include the MeasureFit analysis as a macro in the last step of the eBx routine. However, it is important that you create the eBx routine in a specific way in order to use MeasureFit effectively.

Before you can use MeasureFit, a protection key (dongle) must be installed. Once the MeasureFit software and protection key are installed, you can access MeasureFit by clicking the **MeasureFit** tab above the Image window or by launching the software separately (depends on how the user interface is configured).

Refer to the MeasureFit Fast Start Guide (P/N 790465) and the MeasureFit online Help for information about using the MeasureFit software.

19.2 MeasureFit Preparation

To use MeasureFit effectively, it is important that you follow the guidelines listed below when you create an eBx routine:

- Specify the units shown on the blueprint before you begin creating the routine. The units in the eBx routine and the MeasureFit template must match.
- Enter nominal values and tolerances in each step that will be analyzed in MeasureFit.
  - Be sure to specify nominal values and tolerances for the size, location, and position.
  - All nominal values and tolerances from the eBx routine are available for analysis in MeasureFit.
- Indicate the location of the material in each step that will be evaluated with MeasureFit. This ensures the proper tolerance analysis and graphical display of the whiskers indicating excess material or lack of material.
  - eBx automatically specifies the material condition for you. The software identifies the light and dark sides of the measured feature to determine the location of the material.
  - To indicate the location of the material, click the appropriate Profile button, where the tolerance values correspond with the + sign and - sign on the buttons.
  - For a Line, the + sign should be on the side of the line that has no material. Additional material on this side results in a positive profile deviation from nominal.
  - For an Arc or Circle, the + sign should be on the side of the arc with no material or for a circle the hole. Additional material on this side results in a positive profile deviation from nominal.
- Configure the data stream parameters if you plan to output eBx data to a data stream file and then import the file into MeasureFit for analysis.

19.3 MeasureFit Access (Classic User Interface)

To access and use MeasureFit, click the **MeasureFit** tab. The software displays the MeasureFit Model window, MeasureFit settings, and table tabs, a features list table, and a feature properties results window in the Measurement window area.

Since the MeasureFit analysis can be included as a macro in the last step of the eBx routine, the MeasureFit analysis is typically done after all the steps are measured.

You can also use MeasureFit real time while you're measuring steps in the routine. For example, you can measure a hole, go to MeasureFit and analyze the hole with a soft plug, and then go back to eBx to continue measuring steps.

To go back to eBx, simply click any of the eBx tabs. When you do this, a MeasureFit macro step is displayed in the Measurement window.

- If you click **Cancel**, you can continue adding more measurement Steps to the eBx routine.
- If you click **OK**, this becomes the last step in the routine; you can no longer add more steps to the routine.

This page was left blank intentionally.

20. MeasureMenu Software

20.1 Overview

MeasureMenu is an optional software package that enables you to:

- Organize multiple measurement routines into defined sequences and automatically run them in the selected sequence
- Create and define different user access levels so operators can run only specified routines
- Associate part routines with pictures and text files for easy searching and access

Before organizing eBx routines in MeasureMenu, be sure to follow the MeasureMenu preparation guidelines in eBx.

For more information about installing and using MeasureMenu, see the *MeasureMenu FastStart Guide*, PN 790337.

20.2 MeasureMenu Preparation

To optimize the communication between MeasureMenu and your measurement software, follow the guidelines listed below when you create and save an eBx routine:

- Make sure that the statistics output file name and location are the same as specified in MeasureMenu Options/Stats Input menu.
- eBx must be open without a routine in memory.
- Save the routines in the Routines folder. However, they should be saved in that user's folder if MeasureMenu is run by a specific user.

Be sure that the manual part setup has been done properly before you run a measurement routine in MeasureMenu.

For more information about using MeasureMenu, see the *MeasureMenu FastStart Guide*, PN 790337.

Index

A

B

C

D

E

F

G

H

I

J

K

L

M

N

O

P

R

S

T

U

V

W

X

Z

RAM Optical
Instrumentation
RAM
A Quality Vision
International Company

1175 North Street
Rochester, NY 14621 USA
1.877.764.6397
Fax 1.585.506.4307