PC-DMIS MultiGage

Measurement Tool for the MultiGage Portable Arm

Version 1.0

by

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PC-DMIS MultiGage

Introducing MultiGage

PC-DMIS MultiGage provides an easy-to-use solution for measuring parts. The MultiGage program uses the power of PC-DMIS to accomplish your metrology needs.

This document describes the functionality and methods involved in using the MultiGage program and portable arm. For a general overview of the product, review the MultiGage Tutorial.

The main topics discussed for PC-DMIS MultiGage (version 1.1) are:

- Using the MultiGage Machine
- MultiGage Settings
- Multi Gage Interface
- Dimensioning Features
- Measuring Features
- Constructing Features
- Creating Alignments
- Viewing Measurement Results
- Using Gage Mode
- Program Execution

Using the MultiGage Machine

This chapter describes the general usage of your MultiGage machine:

- Machine Buttons
- Machine Extents
- Mouse Mode
- Changing Probes

Machine Buttons

The three buttons of the MultiGage machine allow you to take hits and erase hits, end tasks, and function as a computer mouse (see "Mouse Mode").

The following functions are available while measuring features:

Measuring Points:

- **Take one hit** - Press and release the ***RED*** machine button. A hit is taken at the current probe location and added to the hit buffer. The hit buffer holds the hits until you press DONE, where they are used to create the intended feature.

- **Take multiple hits** - Press and hold the ***RED*** machine button for more than one second until the desired number of hits have been taken. This allows you to scan the surface of the part as multiple hits are added to the hit buffer. Hits are no longer added to the hit buffer when you release this button.

Erasing Points:

- **Erase a single hit** - Press and release the ***YELLOW*** machine button.
- **Erase multiple hits** - Press and hold the ***YELLOW*** machine button for more than one second until the desired number of hits have been removed.

Finishing a Measurement:

- **End task** - To **Finish** a measurement step or advance to the **Next** step, press the ***WHITE*** machine button. This button functions in this manner once you have taken enough hits to complete the given feature.

Important: This documentation will refer to the button assignments as noted above. Any changes made to button assignments by the user will need to be accounted for in reviewing help topics.

RDS Button Assignments

RDS uses numeric button assignments for the MultiGage machine. These numeric values correspond to the button colors and measurement functions as follows:

- **0** - White - Done
- **1** - Red - Take Hits
- **2** - Yellow - Erase Hits

Machine Extents

The axes of the MultiGage machine allow the flexibility to maneuver your probe around the part to take hits. If any of the axes have reached the extent for that axes (the machine gives an audible tone), you will not be able to take measurements. Just adjust the position of the arm segments so the tone has stopped and you can begin measuring.

Mouse Mode

When the MultiGage program first appears 'Mouse Mode' is enabled. This allows you to make selections using the MultiGage Arm in a similar fashion to using a mouse.

Using the machine you can do the following:

Enable/Disable Mouse Mode: Place **axis 'F'** (the wrist axis) at its limit and turn **axis 'E'** (the axis above the wrist axis) 90 degrees. When 'Mouse Mode' is disabled you can take hits with your probe. When you have successfully enabled 'Mouse Mode' you will receive the message "Mouse Mode Enabled".

Move the Pointer:

- Move **axis 'F'** (the wrist axis) to control *left* and *right* movement.
- Move **axis 'E'** (the axis above the wrist axis) to control *up* and *down* movement.

This works best when you hold the length of Arm that is before the last two axes so that it is parallel to the ground. This may take a little bit of practice to effectively control the mouse pointer.

Mouse Button Assignments: To take mouse clicks at the mouse pointer location.

- **Right-Mouse Click:** Press the *RED* machine button.
- **Left-Mouse Click:** Press the *YELLOW* machine button.
- **Middle-Mouse Click:** Press the *WHITE* machine button.

Automatic 'Mouse Mode' Switching

The MultiGage program can automatically switch in and out of 'Mouse Mode' when performing an alignment, construction or dimension task. To enable Automatic 'Mouse Mode' switching, select the **Automatically toggle mouse mode during measurement** option from the **General** tab of the **MultiGage Settings** window.

Automatic 'Mouse Mode' switching functions as follows:

- 'Mouse Mode' is *disabled* when you initiate a feature measurement step from a construction, alignment or dimension task.
- 'Mouse Mode' is *enabled* when you complete a measurement step that is part of a task and the selection grid is displayed.

NOTE: Automatic switching does not work when you are just measuring a single feature, since you are still in measure mode after measuring a feature. This allows you to continue measuring multiple features of the same type.

Changing Probes

To change MultiGage probes:

1. Hold the MultiGage machine by the wrist.
2. Using the supplied key, rotate the locking mechanism to the ‘open’ position; counter clockwise as you are looking down on it.

3. Remove the probe from the Arm.
4. Align the *red arrow* on the probe to the *red arrow* on the wrist and insert the new probe into the probe adapter.
5. Hold the probe in position and use the supplied key to rotate the locking mechanism to the 'lock' position. The MultiGage program will automatically detect that a new probe has been attached.

Important: Rotation of the lock mechanism past the lock point, can damage the mechanism. DO NOT OVERTIGHTEN.

MultiGage Settings

The **MultiGage Settings** window of the **QuickStart** interface allows you to configure general settings for the MultiGage program. Click **Configure application settings** from the **Utilities** toolbar to open this window.

Once the needed changes have been made, click **OK** to save your changes. Click **Cancel** to close the **Application Settings** without saving your changes.

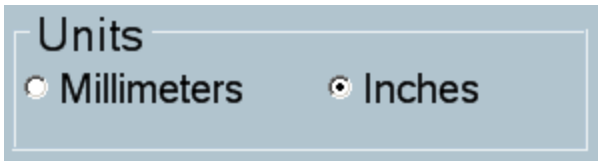
You can configure settings from the following tabs:

- General
- Printing
- Kiosk
- PRG Files

General

The **General** tab of the **MultiGage Settings** window allows you to specify the units and language and toggle automatic mouse mode switching. I also allows you access the RDS configuration.

Units



Specify the units that will be used for displaying measurement results in the **QuickStart** windows, **Status** window, and Report output. Select from either **Millimeters** or **Inches**. The current units (INCH or MM) is displayed in the **Status** bar.

Language

Specify the MultiGage program **Language** by selecting it from drop-down list. Once you press **OK**, MultiGage will be restarted to load the new language.

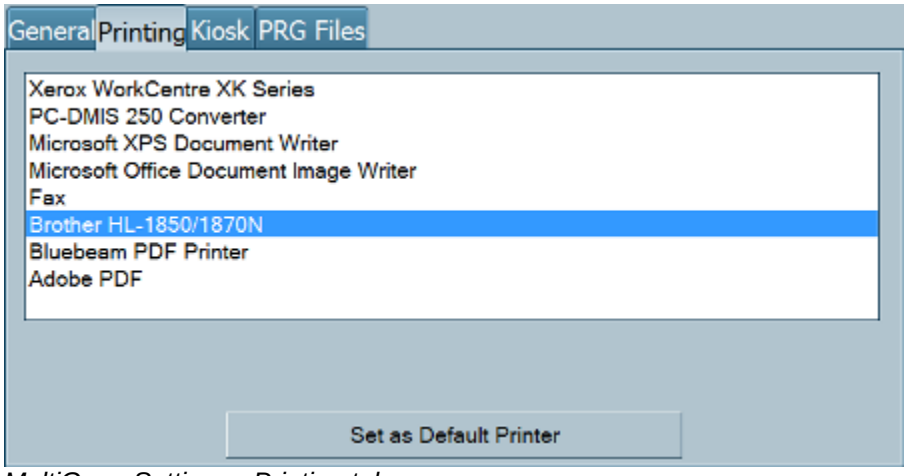
Automatically toggle mouse mode during measurement

When this option is selected, the MultiGage program will automatically toggle in and out of 'Mouse Mode' depending on the current action. See "Mouse Mode".

Open RDS Control Panel

Clicking **Open RDS Control Panel** allows you access the RDS configuration for your MultiGage machine. See the "Appendix B: RDS Control Panel" topic for more information.

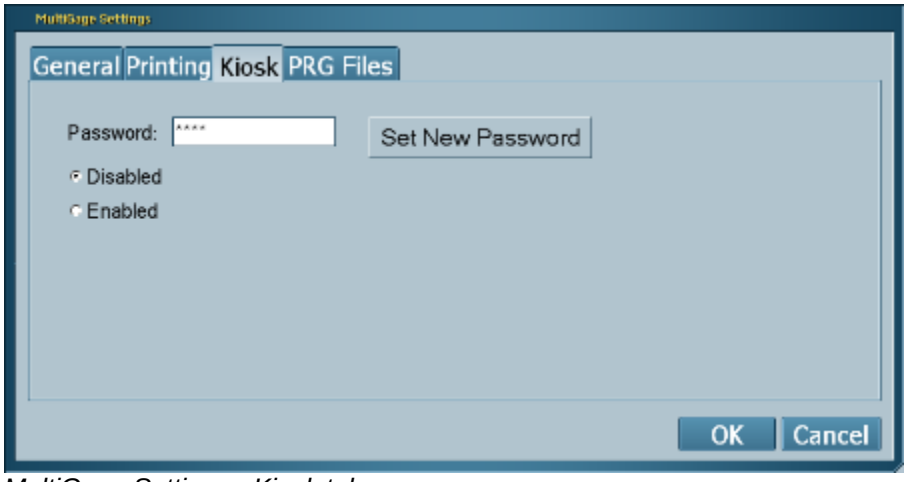
Printing



MultiGage Settings - Printing tab

The **Printing** tab allows you to specify the printer to which reports will be sent when you select **Print Report** from the **Utilities** toolbar. All available printers for your system are displayed. Select the needed printer from the list. Click **Set as Default Printer** to cause the selected printer to become your default printer.

Kiosk

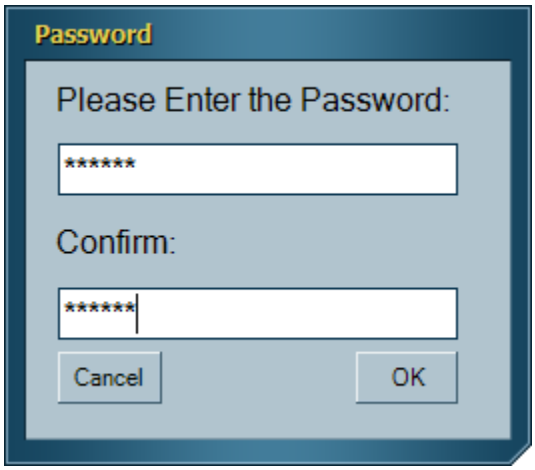


MultiGage Settings - Kiosk tab

'Kiosk Mode' is the default configuration for the MultiGage program. Enabling 'Kiosk Mode' causes the MultiGage program to be launched when your system boots. This limits computer functionality to the MultiGage program only, while all other Window's functionality is disabled. When 'Kiosk Mode' is disabled, Windows will boot normally. You can still launch the MultiGage program from Windows.

Note: Some AntiVirus software may warn that the action of turning this PC into a Kiosk is suspicious. Also, Kiosk mode changes will be realized on the next system reboot.

Enabling or disabling 'Kiosk Mode' can only be done, when the correct **Password** has been provided. To change the password:

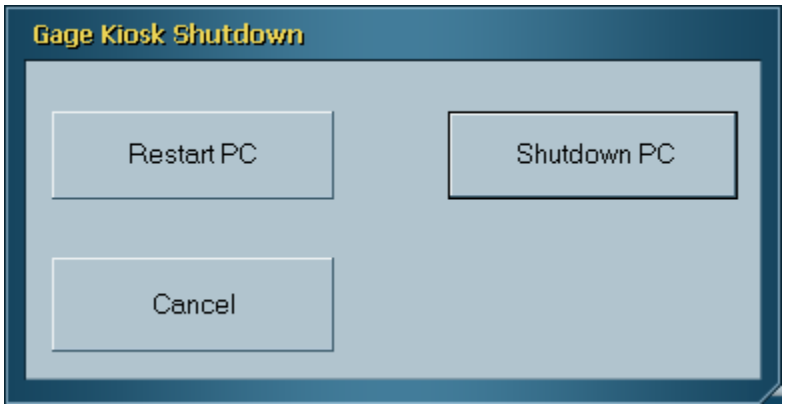


Password dialog box

1. Type in the current **Password**. The **Set New Password** button becomes enabled.
2. Select **Set New Password**. The **Passwor**d dialog box opens.
3. Type the new password in both text boxes and click **OK**. Click **Cancel** to close without saving.

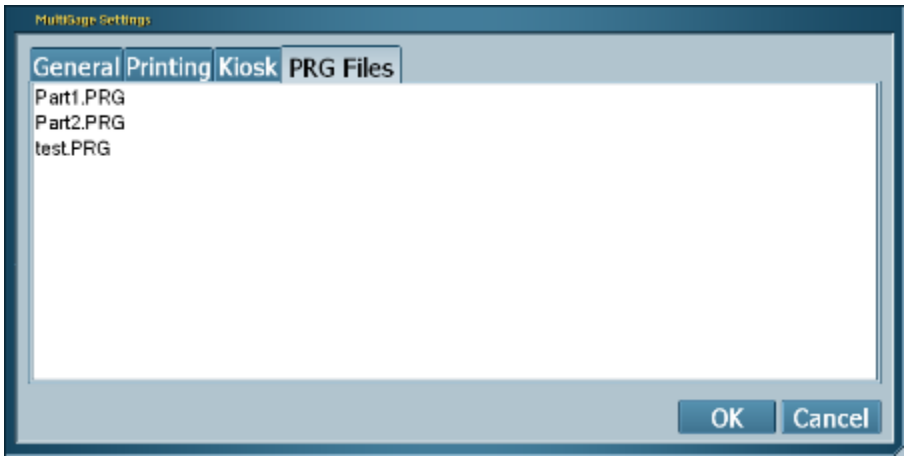
When you enable or disable 'Kiosk Mode' you will be prompted to **Restart** or **Shutdown** your PC after clicking the **OK** button from the **MultiGage Settings** window. If you select **Cancel,** no action

will be taken. In the case where you are disabling 'Kiosk Mode' you will also be prompted to **Exit to Explorer**, which will take you out of 'Kiosk Mode' while launching Windows Explorer.



Gage Kiosk Shutdown prompt

PRG Files



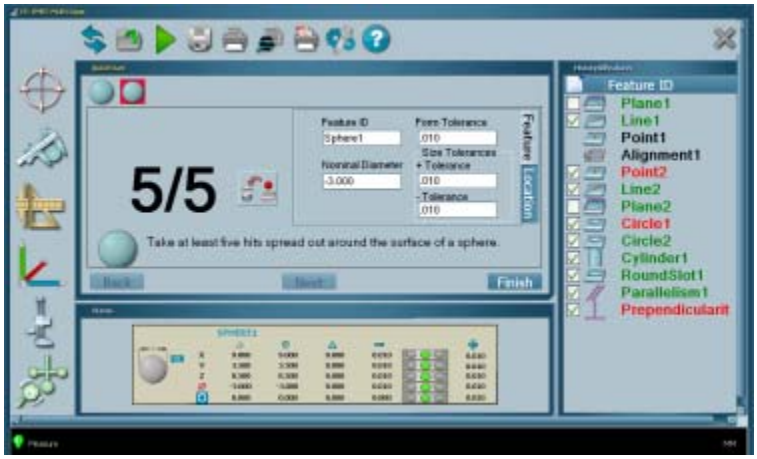
MultiGage Settings - PRG Files tab

The **PRG Files** tab allows you to view the MultiGage part programs that are stored on your system. MultiGage part programs have a PRG extension. By right-clicking on a program file name you can access a popup menu to delete or rename files:

- **Delete** - Select **Delete** to remove the part program file from your system.
- **Rename** - Select **Rename** to highlight the selected part program for editing. Type the new name and press **Enter** or click **OK**.

You can also view part program information by hovering the mouse pointer over the part program name. This will display the **Date Modified** (date and time) and file **Size**.

MultiGage Interface



MultiGage makes use of the following interface elements:

- Main Toolbar
- Utilities Toolbar
- QuickStart Window
- Status Window
- History Window
- Status Bar

Main Toolbar

The **Main Toolbar** allows you to perform the core functions for PC-DMIS MultiGage:

- Dimensioning Features
- Measuring Features
- Constructing Features

- Creating Alignments
- Using Gage Mode
- Calibrating Probes

Utilities Toolbar

From the **Utilities** toolbar you can accomplish the following tasks:

Start New Session will clear all features from the **History** window and **Status** window. MultiGage returns to a state as if you had just opened the program.

If you have already measured features you will be prompted to save your current program. Select **Yes** to open the **Save File** dialog box from which you can save your program, or **No** to discard the current program (all data will be lost). If you are working with a program that you have already saved, the MultiGage program will automatically save any changes.

Open File allows you to select a previously created program from the **Open File** dialog box.

Execute Part will execute the current program in the order it was created. See "Program Execution".

Save File opens the **Save File** dialog box. Type the needed **File Name** and click **Save** to save your current program.

Print Report prints the current report to the specified printer on the Printing tab of the "MultiGage Settings" window.

View Report displays the current report in the Measurement Results window. See "Viewing Measurement Results".

Configure MultiGage Settings allows you to set general parameters that control the way that the MultiGage program operates. See "MultiGage Settings".

MultiGage Help displays the MultiGage help file. Pressing **F1** will display the relative help topic for the current task.

Exit closes the MultiGage program.

QuickStart Window

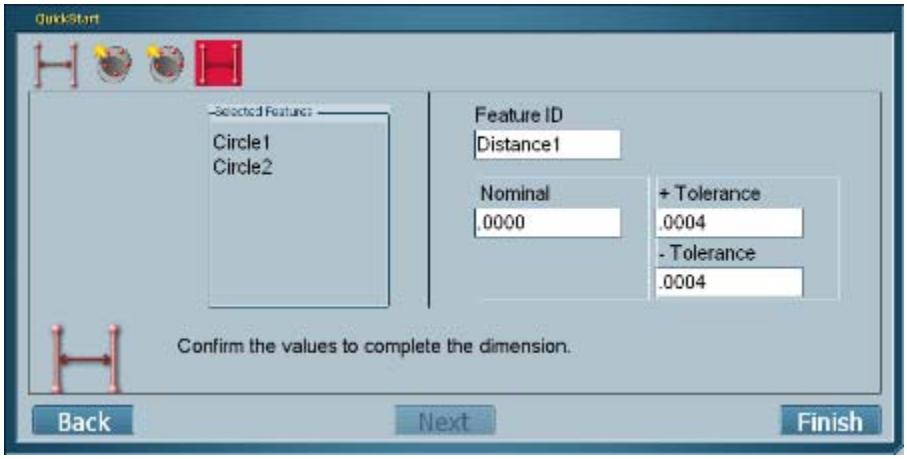
When the MultiGage program first opens, the **QuickStart** window presents you with the option to choose one of the core tasks from the **Main** toolbar. As you choose a task, the **QuickStart** window provides the steps and parameters via the following interfaces:

- Dimension Interface
- Measured Feature Interface
- Constructed Feature Interface
- Alignment Interface

Common controls for the **QuickStart** window include:

- Progress Bar
- QuickStart Buttons

Dimension Interface



QuickStart Window - Distance Dimension

The Dimension interface allows you to select the needed dimension, select or measure input features, and verify the creation of the dimension.

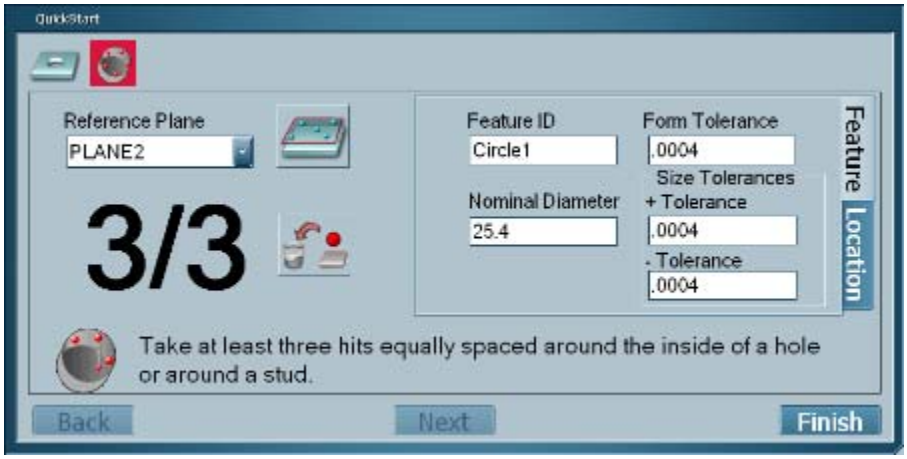
This topic is intended as a reference for the interface elements that are used for dimensions. See "Dimensioning Features" for step by step instructions.

The **QuickStart** interface uses the following user interface elements for the creation of *dimensions*:

Dimension Feature Parameters	
Selected Features	Displays the previously selected features that were provided as input for the dimension.
Feature ID	Provides the ID for the dimension as it will be displayed in the **History** window, **Status** window and **Reports**.
Nominal	Provides the nominal *angle* or *distance* for provided input features.

+ Tolerance	Provides the *plus* tolerance for the dimension.
- Tolerance	Provides the *minus* tolerance for the dimension.
True Position ID	Provides the True Position ID as it will be displayed in the **History** window, **Status** window and **Reports**.
Tolerance	Provides the tolerance for the *true position*, *parallelism* or *perpendicularity* dimensions.
Datum ID	Provides the Feature ID of the feature representing the specified Datum.
Distance X and Y	Provides the X and Y nominal values from the DATUM feature to the feature fro which True Position is being dimensioned.

Measured Feature Interface

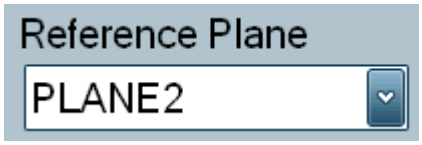


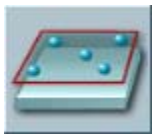
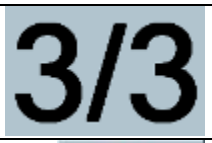
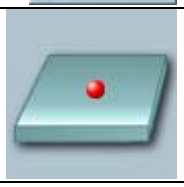
QuickStart Window - Measured Circle Feature

The Measured Feature interface allows you to select the needed feature, measure it, and edit the parameters.

This topic is intended as a reference for the interface elements that are used for measuring features. See "Measuring Features" for step by step instructions.

The **QuickStart** interface uses the following user interface elements for the creation of *measured* features:

Feature Measurement
 **Reference Plane** - Provides a drop-down list that allows you to select an existing reference plane. If you need to measure a reference plane, the new plane is added to this list once it has been measured.

	**New Plane** - Click to insert a plane measurement step into the measurement progress for the current task.
	**Hit Counter** - Displays the current number of hits and the expected number of hits for the given feature. You may take more hits than the expected number of hits.
	Remove Hit - Click to delete the last hit.
	**Current Feature** - Displays the feature type that is currently being measured. Additional instructions are also provided to prompt the user for the current step.

Measured Feature Parameters

These parameters are accessed from the **Feature** tab.

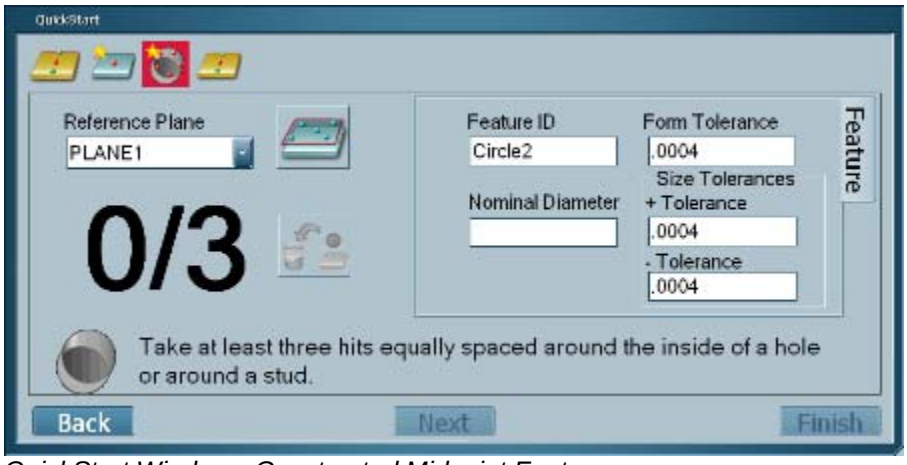
Feature ID	Provides the ID for the measured feature as it will be displayed in the **History** window, **Status** window and **Reports**.
Form Tolerance	Provides the tolerance for a feature's form. Evaluated features whose form exceeds the given value are considered to be out-of-tolerance. This value is used with all measured features except for points.
Nominal Length	Provides the nominal *length* for lines, round slots and square slots.
Nominal Diameter	Provides the nominal *diameter* for circles, cylinders, cones, spheres
Nominal Width	Provides the nominal *width* for round slots and square slots.
+ Tolerance	Provides the *plus* tolerance for feature elements such as length and diameter.
- Tolerance	Provides the *minus* tolerance for feature elements such as length and diameter.

QuickStart Interface - Location Tab

If your program includes alignments, location dimensions will also be reported for the centroid of your feature in relation to the origin of the Alignment axis. These parameters are accessed from the **Location** tab.

Location Parameters	
Nominal	Allows you to type the **X** and **Y** nominal position of the centroid of the feature.
+ Tolerance	Provides the *plus* tolerance for the **X** and **Y** axes by which the location dimension will be evaluated.
- Tolerance	Provides the *minus* tolerance for the **X** and **Y** axes by which the location dimension will be evaluated.

Constructed Feature Interface



QuickStart Window - Constructed Midpoint Feature

The Constructed Feature interface allows you to select the needed constructed feature, select or measure input features, and verify the creation of the constructed feature.

This topic is intended as a reference for the interface elements that are used for constructed features. See "Constructing Features" for step by step instructions.

The **QuickStart** interface uses the following user interface elements for the creation of *constructed* features:

Constructed Feature Parameters	
Feature ID	Provides the ID for the constructed feature as it will be displayed in the **History** window, **Status** window and **Reports**.
Selected Features	Displays the previously selected features that were provided as input for the constructed feature.

Alignment Interface

QuickStart Window - Plane-Line-Point Alignment

The Alignment interface allows you to select the needed alignment, select or measure input features, and verify the creation of the alignment.

This topic is intended as a reference for the interface elements that are used for alignments. See "Creating Alignments" for step by step instructions.

The **QuickStart** interface uses the following user interface elements for the creation of *alignments*:

Constructed Feature Parameters	
Feature ID	Provides the ID for the alignment as it will be displayed in the **History** window, **Status** window and **Reports**.

Progress Bar

The **Progress** bar at the top of the **QuickStart** interface provides information showing the steps that are needed to complete the current task. Tasks might include creating a dimension with included measured features or as simple as measuring a circle feature. The step highlighted in red indicates the current step of the current task.

Elements of the progress bar include:

- **Left-most and right-most icons:** Show the current task
- **Icons between the end icons:** Indicate the steps needed to complete the task.
- **Icon highlighted in red:** Indicates the current step for the task.
- **Icon indicated with star:** Indicates that the feature was measured when creating the dimension, alignment or constructed feature.

These steps will change dynamically as needed. For example, if you have already measured a circle with the associated plane, subsequent circles would not require you to measure a new reference plane. In this case, only a circle would be displayed in the **Progress** bar.

If you click **New Plane** from the circle interface, the plane step would be inserted before the circle step.

Repeating Task Mode

By default, the MultiGage program will repeat measured feature tasks as each feature is finished. The **Repeating Task Mode** toggle button is displayed to the right of the progress bar in the QuickStart interface when this option is available.

This allows you to measure a series of the same measured feature type without having to select the feature from the MultiGage interface.

Repeat On - The QuickStart interface will repeat measurement of the selected measured feature.

Repeat Off - The QuickStart interface will return to the QuickStart start screen when the measured feature task is finished.

Note: Repeating measurements is not used for measured features that are being measured as part of a dimension, alignment or constructed feature task.

QuickStart Buttons

The buttons at the bottom of the **QuickStart** window are common to each of the different feature types. They provide the following functionality:

Back	Returns to the previous step of the current task.
Next	Proceeds to the next step of the current task. **Next** is only available when you have taken the minimum number of hits or the needed features have been selected.
Finish	Completes the current task and saves the measured and nominal data. New features are added to the **History** window.

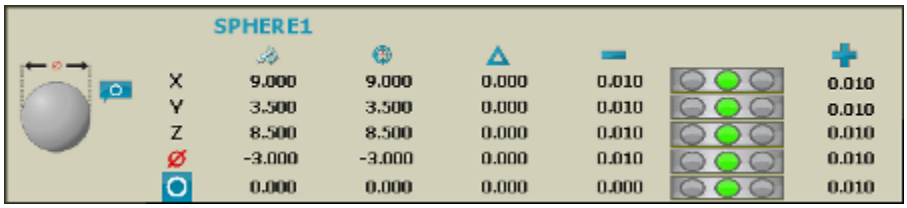
Note: When either the **Next** or **Finish** buttons are available, you can use the ***YELLOW*** machine button to complete step or task. See "Machine Buttons".

Status Window

The **Status** window displays the results of the current **QuickStart** feature. This allows you to view real-time results as you measure. The feature label provides a preview of the data that will be

included with the report. Selecting a feature from the **History** window will also allow you to review feature results.

Note: Millimeter values are displayed with an accuracy of 3 decimal places while Inch values are displayed with an accuracy of 4 decimal places.



Status Window - Sphere Example

Understanding Status Window Results

Labels created in the status window use icons to depict each of the reported values. Common to each label are the Feature ID and an icon representing the feature type.

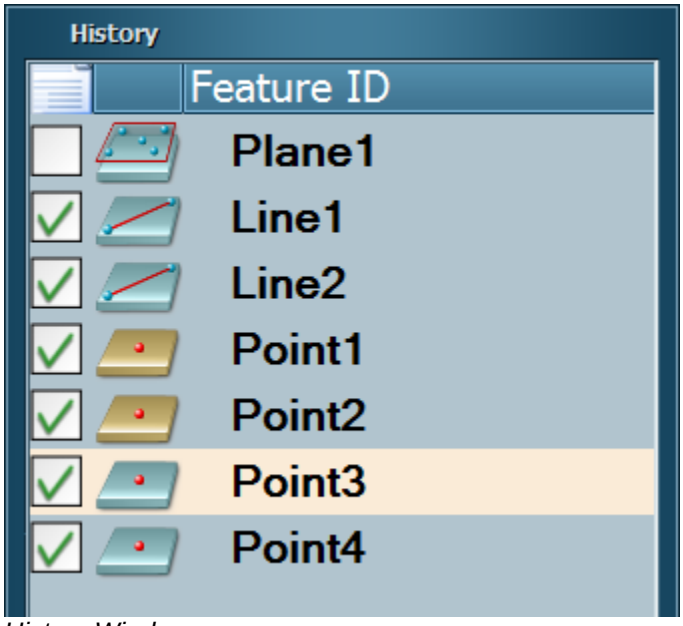
The following icons represent different feature values:

	Measured		**Diameter**
	Nominal/ Theoretical		**Length - Slots**
	Deviation		**Width -** Slots Length or Distance Dimension
	Negative Tolerance		**Cone Angle**
	Positive Tolerance		**Form** - Circularity
	Out of Tolerance - Negative		**Form** - Straightness
	Out of Tolerance - Positive		**Form** - Flatness
	In Tolerance		**Form** - Cylindricity
X	**X Distance -** Location Dimension		**Form** - Angularity
Y	**Y Distance -** Location Dimension		**Parallelism**
Z	**Z Distance -** Location		**Perpendicularity**

	Dimension		
			Concentricity

History Window

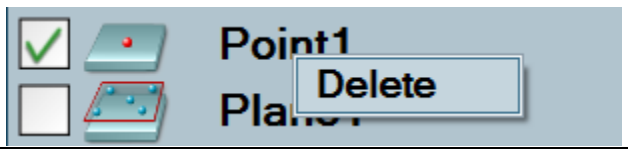
The **History** window lists all previously measured features, constructed features, dimensions and alignments. It is also provides the **Input Feature List** for selecting input features for feature construction, dimension and alignment creation.



History Window

The **History Window** provides the following functionality:

Editing Feature Parameters	Select the needed feature from the **History** window by clicking on the feature icon or feature name. The parameters for the selected feature are displayed, allowing you to make any needed changes. When you click **Finish**, the **History** window will reflect the change and **Status** window is updated with the results of the modified feature.
Including Features in the Report	Click the check box next to the feature icon in the **History** window, under the Report icon. All checked features are included in the report while unchecked features are excluded from the report.
Deleting Features	Right-click on the feature to be removed from the **History** window and click **Delete**.

	
Selecting Input Features to Create Other Features	The **History** window becomes the **Input Feature List** when selecting input features for Constructed Features, Dimensions and Alignments. This allows you to select previously created features as input. See Dimensioning Features, Constructing Features, and Creating Alignments.

Status Bar

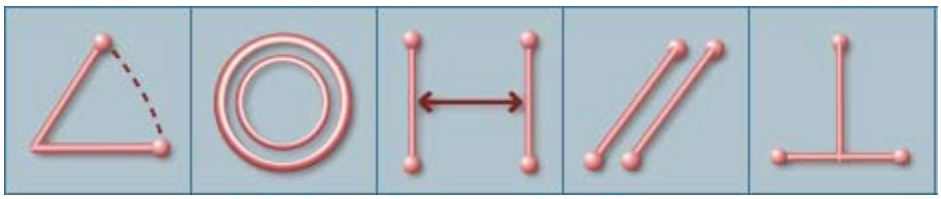
The **Status** bar provides general status information from your MultiGage program as well as status and error messages from the RDS service. The following information is provided:

- Connection Status with PC-DMIS:

Connected	Not Connected

- Mouse-over tool tips display the name of the command at the mouse pointer location.
- Other MultiGage program status messages, such as initializing PC-DMIS, Ready, RDS status or error messages, etc.
- Current units for measuring your part: **MM** or **INCH**

Dimensioning Features



Most MultiGage features have attribute dimensions that are automatically included in the report. Once a feature is measured, the associated dimension(s) are evaluated according to the provided nominal and tolerance values.

Additional dimensions can be reported by selecting the needed dimension with the features to be evaluated. Dimensions that can be added to your report that are not already inherently added include:

- Angle
- Concentricity (inherently included with cones)
- Distance
- True Position
- Parallelism
- Perpendicularity

Here is a list of *form* dimensions that are inherently included in the report:

- Straightness (Line)
- Flatness (Plane)
- Roundness (Circle or Sphere)
- Cylindricity (Cylinder)
- Concentricity (Cone)

Here is a list of *size* dimensions that are inherently included in the report:

- Diameter (Circle, Cylinder and Sphere)
- Cone Angle (Cone)
- Width and Length (Round Slot and Square Slot)

Location Dimension

- Location - When a program includes alignments, Location dimensions are also reported for each subsequent feature.

Input Characteristics

The MultiGage program uses the *point*, *plane*, *line* or *circle* characteristics of features as input for dimension creation. The input characteristics are derived as follows:

Points	• From Points. • From the *centroid* of Circles, Spheres, Square Slots or Round Slots.
Planes	• From Planes.
Lines	• From Lines. • From the *axis* of a Cylinder or Cone. • From the *long axis* of a Round Slot or Square Slot.
Circles	• From Circles. • From the *circular form* of a Sphere, Cylinder, or Cone.

Angle

The Angle dimension is used to measure the angle relationship between the characteristics of two features: *plane to plane*, *line to line*, or *plane to line*

See "Dimensioning Features" for more information on input characteristics.

To create an Angle dimension:

1. Click the **Dimension** button from the **Main** toolbar.

2. Select the **Angle Dimension** button from the **QuickStart** window. The **QuickStart** feature selection grid appears and the progress bar displays question marks for the needed input features.

The highlighted icon indicates the current step of the angle dimension task.

3. Using one of the following methods, select a *plane, line* or a feature with a *line characteristic*:

Input from Selection Grid

Click the needed feature button from the selection grid. See the related topic for information on measuring the selected feature:

- Measured Plane
- Measured Line
- Measured Cylinder
- Measured Cone
- Measured Round Slot
- Measured Square Slot

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed feature from the **Input Feature List**. Select a *plane*, *line*, *cylinder*, *cone*, *round slot* or *square slot*.

Note: Only possible input features will be displayed in the **Input Feature List** for selection.

4. Repeat step 3 for the second *plane* or feature with a *line characteristic*.

5. Verify the angle dimension and click **Finish** or press the ***YELLOW*** machine button. The new angle dimension is added to the **History**

window.

6. Select the check box next to the new angle dimension in the **History** window to toggle the display of the dimension in the report.

Concentricity

The Concentricity dimension is used to calculate the concentricity relationship of two features with a circular characteristic: *circles*, *cylinders*, *cones*, or *spheres*.

See "Dimensioning Features" for more information on input characteristics.

See "Concentricity for a Cone Feature" below.

To create an Concentricity dimension:

1. Click the **Dimension** button from the **Main** toolbar.

2. Select the **Concentricity Dimension** button from the **QuickStart** window. The **QuickStart** feature selection grid appears and the progress bar displays question marks for the needed input features.

The highlighted icon indicates the current step of the angle dimension task.

3. Using one of the following methods, select a *circle* or a feature with a *circle characteristic*:

Input from Selection Grid

Click the needed feature button from the selection grid. See the related topic for information on measuring the selected feature:

- Measured Circle
- Measured Cylinder
- Measured Cone
- Measured Sphere

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed feature from the **Input Feature List**. Select a *circle*, *cylinder*, *cone*, or *sphere*.

Note: Only possible input features will be displayed for selection.

4. Repeat step 3 for the second *circle* or feature with a *circle characteristic*.

5. Verify the concentricity dimension and click **Finish** or press the ***YELLOW*** machine button. The new concentricity dimension is added to the **History** window.

6. Select the check box next to the new concentricity dimension in the **History** window to toggle the display of the dimension in the report.

Concentricity for a Cone Feature

Concentricity of a cone is evaluated by default based upon the **Form Tolerance** value provided for that cone.

This dimension type is considered to be one sided, meaning the positive **Form Tolerance** value is applied. This means that any concentricity that differs from your nominal or theoretical concentricity can still be a valid measurement as long as it falls within the tolerance range specified.

Distance

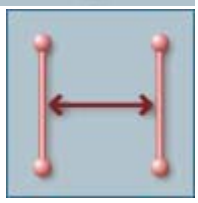
The Distance dimension is used to determine the distance relationship between two features: *point to point*, *point to line*, or *point to plane*.

See "Dimensioning Features" for more information on input characteristics.

Note: The feature with a *point characteristic* is added as an input feature first.

To create an Distance dimension:

1. Click the **Dimension** button from the **Main** toolbar.



2. Select the **Distance Dimension** button from the **QuickStart** window. The **QuickStart** feature selection grid appears and the progress bar displays question marks for the needed input features.

The highlighted icon indicates the current step of the constructed feature task.

3. Using one of the following methods, select an input feature where the centroid represents the *point characteristic*:

Input from Selection Grid

Click the needed feature button from the selection grid. See the related topic for information on measuring the selected feature:

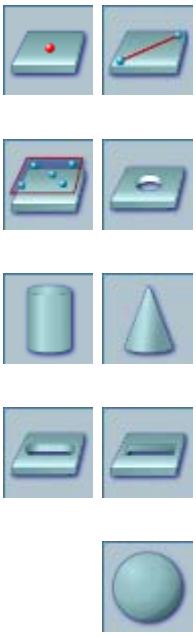
- Measured Point
- Measured Circle
- Measured Sphere
- Measured Round Slot
- Measured Square Slot

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed feature from the **Input Feature List**. Select a *point*, *circle*, *sphere*, *round slot* or *square slot*.

Note: Only possible input features will be displayed in the **Input Feature List** for selection.

4. Select the second input feature with a *point, line* or *plane characteristic* by using one of the following methods:



Input from Selection Grid

Click the needed feature button from the selection grid. See the related topic for information on measuring the selected feature:

- Measured Point
- Measured Line
- Measured Plane
- Measured Circle
- Measured Cylinder
- Measured Cone
- Measured Sphere
- Measured Round Slot
- Measured Square Slot

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed feature from the **Input Feature List**. Select a *point*, *line*, *plane*, *circle*, *cylinder*, *cone*, *round slot*, *square slot* or *sphere*.

Note: Only possible input features will be displayed for selection.

5. Verify the distance dimension and click **Finish** or press the ***YELLOW*** machine button. The new distance dimension is added to the **History** window.

6. Select the check box next to the new distance dimension in the **History** window to toggle the display of the dimension in the report.

True Position

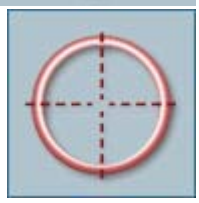
The True Position dimension is used to determine the location of a feature in relation to one or two datum features. The following guidelines are used when measuring True Position:

- Limited to measuring true position for 2D features: *lines*, *circles*, *round slots* and *square slots*.
- The reference plane of the considered feature is used as the *primary datum*.
- The *secondary* and *tertiary datums* must be features on the same plane as the *primary datum*.

- A *secondary datum* is required, while a *tertiary datum* is optional.
- Material conditions of the part *are not* applied.
- Tolerance and nominal distance values from the feature in question to the datum features must be provided.
- The same feature cannot be used for two separate datums.

To create an True Position dimension:

1. Click the **Dimension** button from the **Main** toolbar.



2. Select the **True Position Dimension** button from the **QuickStart** window. The **QuickStart** feature selection grid appears and the progress bar displays question marks for the needed input features.

The highlighted icon indicates the current step of the constructed feature task.

3. Using one of the following methods, select either a *line*, *circle*, *round slot* or *square slot*.

Input from Selection Grid

Click the needed feature button from the selection grid. See the related topic for information on measuring the selected feature:

- Measured Line
- Measured Circle
- Measured Round Slot
- Measured Square Slot

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed feature from the **Input Feature List**. Select a *line*, *circle*, *round slot*, or *square slot*.

Note: Only possible input features will be displayed for selection.

4. Using one of the following methods, select DATUM B which can be a *line*, *plane*, *circle*,

round slot or *square slot*:

Input from Selection Grid

Click the needed feature button from the selection grid. See the related topic for information on measuring the selected feature:

- Measured Line
- Measured Plane
- Measured Circle
- Measured Round Slot
- Measured Square Slot

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed feature from the **Input Feature List**. Select a *line*, *plane*, *circle*, *round slot* or *square slot.*

Note: Only possible input features will be displayed for selection.

5. *This step is **optional** and the possible input features will vary.* Repeat step 4 for DATUM C which could be a *line*, *plane*, *circle*, *round slot* or *square slot*:

6. Provide the **Tolerance** value by which the **X** and **Y** distance values are evaluated.

7. Provide the **Distance X** and **Distance Y** values for each datum. These distances provide the known nominal values.

8. Verify the true position dimension and click **Finish** or press the ***YELLOW*** machine button. The new true position dimension is added to the **History** window.

9. Select the check box next to the new distance dimension in the **History** window to toggle the display of the dimension in the report.

Parallelism

The Parallelism dimension is used to determine how closely two features are parallel to each other. The orientation of the following features can be evaluated in relation to each other: *line,*

plane, cylinder (axis) or *cone (axis)*. As noted, the axis of a cylinder or cone is used for the evaluation of those feature types.

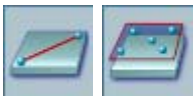
To create an Parallelism dimension:

1. Click the **Dimension** button from the **Main** toolbar.

2. Select the **Parallelism Dimension** button from the **QuickStart** window. The **QuickStart** feature selection grid appears and the progress bar displays question marks for the needed input features.

The highlighted icon indicates the current step of the constructed feature task.

3. Using one of the following methods, select either a *line*, *plane*, *cylinder* or *cone*.



Input from Selection Grid

Click the needed feature button from the selection grid. See the related topic for information on measuring the selected feature:

- Measured Line
- Measured Plane
- Measured Cylinder
- Measured Cone

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed feature from the **Input Feature List**. Select a *line*, *plane*, *cylinder*, or *cone*.

Note: Only possible input features will be displayed for selection.

4. Repeat step 3 for the second *line*, *plane*, *cylinder* of *cone* feature.

5. Verify the parallelism dimension and click **Finish** or press the ***YELLOW*** machine button. The new parallelism dimension is added to the **History**

window.

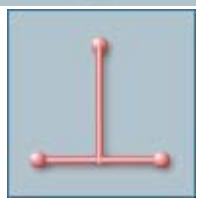
6. Select the check box next to the new parallelism dimension in the **History** window to toggle the display of the dimension in the report.

Perpendicularity

The Perpendicularity dimension is used to determine how closely two features are perpendicular to each other. The orientation of the following features can be evaluated in relation to each other: *line, plane, cylinder (axis)* or *cone (axis)*. As noted, the axis of a cylinder or cone is used for the evaluation of those feature types.

To create an Perpendicularity dimension:

1. Click the **Dimension** button from the **Main** toolbar.



2. Select the **Perpendicularity Dimension** button from the **QuickStart** window. The **QuickStart** feature selection grid appears and the progress bar displays question marks for the needed input features.

The highlighted icon indicates the current step of the constructed feature task.

3. Using one of the following methods, select either a *line*, *plane*, *cylinder* or *cone*.

Input from Selection Grid

Click the needed feature button from the selection grid. See the related topic for information on measuring the selected feature:

- Measured Line
- Measured Plane
- Measured Cylinder
- Measured Cone

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed feature from the **Input Feature List**. Select a *line*, *plane*, *cylinder*, or *cone*.

Note: Only possible input features will be displayed for selection.

4. Repeat step 3 for the second *line*, *plane*, *cylinder* of *cone* feature.

5. Verify the perpendicularity dimension and click **Finish** or press the ***YELLOW*** machine button. The new perpendicularity dimension is added to the **History** window.

6. Select the check box next to the new perpendicularity dimension in the **History** window to toggle the display of the dimension in the report.

Straightness

Straightness of a line is evaluated by default based upon the **Form Tolerance** value provided for that line. A minimum of *three* hits are necessary to determine the straightness of a line.

This dimension type is considered to be one sided, meaning the positive **Form Tolerance** value is applied. This means that any straightness that differs from your nominal or theoretical straightness can still be a valid measurement as long as it falls within the tolerance range specified.

Flatness

Flatness of a plane is evaluated by default based upon the **Form Tolerance** value provided for that plane. A minimum of *four* hits are necessary to determine the flatness of a plane.

This dimension type is considered to be one sided, meaning the positive **Form Tolerance** value is applied. This means that any flatness that differs from your nominal or theoretical flatness can still be a valid measurement as long as it falls within the tolerance range specified.

Roundness

Roundness of a circle or sphere is evaluated by default based upon the **Form Tolerance** value provided for that circle or sphere. A minimum of *four* hits are necessary to determine the roundness of a circle, where a minimum of *six* hits are necessary to determine the roundness of a sphere. Additional hits provide a better representation of the roundness of the entire feature.

This dimension type is considered to be one sided, meaning the positive **Form Tolerance** value is applied. This means that any roundness that differs from your nominal or theoretical roundness can still be a valid measurement as long as it falls within the tolerance range specified.

Cylindricity

Cylindricity of a cylinder is evaluated by default based upon the **Form Tolerance** value provided for that cylinder.

This dimension type is considered to be one sided, meaning the positive **Form Tolerance** value is applied. This means that any cylindricity that differs from your nominal or theoretical cylindricity can still be a valid measurement as long as it falls within the tolerance range specified.

Diameter

Diameter of a circle, cylinder or sphere is evaluated by default based upon the specified *plus* and *minus* **Size Tolerances**. This size dimension type evaluates the measured diameter against the nominal diameter.

Cone Angle

The Cone Angle of the cone vertex is evaluated by default based upon the specified *plus* and *minus* **Size Tolerances**. This size dimension type evaluates the measured vertex angle against the nominal vertex angle.

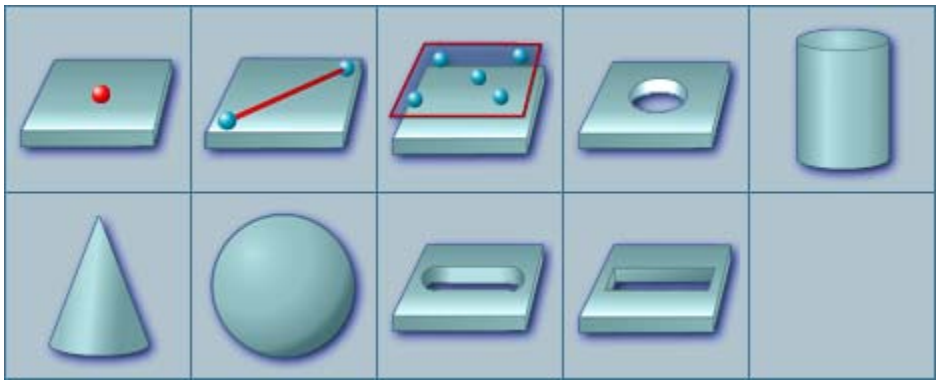
Width and Length

Width and Length of a round slot or square slot are evaluated by default based upon the specified *plus* and *minus* **Size Tolerances**. These size dimension types evaluate the measured width and length against the nominal width and length.

Location

The Location dimension is used to determine the location of the centroid of a feature in relation to a part alignment. Location is reported for all measured features when an alignment exists in the program prior to the feature being added. See "Creating Alignments".

Measuring Features



MultiGage supports measurement of the following features:

- Measured Point

- Measured Line
- Measured Plane
- Measured Circle
- Measured Cylinder
- Measured Cone
- Measured Sphere
- Measured Round Slot
- Measured Square Slot

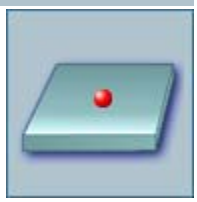
For 2D features requiring a reference plane, see "Measuring a Reference Plane".

See also - "Repeating Measurements".

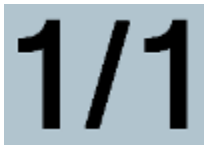
Measured Point

To measure a point:

1. Click the **Measure** button from the **Main** toolbar.



2. Select the **Measure Point** button from the **QuickStart** window. The **QuickStart** point interface appears.



3. Using the ***RED*** button of your MultiGage Arm, take one hit to define the point feature. As the hits are received the MultiGage program updates the number of hits. Nominal and dimension results for the point are also updated in the **Status** window.

Note: If you are not satisfied with a hit, you can remove the last hit by clicking the **Remove Hit** button or by pressing the ***WHITE*** machine button.

4. Edit the **Feature ID** if needed. See "Editing Point Parameters" below.

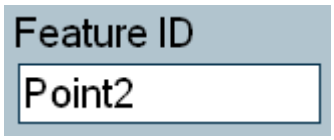
5. Click **Finish** or press the ***YELLOW*** machine button. The new point feature is added to the **History** window.

6. Select the check box next to the new measured point in the **History** window to toggle the display of the point in the report.

Editing Point Parameters

To edit point parameters:

1. Click the needed point icon from the **History** window. The **Feature ID** for your point is displayed.



2. Modify the **Feature ID** as needed. MultiGage automatically adds a feature ID of *Point* plus an incremented number value (e.g. Point3). You can change this value if needed.
3. When using alignments you may modify the **Nominal** and **+/- Tolerance** values on the **Location** tab to control how the Location dimension is reported.

3. Click

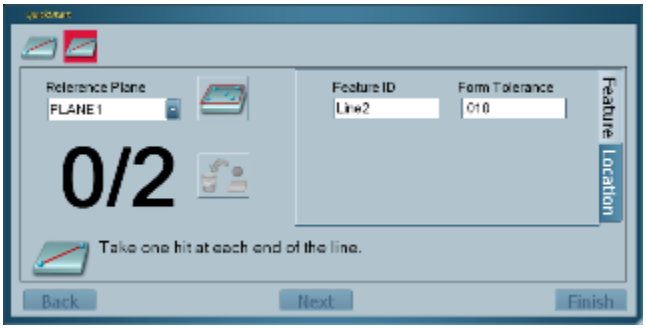
to save your changes.

Measured Line

To measure a line:

1. Click the **Measure** button from the **Main** toolbar.

2. Select the **Measure Line** button from the **QuickStart** window. The **QuickStart** line interface appears. If no **Reference Plane** exists at this point, you will need to measure it. See "Measuring a Reference Plane".



3. Select the line's **Reference Plane** from the drop-down list, or click "New Plane" button to create a new reference plane.

4. Using the ***RED*** machine button, take one hit at each end of the line feature. As the hits are received the MultiGage program updates the number of hits. When the minimum number of hits have been received, the nominal and dimension results for the line are updated in the **Status** window.

Note: If you are not satisfied with a hit, you can remove the last hit by clicking the **Remove Hit** button or by pressing the ***WHITE*** machine button.

Note: You may take more hits than the specified minimum. For example 8/2 would display if you took 8 hits where you only need 2. This allows you to further define the line's geometry. When dimensioning straightness of a line you must take at least 3 hits. It is recommended to take many hits to accurately define the line's form.

5. Edit the line's parameters if needed. See "Editing Line Parameters" below.

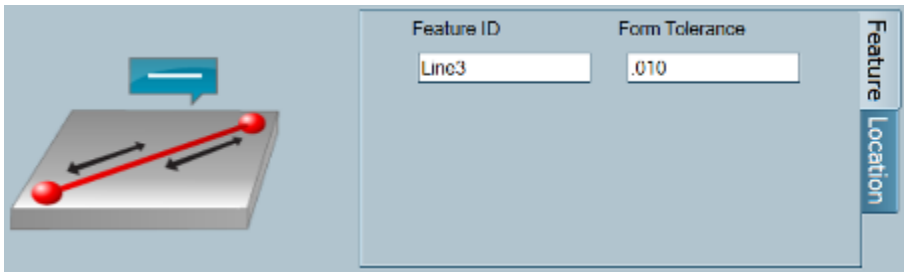
6. Click **Finish** or press the ***YELLOW*** machine button. The new line feature is added to the **History** window. The **Feature ID** text in the **History** window is colored to indicate whether the feature is *in tolerance* (**GREEN**) or *out of tolerance* (**RED**).

7. Select the check box next to the new measured line in the **History** window to toggle the display of the line in the report. A measured line includes evaluation of the Straightness dimension. A line may also be evaluated in terms of True Position.

Editing Line Parameters

To edit line parameters:

1. Click the needed line icon from the **History** window. The parameters for your line are displayed.



2. Modify the line's parameters as needed:
   - **Feature ID:** The MultiGage program automatically adds a feature ID of *Line* plus an incremented number value (e.g. Line4). You can change this value if needed.
   - **Form Tolerance:** Provide the tolerance value for evaluating the line's form in terms of the Straightness dimension.

Note: Features excluded from the report do not require you to define nominal and tolerance values.

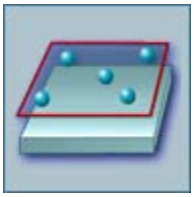
3. When using alignments you may modify the **Nominal** and **+/- Tolerance** values on the **Location** tab to control how the Location dimension is reported.
4. Click

to save your changes.

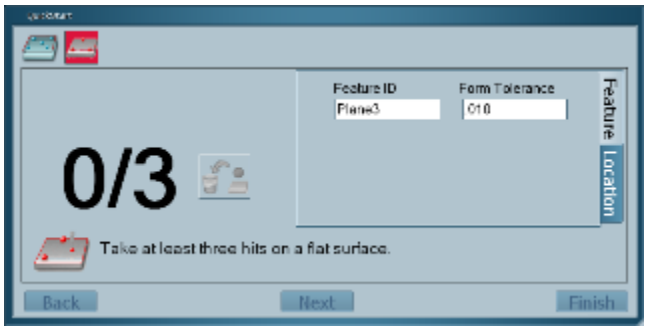
Measured Plane

To measure a plane:

1. Click the **Measure** button from the **Main** toolbar.



2. Select the **Measure Plane** button from the **QuickStart** window. The **QuickStart** plane interface appears.



3. Using the ***RED*** machine button, take at least three hits to define the plane feature. As the hits are received the MultiGage program updates the number of hits. When the minimum number of hits have been received, the nominal and dimension results for the plane are updated in the **Status** window.

Note: If you are not satisfied with a hit, you can remove the last hit by clicking the **Remove Hit** button or by pressing the ***WHITE*** machine button.

Note: You may take more hits than the specified minimum. For example 6/3 would display if you took 6 hits where you only need 3. This allows you to further define the plane's geometry. When dimensioning flatness of a plane you must take at least 4 hits. It is recommended to take many hits to accurately define the plane's form.

4. Edit the plane's parameters if needed. See "Editing Plane Parameters" below.

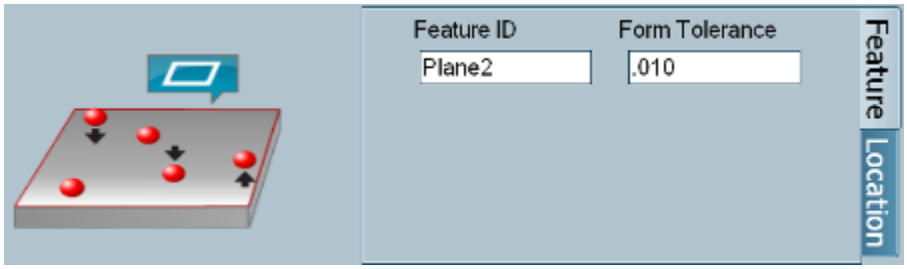
5. Click **Finish** or press the ***YELLOW*** machine button. The new plane feature is added to the **History** window. The **Feature ID** text in the **History** window is colored to indicate whether the feature is *in tolerance* (**GREEN**) or *out of tolerance* (**RED**).

6. Select the check box next to the new measured plane in the **History** window to toggle the display of the plane in the report. A measured plane includes evaluation of the Flatness dimension.

Editing Plane Parameters

To edit plane parameters:

1. Click the needed plane icon from the **History** window. The parameters for your plane are displayed.



2. Modify the plane's parameters as needed:
   - **Feature ID:** The MultiGage program automatically adds a feature ID of *Plane* plus an incremented number value (e.g. Plane2). You can change this value if needed.
   - **Form Tolerance:** Provide the tolerance value for evaluating the plane's form.

Note: Features excluded from the report do not require you to define nominal and tolerance values.

3. When using alignments you may modify the **Nominal** and **+/- Tolerance** values on the **Location** tab to control how the Location dimension is reported.
4. Click

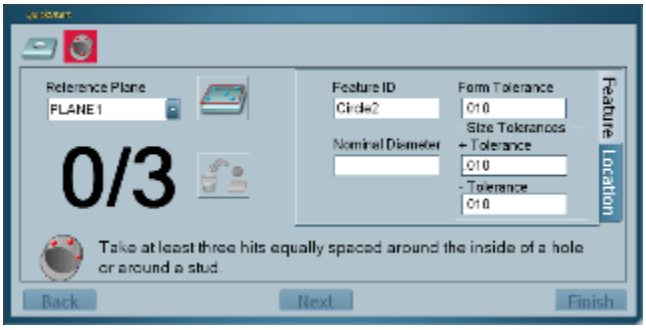
to save your changes.

Measured Circle

To measure a circle:

1. Click the **Measure** button from the **Main** toolbar.

2. Select the **Measure Circle** button from the **QuickStart** window. The **QuickStart** circle interface appears. If no **Reference Plane** exists at this point, you will need to measure it. See "Measuring a Reference Plane".



3. Select the circle's **Reference Plane** from the drop-down list, or click "New Plane" button to create a new reference plane.

4. Using the ***RED*** machine button, take at least three hits to define the circle feature. Hits should be taken so they are uniformly distributed around the circumference of the circle. As the hits are received the MultiGage program updates the number of hits. When the minimum number of hits have been received, the nominal and dimension results for the circle are updated in the **Status** window.

Note: If you are not satisfied with a hit, you can remove the last hit by clicking the **Remove Hit** button or by pressing the ***WHITE*** machine button.

Note: You may take more hits than the specified minimum. For example 5/3 would display if you took 5 hits where you only need 3. This allows you to further define the circle's geometry. When dimensioning roundness of a circle you must take at least 4 hits. It is recommended to take many hits to accurately define the circle's form.

5. Edit the circle's parameters if needed. See "Editing Circle Parameters" below.

6. Click **Finish** or press the ***YELLOW*** machine button. The new circle feature is added to the **History** window. The **Feature ID** text in the **History** window is colored to indicate whether

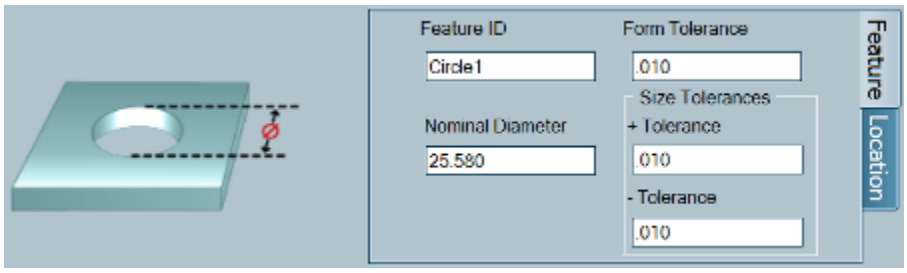
the feature is *in tolerance* (**GREEN**) or *out of tolerance* (**RED**).

7. Select the check box next to the new measured circle in the **History** window to toggle the display of the circle in the report. A measured circle includes evaluation of the Roundness and Diameter dimensions. A circle may also be evaluated in terms of True Position.

Editing Circle Parameters

To edit circle parameters:

1. Click the needed circle icon from the **History** window. The parameters for your circle are displayed.



2. Modify the circle's parameters as needed:
   - **Feature ID:** The MultiGage program automatically adds a feature ID of *Circle* plus an incremented number value (e.g. Circle4). You can change this value if needed.
   - **Nominal Diameter:** Define the nominal diameter value for the circle.

   Note: MultiGage guesses nominal values by rounding the measured value.

   - **Form Tolerance:** Provide the tolerance value for evaluating the circle's form.
   - **+ Tolerance** and - **Tolerance:** Provide the tolerance values for evaluating the circle's diameter.

   Note: Features excluded from the report do not require you to define nominal and tolerance values.

3. When using alignments you may modify the **Nominal** and **+/- Tolerance** values on the **Location** tab to control how the Location dimension is reported.
4. Click

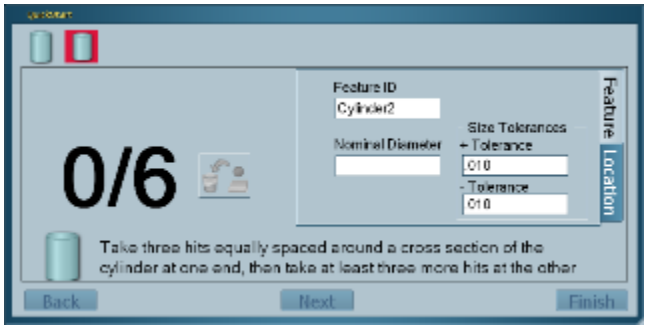
to save your changes.

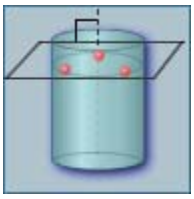
Measured Cylinder

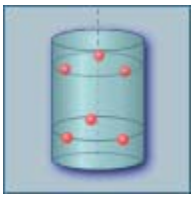
To measure a cylinder:

1. Click the **Measure** button from the **Main** toolbar.

2. Select the **Measure Cylinder** button from the **QuickStart** window.







3. Using the ***RED*** machine button, take the first three hits at one end of the cylinder around a cross section that is perpendicular to the cylinder's axis. This defines the cylinder's orientation. Take at least three more hits at the other end of the cylinder. Hits should be taken so they are uniformly distributed around the circumference of the cylinder. As the hits are received, the MultiGage program updates the number of hits. When the minimum number of hits have been received, the nominal and dimension results for the cylinder are updated in the **Status** window.

Note: If you are not satisfied with a hit, you can remove the last hit by clicking the **Remove Hit** button or by pressing the ***WHITE*** machine button.

Note: You may take more hits than the specified minimum. For example 8/6 would display if you took 8 hits where you only need 6. This allows you to further define the cylinder's geometry. When dimensioning cylindricity of a cylinder, it is recommended to take many hits to accurately define the cylinder's form.

4. Edit the cylinder's parameters if needed. See "Editing Cylinder Parameters" below.

5. Click **Finish** or press the ***YELLOW*** machine button. The new cylinder feature is added to the **History**

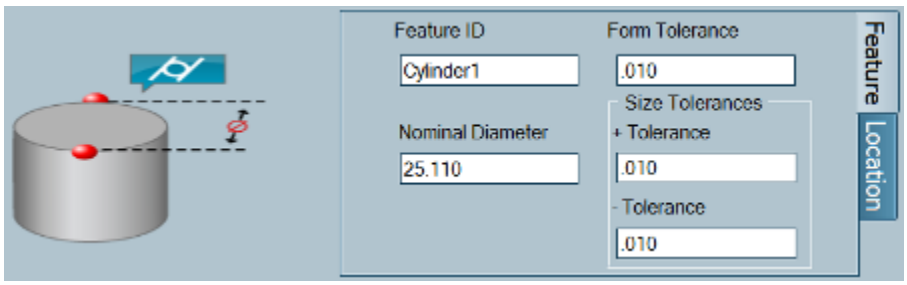
window. The **Feature ID** text in the **History** window is colored to indicate whether the feature is *in tolerance* (**GREEN**) or *out of tolerance* (**RED**).

6. Select the check box next to the new measured cylinder in the **History** window to toggle the display of the cylinder in the report. A measured cylinder includes evaluation of the Cylindricity and Diameter dimensions. A cylinder may also be evaluated in terms of True Position.

Editing Cylinder Parameters

To edit cylinder parameters:

1. Click the needed cylinder icon from the **History** window. The parameters for your cylinder are displayed.



2. Modify the cylinder's parameters as needed:

   - **Feature ID:** The MultiGage program automatically adds a feature ID of *Cylinder* plus an incremented number value (e.g. Cylinder4). You can change this value if needed.
   - **Nominal Diameter:** Define the nominal diameter value for the cylinder.

   Note: MultiGage guesses nominal values by rounding the measured value.

   - **Form Tolerance:** Provide the tolerance value for evaluating the cylinder's form.
   - **+ Tolerance** and - **Tolerance:** Provide the tolerance values for evaluating the cylinder's diameter.

   Note: Features excluded from the report do not require you to define nominal and tolerance values.

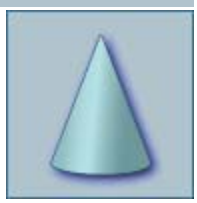
3. When using alignments you may modify the **Nominal** and **+/- Tolerance** values on the **Location** tab to control how the Location dimension is reported.
4. Click

to save your changes.

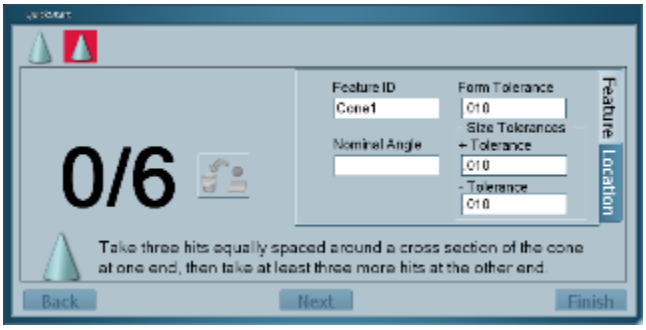
Measured Cone

To measure a cone:

1. Click the **Measure** button from the **Main** toolbar.



2. Select the **Measure Cone** button from the **QuickStart** window.



3. Using the ***RED*** machine button, take the first three hits at one end of the cone around a cross section that is perpendicular to the cone's axis. This defines the cone's orientation. Take at least three more hits at the other end of the cone. Hits should be taken so they are uniformly distributed around the circumference of the cone. As the hits are received, the MultiGage program updates the number of hits. When the minimum number of hits have been received, the nominal and dimension results for the cone are updated in the **Status** window.

Note: If you are not satisfied with a hit, you can remove the last hit by clicking the **Remove Hit** button or by pressing the ***WHITE*** machine button.

Note: You may take more hits than the specified minimum. For example 8/6 would display if you took 8 hits where you only need 6. This allows you to further define the cone's geometry. When dimensioning concentricity of a cone, it is recommended to take many hits to accurately define the cone's form.

4. Edit the cone's parameters if needed. See "Editing Cone Parameters" below.

5. Click **Finish** or press the ***YELLOW*** machine button. The new cone feature is added to the

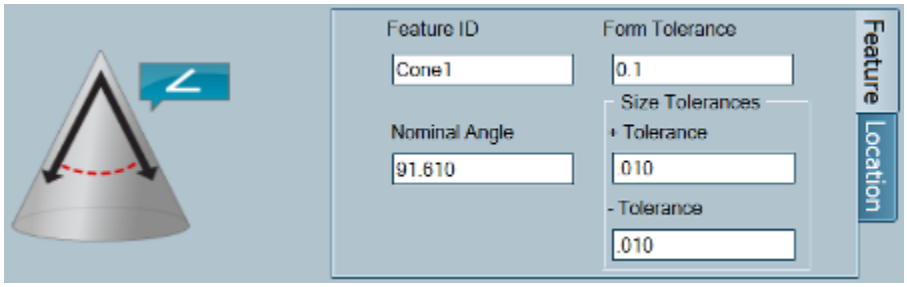
History window. The **Feature ID** text in the **History** window is colored to indicate whether the feature is *in tolerance* (**GREEN**) or *out of tolerance* (**RED**).

6. Select the check box next to the new measured cone in the **History** window to toggle the display of the cone in the report. A measured cone includes evaluation of the Concentricity and Cone Angle dimensions.

Editing Cone Parameters

To edit cone parameters:

1. Click the needed cone icon from the **History** window. The parameters for your cone are displayed.



2. Modify the cone's parameters as needed:
   - **Feature ID:** The MultiGage program automatically adds a feature ID of *Cone* plus an incremented number value (e.g. Cone4). You can change this value if needed.
   - **Nominal Angle:** Define the nominal angle value for the cone vertex.

   Note: MultiGage guesses nominal values by rounding the measured value.

   - **Form Tolerance:** Provide the tolerance value for evaluating the cone's form.
   - **+ Tolerance** and - **Tolerance:** Provide the tolerance values for evaluating the cone's vertex angle.

   Note: Features excluded from the report do not require you to define nominal and tolerance values.

3. When using alignments you may modify the **Nominal** and **+/- Tolerance** values on the **Location** tab to control how the Location dimension is reported.
4. Click

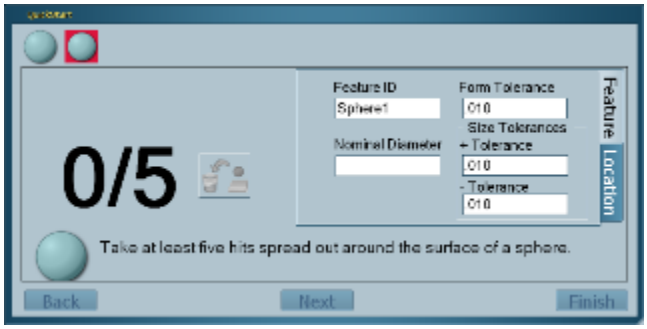
to save your changes.

Measured Sphere

To measure a sphere:

1. Click the **Measure** button from the **Main** toolbar.

2. Select the **Measure Sphere** button from the **QuickStart** window. The **QuickStart** sphere interface appears.



3. Using the ***RED*** machine button, take at least five hits to define the sphere feature. Hits should be taken so they are uniformly distributed around the circumference of the sphere. As the hits are received, the MultiGage program updates the number of hits. When the minimum number of hits have been received, the nominal and dimension results for the sphere are updated in the **Status** window.

Note: If you are not satisfied with a hit, you can remove the last hit by clicking the **Remove Hit** button or by pressing the ***WHITE*** machine button.

Note: You may take more hits than the specified minimum. For example 10/5 would display if you took 10 hits where you only need 5. This allows you to further define the sphere's geometry. When dimensioning roundness of a sphere you must take at least 6 hits. It is recommended to take many hits to accurately define the sphere's form.

4. Edit the sphere's parameters if needed. See "Editing Sphere Parameters" below.

5. Click **Finish** or press the ***YELLOW*** machine button. The new sphere feature is added to the **History** window. The **Feature ID** text in the

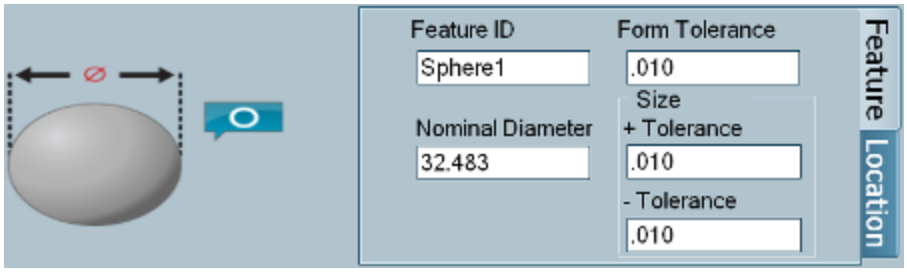
History window is colored to indicate whether the feature is *in tolerance* (**GREEN**) or *out of tolerance* (**RED**).

6. Select the check box next to the new measured sphere in the **History** window to toggle the display of the sphere in the report. A measured sphere includes evaluation of the Roundness and Diameter dimensions.

Editing Sphere Parameters

To edit sphere parameters:

1. Click the needed sphere icon from the **History** window. The parameters for your sphere are displayed.



2. Modify the sphere's parameters as needed:
   - **Feature ID:** The MultiGage program automatically adds a feature ID of *Sphere* plus an incremented number value (e.g. Sphere4). You can change this value if needed.
   - **Nominal Diameter:** Define the nominal diameter value for the sphere.

   Note: MultiGage guesses nominal values by rounding the measured value.

   - **Form Tolerance:** Provide the tolerance value for evaluating the sphere's form.
   - **+ Tolerance** and - **Tolerance:** Provide the tolerance values for evaluating the sphere's diameter.

   Note: Features excluded from the report do not require you to define nominal and tolerance values.

3. When using alignments you may modify the **Nominal** and **+/- Tolerance** values on the **Location** tab to control how the Location dimension is reported.
4. Click

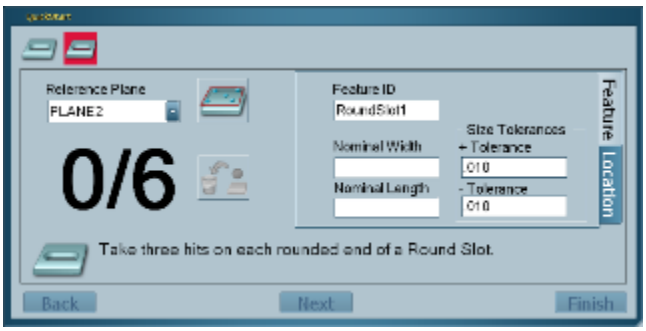
to save your changes.

Measured Round Slot

To measure a round slot:

1. Click the **Measure** button from the **Main** toolbar.

2. Select the **Measure Round Slot** button from the **QuickStart** window. The **QuickStart** round slot interface appears. If no **Reference Plane** exists at this point, you will need to measure it. See "Measuring a Reference Plane".



3. Select the round slot's **Reference Plane** from the drop-down list, or click "New Plane" button to create a new reference plane.

4. Using the ***RED*** machine button, take three hits on each of the rounded ends of the round slot feature. Hits should be taken so they are uniformly distributed around the circumference of the rounded ends. As the hits are received, the MultiGage program updates the number of hits. When the minimum number of hits have been received, the nominal and dimension results for the round slot are updated in the **Status** window.

Note: If you are not satisfied with a hit, you can remove the last hit by clicking the **Remove Hit** button or by pressing the ***WHITE*** machine button.

5. Edit the round slot's parameters if needed. See "Editing Round Slot Parameters" below.

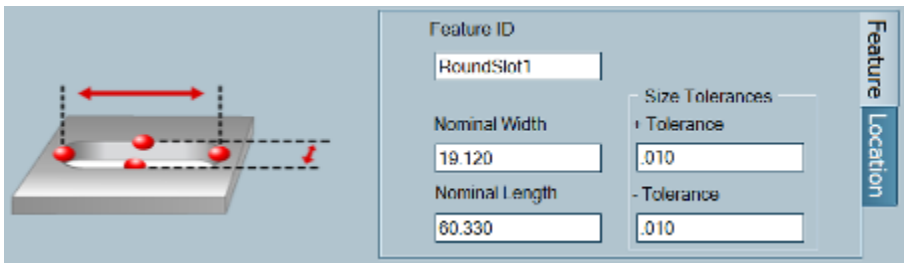
6. Click **Finish** or press the ***YELLOW*** machine button. The new round slot feature is added to the **History** window. The **Feature ID** text in the **History** window is colored to indicate whether the feature is *in tolerance* (**GREEN**) or *out of tolerance* (**RED**).

7. Select the check box next to the new measured round slot in the **History** window to toggle the display of the round slot in the report. A measured round slot includes evaluation of Width and Length dimensions. A round slot may also be evaluated in terms of True Position.

Editing Round Slot Parameters

To edit round slot parameters:

1. Click the needed round slot icon from the **History** window. The parameters for your round slot are displayed.



2. Modify the round slot's parameters as needed:
   - **Feature ID:** The MultiGage program automatically adds a feature ID of *RoundSlot* plus an incremented number value (e.g. RoundSlot2). You can change this value if needed.
   - **Nominal Width:** Define the nominal width value for the round slot.
   - **Nominal Length:** Define the nominal length value for the round slot.

   Note: MultiGage guesses nominal values by rounding the measured value.

   - **+ Tolerance** and - **Tolerance:** Provide the tolerance values for evaluating the round slot's width and length.

   Note: Features excluded from the report do not require you to define nominal and tolerance values.

3. When using alignments you may modify the **Nominal** and **+/- Tolerance** values on the **Location** tab to control how the Location dimension is reported.
4. Click

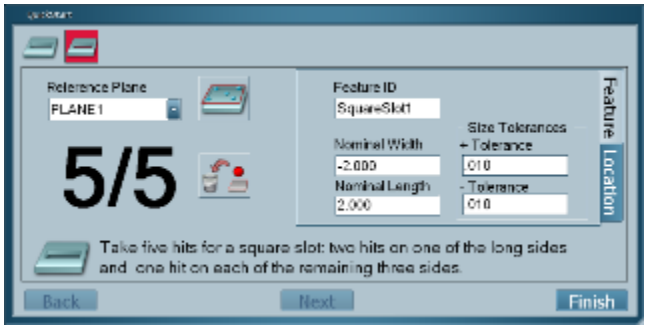
to save your changes.

Measured Square Slot

To measure a square slot:

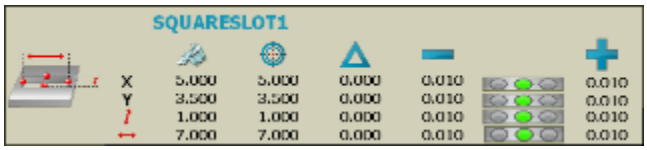
1. Click the **Measure** button from the **Main** toolbar.

2. Select the **Measure Round Slot** button from the **QuickStart** window. The **QuickStart** square slot interface appears. If no **Reference Plane** exists at this point, you will need to measure it. See "Measuring a Reference Plane".



3. Select the square slot's **Reference Plane** from the drop-down list, or click "New Plane" button to create a new reference plane.

4. Using the ***RED*** machine button, take two hits on one of the long sides of the square slot feature. Take one hit on each of the remaining three sides. As the hits are received the MultiGage program updates the number of hits. When the minimum number of hits have been received, the nominal and dimension results for the square slot are updated in the **Status** window.



Note: If you are not satisfied with a hit, you can remove the last hit by clicking the **Remove Hit** button or by pressing the ***WHITE*** machine button.

5. Edit the square slot's parameters if needed. See "Editing Square Slot Parameters" below.

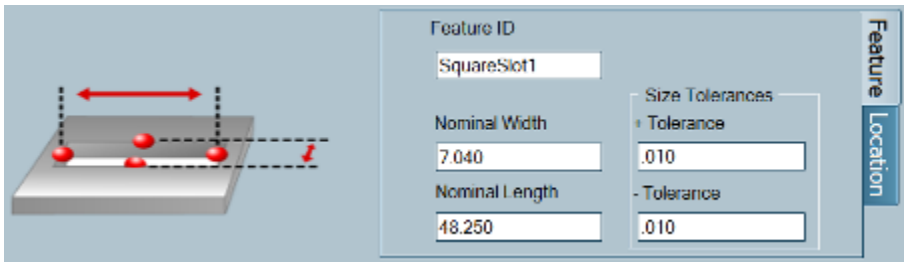
6. Click **Finish** or press the ***YELLOW*** machine button. The new square slot feature is added to the **History** window. The **Feature ID** text in the **History** window is colored to indicate whether the feature is *in tolerance* (**GREEN**) or *out of tolerance* (**RED**).

7. Select the check box next to the new measured square slot in the **History** window to toggle the display of the square slot in the report. A measured square slot includes evaluation of Width and Length dimensions. A square slot may also be evaluated in terms of True Position.

Editing Square Slot Parameters

To edit square slot parameters:

1. Click the needed square slot icon from the **History** window. The parameters for your square slot are displayed.



2. Modify the square slot's parameters as needed:
   - **Feature ID:** The MultiGage program automatically adds a feature ID of *SquareSlot* plus an incremented number value (e.g. SquareSlot2). You can change this value if needed.
   - **Nominal Width:** Define the nominal width value for the square slot.
   - **Nominal Length:** Define the nominal length value for the square slot.

   Note: MultiGage guesses nominal values by rounding the measured value.

   - **+ Tolerance** and - **Tolerance:** Provide the tolerance values for evaluating the square slot's width and length.

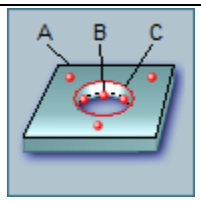
   Note: Features excluded from the report do not require you to define nominal and tolerance values.

3. When using alignments you may modify the **Nominal** and **+/- Tolerance** values on the **Location** tab to control how the Location dimension is reported.
4. Click

to save your changes.

Measuring a Reference Plane

2D Features such as a circle require the use of a reference plane to establish the feature's position. The reference plane coincides with the feature's nominal values. For example, a circle's nominal position is created at the location where the measured circle of a hole is projected to the surface where the hole resides.

	*Example of Reference Plane for a Circle* **A** - Reference plane as determined by hits on surface. **B** - Measured Circle as determined by hits inside of circle. **C** - Nominal Circle position where measured circle is projected to the reference plane.

If you begin measuring a circle feature and there are no previously measured reference planes, you will be required measure a reference plane.

Features that require a reference plane are:

- Measured Line
- Measured Circle
- Measured Round Slot
- Measured Square Slot

To measure a reference plane:

1. Begin measuring a feature where a reference plane is required.

   - If no reference plane exists in your part program, the reference plane QuickStart interface will open to create a reference plane.
   - If a plane does exist, but you still need to measure a new reference plane, click the **New Plane** button next to the **Reference Plane** drop-down list.

2. Using the ***RED*** machine button, measure the plane by taking at least three hits. As the hits are received the MultiGage program updates the number of hits. When the minimum number of hits have been received, nominal and dimension results for the plane are updated in the **Status** window.

Note: If you are not satisfied with a hit, you can remove the last hit by clicking the **Remove Hit** button or by pressing the ***WHITE*** machine button.

3. Edit the reference plane's parameters if needed.

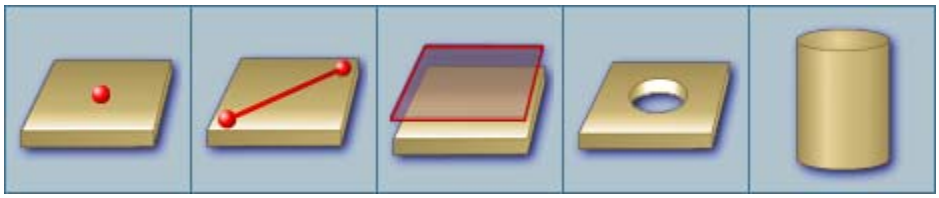
For information on these parameters, see the "Measured Plane" topic.

Note: Features excluded from the report do not require you to define nominal and tolerance values.

4. Click **Finish** or press the ***YELLOW*** machine button. The new plane feature is added to the **History** window. The QuickStart interface returns to feature type for which you were measuring a reference plane; you are now ready to measure the needed feature.

5. Select the check box next to the new reference plane in the **History** window to toggle the display of the plane's flatness dimension in the report.

Constructing Features



From existing features (already measured or constructed) you can construct new features such as points, lines, circles, etc.

The following sections provides steps to construct features:

- Constructed Point
- Constructed Line
- Constructed Plane
- Constructed Circle
- Constructed Cylinder

After the needed constructed feature has been selected from the **QuickStart** window, only the possible input features for that constructed feature are available for selection; all others are grayed out.

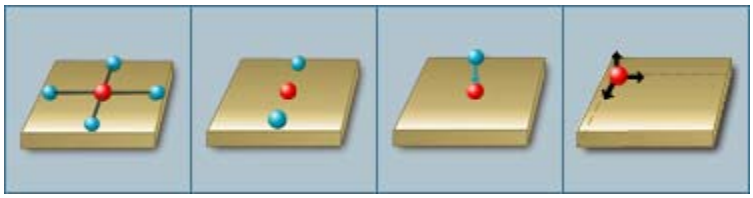
Additionally, based on the selected constructed feature and first input feature, only the possible secondary input features are available for selection; all others are grayed out. Selection for more than two input features functions in a similar fashion.

Input Characteristics

The MultiGage program uses the *point*, *plane*, *line* or *circle* characteristics of features as input for feature construction. The input characteristics are derived as follows:

Points	• From Points. • From the *centroid* of Circles, Spheres, Square Slots or Round Slots.
Planes	• From Planes.
Lines	• From Lines. • From the *axis* of a Cylinder or Cone. • From the *long axis* of a Round Slot or Square Slot.
Circles	• From Circles. • From the *circular form* of a Sphere, Cylinder, or Cone.

Constructed Point



Constructing a point can be done using the following methods:

- Intersection - Line and Plane
- Intersection - Two Lines
- Mid Point - Two Points
- Projection - Point to Line
- Projection - Point to Plane
- Corner - Three Planes

Intersection - Line and Plane

A constructed point can be created using *one line* and *one plane* as input features. The constructed point is created at the intersection of the line and plane so they must intersect.

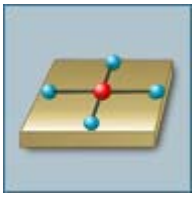
The line input can be derived from the *line characteristic* of a line, cylinder, cone, round slot or square slot. See "Constructing Features" for more information on input characteristics.

Note: The line feature is added as an input feature before the plane feature.

To create an intersected point from *one line* and *one plane*:

1. Click the **Construct** button from the **Main** toolbar.

2. From the **Constructed Point** sub-menu in the **QuickStart** window, select the **Point Intersection** button. The **QuickStart** feature selection grid appears and the progress bar displays question marks for the needed input features.



The highlighted icon indicates the current step of the constructed feature task.

3. Using one of the following methods, select a *line* or a feature with a *line characteristic*:

Input from Selection Grid

Click the needed feature button from the selection grid. See the related topic for information on measuring the selected feature:

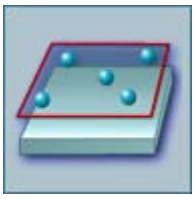
- Measured Line
- Measured Cylinder
- Measured Cone
- Measured Round Slot
- Measured Square Slot

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed feature from the **Input Feature List**. Select a *line*, *cylinder*, *cone*, *round slot*, or *square slot*.

Note: Only possible input features will be displayed for selection.

4. Using one of the following methods, select a *plane*:



Input From Selection Grid

i. Click the plane button from the feature selection grid.
ii. Take at least three hits to define the plane feature.
iii. Edit the parameters for the plane if needed.
iv. Click **Next** or press the ***YELLOW*** machine button.

See the "Measured Plane" topic for more detailed information on measuring planes.

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed plane feature from the **Input Feature List**.

5. Verify the constructed point and click **Finish** or press the ***YELLOW*** machine button. The new constructed point is added to the **History** window.

6. Select the check box next to the new constructed point feature in the **History** window to toggle the display of the point in the report.

Intersection - Two Lines

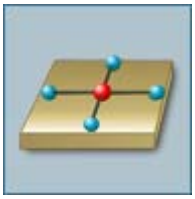
A constructed point can be created using *two lines* as input features. These lines can be either 2D (residing in the same plane) or 3D (not on the same plane).

The line input can be derived from the *line characteristic* of a line, cylinder, cone, round slot or square slot. See "Constructing Features" for more information on input characteristics.

To create a intersected point from *two lines*:

1. Click the **Construct** button from the **Main** toolbar.

2. From the **Constructed Point** sub-menu in the **QuickStart** window, select the **Point Intersection** button. The **QuickStart** feature selection grid appears and the progress bar displays question marks for the needed input features.



The highlighted icon indicates the current step of the constructed feature task.

3. Using one of the following methods, select a *line* or a feature with a *line characteristic*:

Input from Selection Grid

Click the needed feature button from the selection grid. See the related topic for information on measuring the selected feature:

- Measured Line
- Measured Cylinder
- Measured Cone
- Measured Round Slot
- Measured Square Slot

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed feature from the **Input Feature List**. Select a *line*, *cylinder*, *cone*, *round slot*, or *square slot*.

Note: Only possible input features will be displayed for selection.

4. Repeat step 3 for the second feature with a *line characteristic*.

5. Verify the constructed point and click **Finish** or press the ***YELLOW*** machine button. The new constructed point is added to the **History** window.

6. Select the check box next to the new constructed point feature in the **History** window to toggle the display of the point in the report.

Mid Point - Two Points

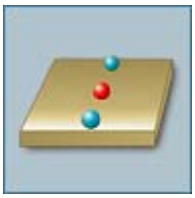
A constructed point can be created at the midpoint location between *two points*.

The point input can be derived from the *centroid* of a point, circle, round slot, square slot or sphere. See "Constructing Features" for more information on input characteristics.

To create a mid point between *two points*:

1. Click the **Construct** button from the **Main** toolbar.

2. From the **Constructed Point** sub-menu in the **QuickStart** window, select the **Mid Point** button. The **QuickStart** feature selection grid appears and the progress bar displays question marks for the needed input features.



The highlighted icon indicates the current step of the constructed feature task.

3. Using one of the following methods, select an input feature where the centroid represents the *point characteristic*:

Input from Selection Grid

Click the needed feature button from the selection grid. See the related topic for information on measuring the selected feature:

- Measured Point
- Measured Circle
- Measured Sphere
- Measured Round Slot
- Measured Square Slot

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed feature from the **Input Feature List**. Select a *point*, *circle*, *sphere*, *round slot* or *square slot*.

Note: Only possible input features will be displayed in the **Input Feature List** for selection.

4. Repeat step 3 for the *second point* feature.

5. Verify the constructed point and click **Finish** or press the ***YELLOW*** machine button. The new

constructed point is added to the **History** window.

6. Select the check box next to the new constructed point feature in the **History** window to toggle the display of the point in the report.

Projection - Point to Line

A constructed point can be created using the projection of *one point* to *one line.* The constructed point is created at the location where the projected point intersects the line. The point is projected along a line that is perpendicular to the line.

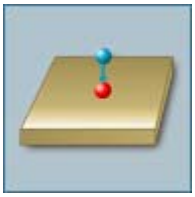
The point input can be derived from the *centroid* of a point, circle, round slot, square slot or sphere. The line input can be derived from the *line characteristic* of a line, cylinder, cone, round slot or square slot. See "Constructing Features" for more information on input characteristics.

Note: When creating a projected point, the feature with the *point characteristic* is selected first.

To create a projected point from *one point* and *one line*:

1. Click the **Construct** button from the **Main** toolbar.

2. From the **Constructed Point** sub-menu in the **QuickStart** window, select the **Point Projection** button. The **QuickStart** feature selection grid appears and the progress bar displays question marks for the needed input features.



The highlighted icon indicates the current step of the constructed feature task.

3. Using one of the following methods, select an input feature where the centroid represents the *point characteristic*:

Input from Selection Grid

Click the needed feature button from the selection grid. See the related topic for

information on measuring the selected feature:

- Measured Point
- Measured Circle
- Measured Sphere
- Measured Round Slot
- Measured Square Slot

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed feature from the **Input Feature List**. Select a *point*, *circle*, *sphere*, *round slot* or *square slot*.

Note: Only possible input features will be displayed in the **Input Feature List** for selection.

4. Using one of the following methods, select a *line* or a feature with a *line characteristic*:

Input from Selection Grid

Click the needed feature button from the selection grid. See the related topic for information on measuring the selected feature:

- Measured Line
- Measured Cylinder
- Measured Cone
- Measured Round Slot
- Measured Square Slot

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed feature from the **Input Feature List**. Select a *line*, *cylinder*, *cone*, *round slot*, or *square slot*.

Note: Only possible input features will be displayed for selection.

5. Verify the constructed point and click **Finish** or press the ***YELLOW*** machine button. The new constructed point is added to the **History** window.

6. Select the check box next to the new constructed point feature in the **History** window

to toggle the display of the point in the report.

Projection - Point to Plane

A constructed point can be created using the projection of *one point* to *one plane.* The constructed point is created at the location where the projected point intersects the plane. The point is projected along a line that is perpendicular to the plane.

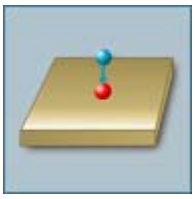
The point input can be derived from the *centroid* of a point, circle, round slot, square slot or sphere. See "Constructing Features" for more information on input characteristics.

Note: When creating a projected point, the feature with the *point characteristic* is selected first.

To create a projected point from *one point* and *one plane*:

1. Click the **Construct** button from the **Main** toolbar.

2. From the **Constructed Point** sub-menu in the **QuickStart** window, select the **Point Projection** button. The **QuickStart** feature selection grid appears and the progress bar displays question marks for the needed input features.



The highlighted icon indicates the current step of the constructed feature task.

3. Using one of the following methods, select an input feature where the centroid represents the *point characteristic*:

Input from Selection Grid

Click the needed feature button from the selection grid. See the related topic for information on measuring the selected feature:

- Measured Point
- Measured Circle
- Measured Sphere

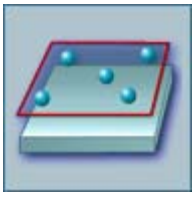
- Measured Round Slot
- Measured Square Slot

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed feature from the **Input Feature List**. Select a *point*, *circle*, *sphere*, *round slot* or *square slot*.

Note: Only possible input features will be displayed in the **Input Feature List** for selection.

4. Using one of the following methods, select a *plane*:



Input From Selection Grid

i. Click the plane button from the feature selection grid.
ii. Take at least three hits to define the plane feature.
iii. Edit the parameters for the plane if needed.
iv. Click **Next** or press the ***YELLOW*** machine button.

See the "Measured Plane" topic for more detailed information on measuring planes.

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed plane feature from the **Input Feature List**.

5. Verify the constructed point and click **Finish** or press the ***YELLOW*** machine button. The new constructed point is added to the **History** window.

6. Select the check box next to the new constructed point feature in the **History** window to toggle the display of the point in the report.

Corner - Three Planes

A constructed corner point can be created using *three planes* as input features. The constructed corner point is created at the intersection of the three planes so all three planes must intersect.

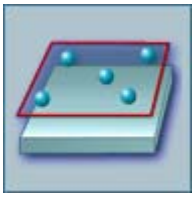
To create a corner point from *three planes*:

1. Click the **Construct** button from the **Main** toolbar.

2. From the **Constructed Point** sub-menu in the **QuickStart** window, select the **Corner Point** button. The **QuickStart** feature selection grid appears and the progress bar displays question marks for the needed input features.

The highlighted icon indicates the current step of the constructed feature task.

3. Using one of the following methods, select a *plane*:



Input From Selection Grid

i. Click the plane button from the feature selection grid.
ii. Take at least three hits to define the plane feature.
iii. Edit the parameters for the plane if needed.
iv. Click **Next** or press the ***YELLOW*** machine button.

See the "Measured Plane" topic for more detailed information on measuring planes.

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed plane feature from the **Input Feature List**.

4. Repeat step 3 for the *second plane* feature.

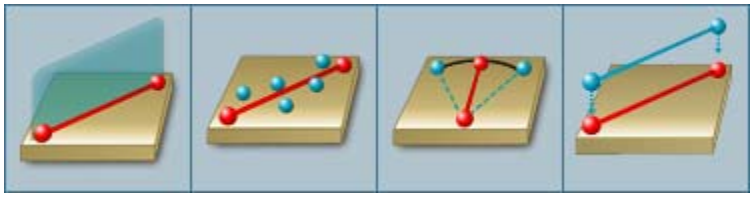
5. Repeat step 3 for the *third plane* feature.

6. Verify the constructed point and click **Finish** or press the ***YELLOW*** machine button. The new constructed point is added to the **History** window.

7. Select the check box next to the new constructed point feature in the **History** window

to toggle the display of the point in the report.

Constructed Line



Constructing a line can be done using the following methods:

- Intersection - Two Planes
- Best Fit Line - Multiple Points
- Mid Line - Two Lines
- Projection - Line to Plane

Intersection - Two Planes

A constructed line can be created using *two planes* as input features. The constructed line is created at the intersection of the two planes so they must intersect.

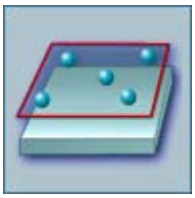
To create a intersected line from *two planes*:

1. Click the **Construct** button from the **Main** toolbar.

2. From the **Constructed Line** sub-menu in the **QuickStart** window, select the **Line Intersection** button. The **QuickStart** feature selection grid appears and the progress bar displays question marks for the needed input features.

The highlighted icon indicates the current step of the constructed feature task.

3. Using one of the following methods, select a *plane*:



Input From Selection Grid

i. Click the plane button from the feature selection grid.
ii. Take at least three hits to define the plane feature.
iii. Edit the parameters for the plane if needed.
iv. Click **Next** or press the ***YELLOW*** machine button.

See the "Measured Plane" topic for more detailed information on measuring planes.

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed plane feature from the **Input Feature List**.

4. Repeat step 3 for the *second plane* feature.

5. Verify the constructed line and click **Finish** or press the ***YELLOW*** machine button. The new constructed line is added to the **History** window.

6. Select the check box next to the new constructed line feature in the **History** window to toggle the display of the line in the report.

Best Fit Line - Multiple Points

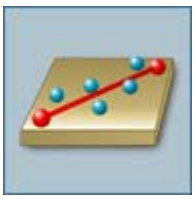
A constructed line can be created using the best fit calculation from *two or more points*.

The point input can be derived from the *centroid* of a point, circle, round slot, square slot or sphere. See "Constructing Features" for more information on input characteristics.

To create a best fit line from *two or more points*:

1. Click the **Construct** button from the **Main** toolbar.

2. From the **Constructed Line** sub-menu in the **QuickStart** window, select the **Best Fit Line** button. The **QuickStart** feature selection grid appears and the progress bar displays question marks for the needed input features.



The highlighted icon indicates the current step of the constructed feature task.

3. Using one of the following methods, select an input feature where the centroid represents the *point characteristic*:

Input from Selection Grid

Click the needed feature button from the selection grid. See the related topic for information on measuring the selected feature:

- Measured Point
- Measured Circle
- Measured Sphere
- Measured Round Slot
- Measured Square Slot

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed feature from the **Input Feature List**. Select a *point*, *circle*, *sphere*, *round slot* or *square slot*.

Note: Only possible input features will be displayed in the **Input Feature List** for selection.

4. Repeat step 3 for any *additional points* until all the needed features have been added. You must have at least three input features.

5. Verify the constructed line and click **Finish** or press the ***YELLOW*** machine button. The new constructed line is added to the **History** window.

6. Select the check box next to the new constructed line feature in the **History** window to toggle the display of the line in the report.

Mid Line - Two Lines

A constructed point can be created at the mid location between *two lines*. A mid line is also referred to as a average line or a bisector line.

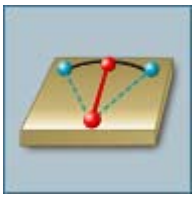
The line input can be derived from the *line characteristic* of a line, cylinder, cone, round slot or square slot. See "Constructing Features" for more information on input characteristics.

To create a mid line between *two lines*:

1. Click the **Construct** button from the **Main** toolbar.

2. From the **Constructed Line** sub-menu in the **QuickStart** window, select the **Mid Line** button. The **QuickStart** feature selection grid appears and the progress bar displays question marks for the needed input features.

The highlighted icon indicates the current step of the constructed feature task.



3. Using one of the following methods, select a *line* or a feature with a *line characteristic*:

Input from Selection Grid

Click the needed feature button from the selection grid. See the related topic for information on measuring the selected feature:

- Measured Line
- Measured Cylinder
- Measured Cone
- Measured Round Slot
- Measured Square Slot

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed feature from the **Input Feature List**. Select a *line*, *cylinder*, *cone*, *round slot*, or *square slot*.

Note: Only possible input features will be displayed for selection.

4. Repeat step 3 for the *second line* feature.

5. Verify the constructed line and click **Finish** or press the ***YELLOW*** machine button. The new constructed line is added to the **History** window.

6. Select the check box next to the new constructed line feature in the **History** window to toggle the display of the line in the report.

Projection - Line to Plane

A constructed line can be created using the projection of *one line* to *one plane.* The constructed line is created at the location where the projected line intersects the plane. The line is projected along a plane that is perpendicular to the plane.

The line input can be derived from the *line characteristic* of a line, cylinder, cone, round slot or square slot. See "Constructing Features" for more information on input characteristics.

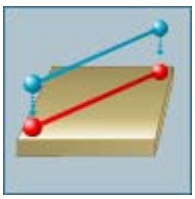
Note: When creating a projected line, the feature with the *line characteristic* is selected first.

To create a projected line from *one line* and *one plane*:

1. Click the **Construct** button from the **Main** toolbar.

2. From the **Constructed Line** sub-menu in the **QuickStart** window, select the **Line Projection** button. The **QuickStart** feature selection grid appears and the progress bar displays question marks for the needed input features.

The highlighted icon indicates the current step of the constructed feature task.



3. Using one of the following methods, select a *line* or a feature with a *line characteristic*:

Input from Selection Grid

Click the needed feature button from the selection grid. See the related topic for information on measuring the selected feature:

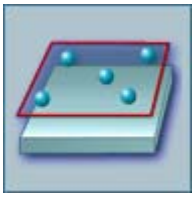
- Measured Line
- Measured Cylinder
- Measured Cone
- Measured Round Slot
- Measured Square Slot

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed feature from the **Input Feature List**. Select a *line*, *cylinder*, *cone*, *round slot*, or *square slot*.

Note: Only possible input features will be displayed for selection.

4. Using one of the following methods, select a *plane*:



Input From Selection Grid

i. Click the plane button from the feature selection grid.
ii. Take at least three hits to define the plane feature.
iii. Edit the parameters for the plane if needed.
iv. Click **Next** or press the ***YELLOW*** machine button.

See the "Measured Plane" topic for more detailed information on measuring planes.

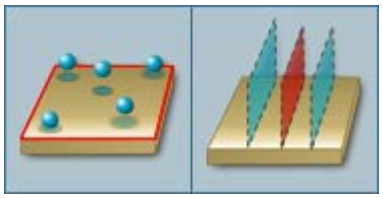
Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed plane feature from the **Input Feature List**.

5. Verify the constructed line and click **Finish** or press the ***YELLOW*** machine button. The new constructed line is added to the **History** window.

6. Select the check box next to the new constructed line feature in the **History** window to toggle the display of the line in the report.

Constructed Plane



Constructing a plane can be done using the following methods:

- Best Fit Plane - Multiple Points
- Mid Plane - Two Planes

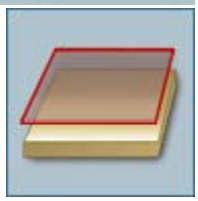
Best Fit Plane - Multiple Points

A constructed plane can be created using the best fit calculation from *two or more points*.

The point input can be derived from the *centroid* of a point, circle, round slot, square slot or sphere. See "Constructing Features" for more information on input characteristics.

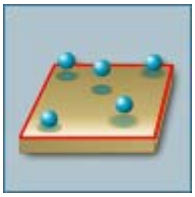
To create a best fit plane from *two or more points*:

1. Click the **Construct** button from the **Main** toolbar.



2. From the **Constructed Plane** sub-menu in the **QuickStart** window, select the **Best Fit Plane** button. The **QuickStart** feature selection grid appears and the progress bar displays question marks for the needed input features.

The highlighted icon indicates the current step of the constructed feature task.



3. Using one of the following methods, select an input feature where the centroid represents the *point characteristic*:

Input from Selection Grid

Click the needed feature button from the selection grid. See the related topic for

information on measuring the selected feature:

- Measured Point
- Measured Circle
- Measured Sphere
- Measured Round Slot
- Measured Square Slot

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed feature from the **Input Feature List**. Select a *point*, *circle*, *sphere*, *round slot* or *square slot*.

Note: Only possible input features will be displayed in the **Input Feature List** for selection.

4. Repeat step 3 for any *additional points* until all the needed features have been added. You must have at least three input features.

5. Verify the constructed plane and click **Finish** or press the ***YELLOW*** machine button. The new constructed plane is added to the **History** window.

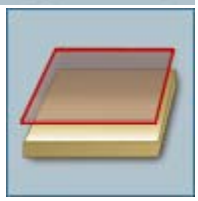
6. Select the check box next to the new constructed plane feature in the **History** window to toggle the display of the plane in the report.

Mid Plane - Two Planes

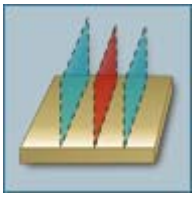
A constructed plane can be created at the mid location between *two planes*.

To create a mid plane between *two planes*:

1. Click the **Construct** button from the **Main** toolbar.

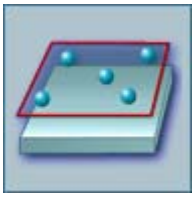


2. From the **Constructed Plane** sub-menu in the **QuickStart** window, select the **Mid Plane** button. The **QuickStart** feature selection grid appears and the progress bar displays question marks for the needed input features.



The highlighted icon indicates the current step of the constructed feature task.

3. Using one of the following methods, select a *plane*:



Input From Selection Grid

i. Click the plane button from the feature selection grid.
ii. Take at least three hits to define the plane feature.
iii. Edit the parameters for the plane if needed.
iv. Click **Next** or press the ***YELLOW*** machine button.

See the "Measured Plane" topic for more detailed information on measuring planes.

Input from Input Feature List (History Window)

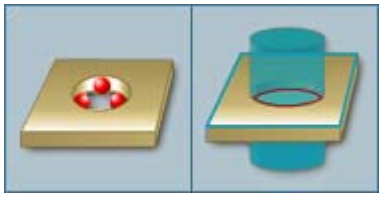
Select the **Feature ID** or feature icon for the needed plane feature from the **Input Feature List**.

4. Repeat step 3 for the *second plane* feature.

5. Verify the constructed plane and click **Finish** or press the ***YELLOW*** machine button. The new constructed plane is added to the **History** window.

6. Select the check box next to the new constructed plane feature in the **History** window to toggle the display of the plane in the report.

Constructed Circle



Constructing a circle can be done using the following methods:

- Best Fit Circle - Multiple Points

- Intersection - Plane with Cone or Cylinder

Best Fit Circle - Multiple Points

A constructed circle can be created using the best fit calculation from *two or more points*.

The point input can be derived from the *centroid* of a point, circle, round slot, square slot or sphere. See "Constructing Features" for more information on input characteristics.

To create a best fit circle from *two or more points*:

1. Click the **Construct** button from the **Main** toolbar.

2. From the **Constructed Circle** sub-menu in the **QuickStart** window, select the **Best Fit Circle** button. The **QuickStart** feature selection grid appears and the progress bar displays question marks for the needed input features.

The highlighted icon indicates the current step of the constructed feature task.

3. Using one of the following methods, select an input feature where the centroid represents the *point characteristic*:

Input from Selection Grid

Click the needed feature button from the selection grid. See the related topic for information on measuring the selected feature:

- Measured Point
- Measured Circle
- Measured Sphere
- Measured Round Slot
- Measured Square Slot

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed feature from the **Input Feature List**. Select a *point*, *circle*, *sphere*, *round slot* or *square slot*.

Note: Only possible input features will be displayed in the **Input Feature List** for selection.

4. Repeat step 3 for any *additional points* until all the needed features have been added. You must have at least three input features.

5. Verify the constructed circle and click **Finish** or press the ***YELLOW*** machine button. The new constructed circle is added to the **History** window.

6. Select the check box next to the new constructed circle feature in the **History** window to toggle the display of the circle in the report.

Intersection - Plane with Cone or Cylinder

A constructed circle can be created using *one plane* and either a *cone* or a *cylinder* as input features. The constructed circle is created at the intersection of the plane and a cone or cylinder so they must intersect.

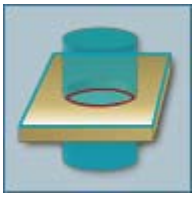
Note: You can specify the input plane and (cone or cylinder) features in any order, but this example shows the plane being added first.

To create a constructed point from *one plane* and *one cone* or *one cylinder*:

1. Click the **Construct** button from the **Main** toolbar.

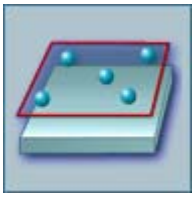
2. From the **Constructed Circle** sub-menu in the **QuickStart** window, select the **Circle Intersection** button. The **QuickStart** feature selection grid appears and the progress bar displays question marks for the needed input features.

The highlighted icon indicates the current step of the



constructed feature task.

3. Using one of the following methods, select a *plane*:



Input From Selection Grid

i. Click the plane button from the feature selection grid.
ii. Take at least three hits to define the plane feature.
iii. Edit the parameters for the plane if needed.
iv. Click **Next** or press the ***YELLOW*** machine button.

See the "Measured Plane" topic for more detailed information on measuring planes.

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed plane feature from the **Input Feature List**.

4. Using one of the following methods, select a *cone* or *cylinder*:

Input From Selection Grid

i. Click the cone or cylinder button from the feature selection grid.
ii. Take at least six hits to define the cone or cylinder feature.
iii. Edit the parameters for the cone or cylinder if needed.
iv. Click **Next** or press the ***YELLOW*** machine button.

or

See the "Measured Cone" or "Measured Cylinder" topics for more detailed information on measuring cones or cylinders.

Input From History Window

Select the **Feature ID** or feature icon for the needed cone or cylinder feature from the **Input Feature List**.

5. Verify the constructed circle and click **Finish** or press the ***YELLOW*** machine button. The new constructed circle is added to the **History** window.

6. Select the check box next to the new constructed circle feature in the **History** window to toggle the display of the circle in the report.

Constructed Cylinder

A constructed cylinder can be created using the best fit calculation from *two circles*.

To create a cylinder from *two circles*:

1. Click the **Construct** button from the **Main** toolbar.

2. Select the **Cylinder** button from the Construction toolbar. The **QuickStart** feature selection grid appears and the progress bar displays question marks for the needed input features.

The highlighted icon indicates the current step of the constructed feature task.

3. Using one of the following methods, select a *circle*:

Input From Selection Grid

i. Click the circle button from the feature selection grid.
ii. Take at least three hits to define the circle feature.
iii. Edit the parameters for the circle if needed.
iv. Click **Next** or press the ***YELLOW*** machine button.

See the "Measured Circle" topic for more detailed information on measuring circles.

Input From History Window

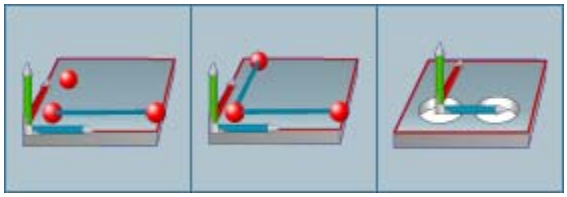
Select the **Feature ID** or feature icon for the needed circle feature from the **Input Feature List**.

4. Repeat step 3 for the *second circle* feature.

5. Verify the constructed cylinder and click **Finish** or press the ***YELLOW*** machine button. The new constructed cylinder is added to the **History** window.

6. Select the check box next to the new constructed cylinder feature in the **History** window to toggle the display of the cylinder in the report.

Creating Alignments



Alignments are used to align the part coordinate system with the machine coordinate system. To do this, you must align each major axis by selecting features that represent each axis. The 3-2-1 alignment is the most common alignment, which uses a plane, line and point as inputs. MultiGage makes use of the following alignments:

- Plane-Line-Point Alignment
- Plane-Line-Line Alignment
- Plane-Circle-Circle Alignment

The process of creating an alignment involves:

1. 'Leveling' the *plane* by aligning the plane's normal vector to the *first* axis.
2. 'Rotating' the *line* about the first axis to the *second* axis. The line may be defined by either a line feature or the implied line between two circles. The positive direction of the line goes from the first point (circle) to the second point (circle).
3. "Translating' the *point*, *line* or *circle* to the origin of the coordinate system.

Note: Constructed features may also be selected from the **Input Feature List** as input for alignments. Only valid constructed features will be available for selection.

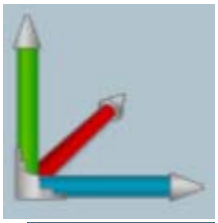
Input Characteristics

The MultiGage program uses the *point*, *plane, line,* or *circle* characteristics of features as input for creating alignments. The input characteristics are derived as follows:

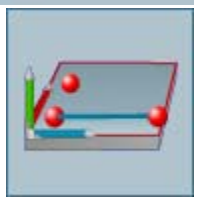
Points	• From Points. • From the *centroid* of Circles, Spheres, Square Slots or Round Slots.
Planes	• From Planes.
Lines	• From Lines. • From the *axis* of a Cylinder or Cone. • From the *long axis* of a Round Slot or Square Slot.
Circles	• From Circles. • From the *circular form* of a Sphere, Cylinder, or Cone.

Plane-Line-Point Alignment

To create a *plane-line-point* (3-2-1) alignment:



1. Click the **Alignment** button from the **Main** toolbar.



2. Select the **Create a Plane-Line-Point Alignment** button from the **QuickStart** window. The **QuickStart** feature selection grid appears and the progress bar displays question marks for the needed input features.

The highlighted icon indicates the current alignment step.

3. Using one of the following methods, select a *plane*:

Input From Selection Grid

i. Click the plane button from the feature selection grid.
ii. Take at least three hits to define the plane feature.
iii. Edit the parameters for the plane if needed.
iv. Click **Next** or press the ***YELLOW*** machine button.

See the "Measured Plane" topic for more detailed information on measuring planes.

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed plane feature from the **Input Feature List**.

4. Using one of the following methods, select a *line* or a feature with a *line characteristic*:

Input from Selection Grid

Click the needed feature button from the selection grid. See the related topic for information on measuring the selected feature:

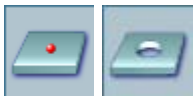
- Measured Line
- Measured Cylinder
- Measured Cone
- Measured Round Slot
- Measured Square Slot

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed line feature from the **Input Feature List**.

Note: Only possible input features will be displayed for selection.

5. Using one of the following methods, select an input feature where the centroid represents the *point characteristic*:



Input from Selection Grid

Click the needed feature button from the selection grid. See the related topic for information on measuring the selected feature:

- Measured Point
- Measured Circle
- Measured Sphere
- Measured Round Slot
- Measured Square Slot

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed line feature from the **Input Feature List**.

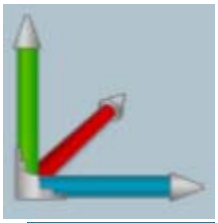
Note: Only possible input features will be displayed for selection.

6. Verify the plane-line-point alignment and click **Finish** or press the ***YELLOW*** machine button. The new alignment is added to the **History** window.

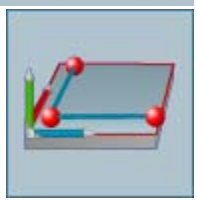
7. Select the check box next to the new alignment in the **History** window to toggle the display of the alignment in the report.

Plane-Line-Line Alignment

To create a *plane-line-line* alignment:



1. Click the **Alignment** button from the **Main** toolbar.



2. Select the **Create a Plane-Line-Line Alignment** button from the **QuickStart** window. The **QuickStart** feature selection grid appears and the progress bar displays question marks for the needed input features.

The highlighted icon indicates the current alignment step.

3. Using one of the following methods, select a *plane*:

Input From Selection Grid

i. Click the plane button from the feature selection grid.
ii. Take at least three hits to define the plane feature.
iii. Edit the parameters for the plane if needed.

iv. Click **Next** or press the ***YELLOW*** machine button.

See the "Measured Plane" topic for more detailed information on measuring planes.

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed plane feature from the **Input Feature List**.

4. Using one of the following methods, select a *line* or a feature with a *line characteristic*:

Input from Selection Grid

Click the needed feature button from the selection grid. See the related topic for information on measuring the selected feature:

- Measured Line
- Measured Cylinder
- Measured Cone
- Measured Round Slot
- Measured Square Slot

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed line feature from the **Input Feature List**.

Note: Only possible input features will be displayed for selection.

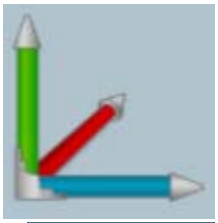
5. Repeat Step 4 for the second feature with a *line characteristic*.

6. Verify the plane-line-line alignment and click **Finish** or press the ***YELLOW*** machine button. The new alignment is added to the **History** window.

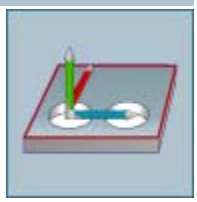
7. Select the check box next to the new alignment in the **History** window to toggle the display of the alignment in the report.

Plane-Circle-Circle Alignment

To create a *plane-circle-circle* alignment:



1. Click the button from the **Main** toolbar.



2. Select the **Create a Plane-Circle-Circle Alignment** button from the **QuickStart** window. The **QuickStart** feature selection grid appears and the progress bar displays question marks for the needed input features.

The highlighted icon indicates the current alignment step.

3. Using one of the following methods, select a *plane*:

Input From Selection Grid

i. Click the plane button from the feature selection grid.
ii. Take at least three hits to define the plane feature.
iii. Edit the parameters for the plane if needed.
iv. Click **Next** or press the ***YELLOW*** machine button.

See the "Measured Plane" topic for more detailed information on measuring planes.

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed plane feature from the **Input Feature List**.

4. Using one of the following methods, select a *line*:

Input From Selection Grid

i. Click the line button from the feature selection grid.
ii. Take one hit at each end of the line feature.
iii. Edit the parameters for the line if needed.
iv. Click **Next** or press the

YELLOW machine button.

See the "Measured Line" topic for more detailed information on measuring lines.

Input from Input Feature List (History Window)

Select the **Feature ID** or feature icon for the needed line feature from the **Input Feature List**.

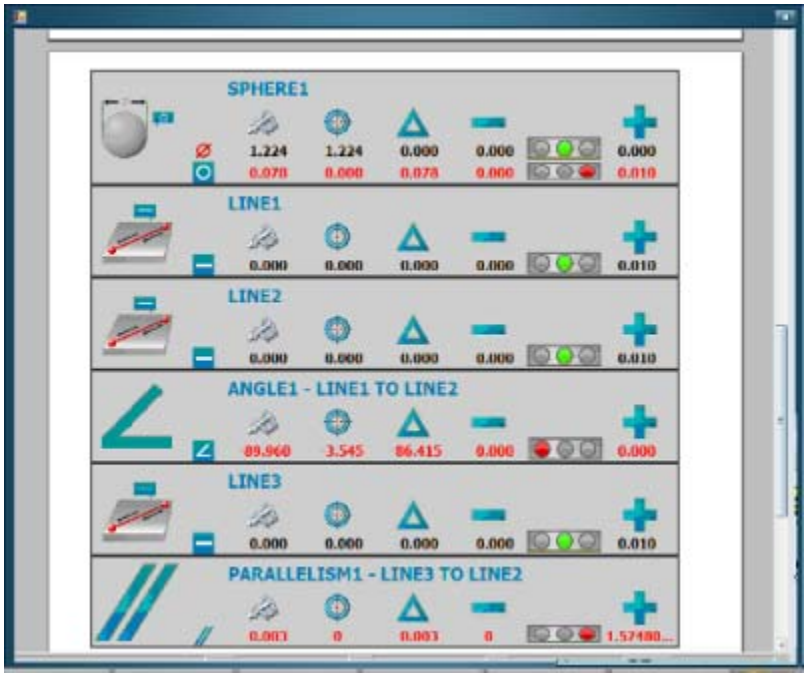
Note: Only possible input features will be displayed for selection.

5. Verify the plane-circle-circle alignment and click **Finish** or press the ***YELLOW*** machine button. The new alignment is added to the **History** window.

6. Select the check box next to the new distance dimension in the **History** window to toggle the display of the dimension in the report.

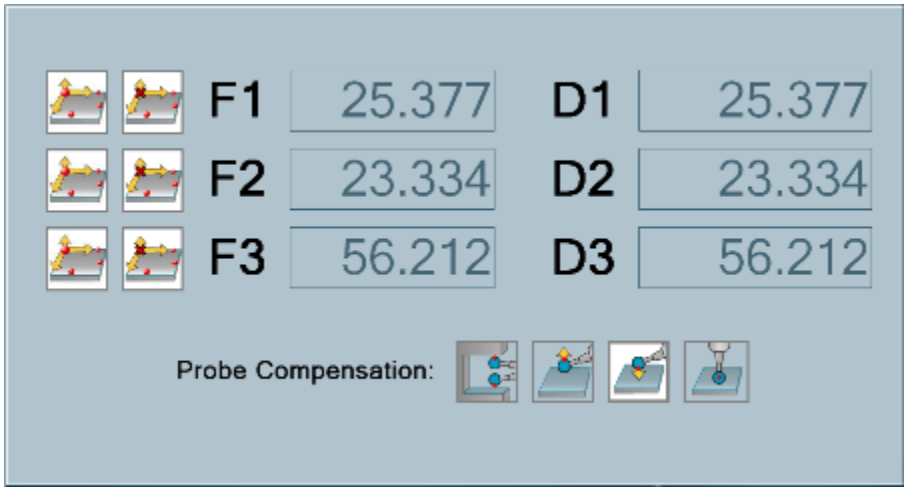
Viewing Measurement Results

The MultiGage Report Viewer allows you to review the results for your measurements. To open the viewer, click the **View Report** button on the **Utilities** toolbar. A report, similar to the example below will be displayed.



Example Report Viewer window

Using Gage Mode



Gage Mode interface

'Gage Mode' allows you to quickly measure distances in a manner similar to using a height gage. This is done by establishing 'base faces' from which measured points are evaluated.

Click **Gage** to open the **Digital Readouts** window (shown above).

You can use Gage Mode by doing the following:

- Setting Base Faces
- Setting Probe Compensation

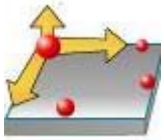
'Gage Mode' is similar to using a height gage with a few differences:

Height Gage	MultiGage
Measures up and down perpendicular to a surface	Allows you to 'zero' the measurement axis to any measured surface.
Measures in one or at most two axes.	Allows you to ‘zero’ to as many as three axes and report measurements to all three axes at one time.

Setting Base Faces

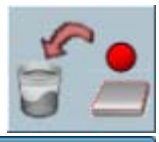
Before measuring points, you need to set the base face(s). The base face defines a flat surface from which distance values are given for measured points. Typically these faces will relate to the XYZ axes for the part being measured. For example, you might want to set each of the three faces by measuring planes on the part or work surface that coincide with XYZ axes.

To set and clear base faces:



1. Click **Set** to open the **Measure Base Face** interface for the **F1** face.

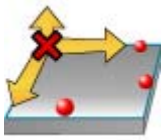
2. Using the ***RED*** machine button, take at least three hits to define the base face. As the hits are received, the MultiGage program updates the number of hits.



Note: If you are not satisfied with a hit, you can remove the last hit by clicking the **Remove Hit** button or by pressing the ***WHITE*** machine button.

3. Click **Finish** or press the ***YELLOW*** machine button to set the face definition. The **F1** readout will now update with the current probe distance from the face. You can now measure points. The **D1** distance value will display the distance of the measured point from the base face.

4. Repeat steps 1 through 3 for the second and third faces as needed. The **F2** and **F3** readouts will now update with the current probe distance from those faces. The **D2** and **D3** distance values will display the distance of the measured point from these base faces.



5. Click **Reset** to remove the set face definition. The readout for that face will not longer update values until you **Set** the face again.

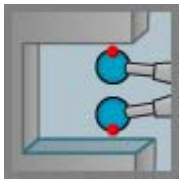
Setting Probe Compensation

Probe compensation controls how a measured point is reported in relation to the defined base face(s). The measured value of the center of the probe can be compensated by the radius of the probe tip either *away* or *toward* the face.

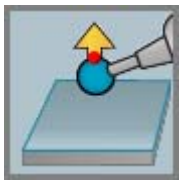
Click on the needed compensation method to make it active. The active method is highlighted with a white background.

The following probe compensation methods are available:

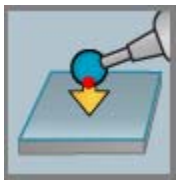
Method	Point Measurement



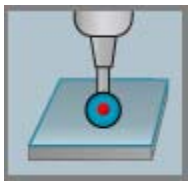
Auto - MultiGage will automatically determine which compensation method to apply to measured points.



Away - The measured value is adjusted *away* from the reference face by the probe tip radius.



Toward - The measured value is adjusted *toward* the reference face by the probe tip radius.



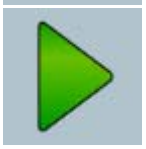
Off - The measured value is exactly at the center of the probe tip center in relation to the reference face.

Program Execution

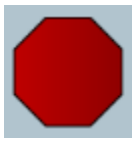
The MultiGage program allows you to execute commands in the order that they were included in your part program. By doing so, you can create a part program for all the needed features and then execute this part program to acquire measured results.

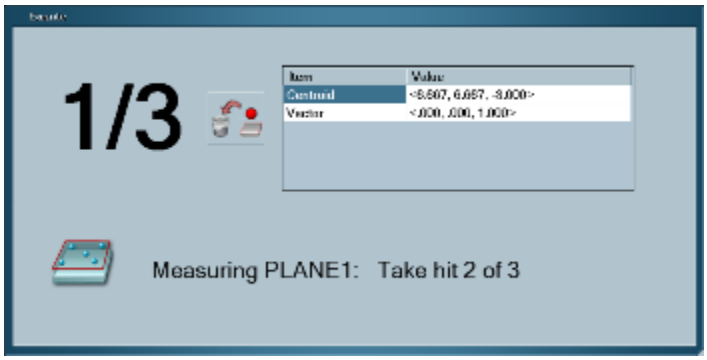
To execute part programs:

1. Create a part program by measuring and defining the parameters for each needed feature.
2. Save your part program.



3. Click **Execute Part** to begin execution. The **Execute** window is displayed and the **Execute Part** button changes to a **Stop** button. The **Stop** button allows you to stop execution at any point (which closes the **Execute** window).





QuickStart Execute interface

4. Follow the on-screen instructions and take hits for the specified measured feature. For each feature, nominal information is displayed, such as the centroid location, feature vector, diameter, etc.

Item	Value
Centroid	<2.500, 2.500, -3.000>
Vector	<.000, .000, 1.000>
Diameter	1.07

Nominal Information for a Measured Circle

Note: If you are not satisfied with a hit, you can remove the last hit by clicking the **Remove Hit** button or by pressing the ***WHITE*** machine button.

5. Once the needed number of hits have been taken, press the ***YELLOW*** machine button. The measured values are updated and the next feature is displayed.
6. Continue measuring each feature in the program as indicated in steps 4 and 5. When you press the ***YELLOW*** machine button for the last feature in your part program, the **Execute** window will close. As each feature is completed, the measured values are updated.

Appendix A: Calibrating Probes

To obtain accurate measurement results, each probe must be connected to the MultiGage machine and calibrated. Calibration data for each probe is stored in the memory on the machine.

The machine checks each time when a probe was changed. If the new probe has not previously been calibrated with this particular machine, the user will be prompted to proceed with the probe calibration procedure described below.

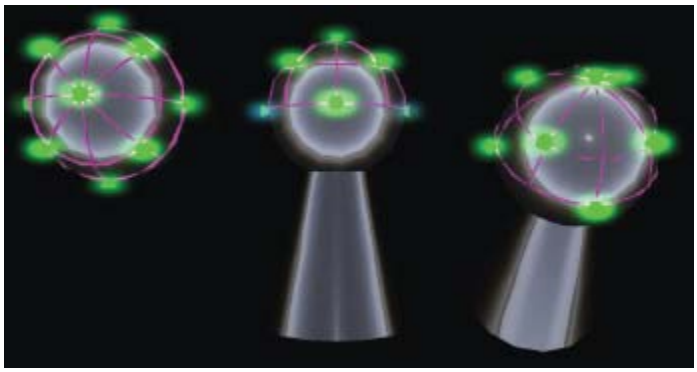
To calibrate a MultiGage probe:

1. Click **Calibrate** from the **Main** toolbar. This launches the WinRDS calibration tool which will guide you through the calibration process.
2. Place the *Reference Sphere* on the work surface so that the forearm and the wrist point straight down at the pole of the sphere, as shown below.

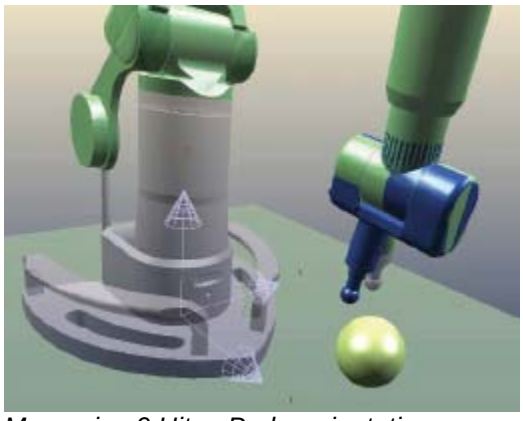
Placing the Reference Sphere

3. Type the Calibration sphere size, in millimeters, in the box provided. The size of the sphere should be noted on the stem of the artifact.
4. Click **OK** to continue. You will be prompted with "**Please insert reference probe**".
5. Attach the 15mm steel reference probe and click **OK**. See "Changing Probes".
6. Measure nine (9) points on the upper hemisphere of the reference sphere as shown below.

Note: the orientation of the wrist, the button side of the wrist should be pointing toward the base of the Arm as shown below.

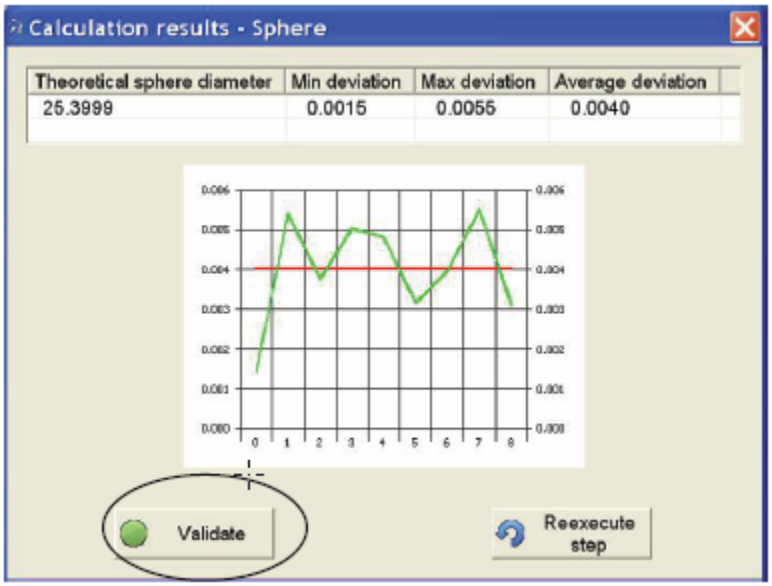


Measuring 9 Points - Measured hit locations



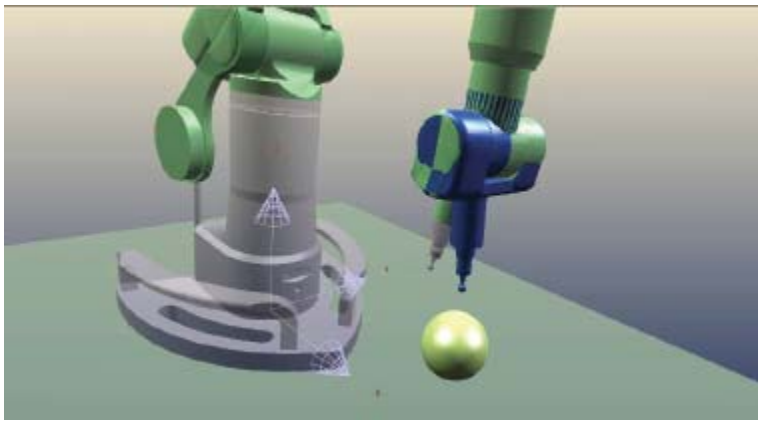
Measuring 9 Hits - Probe orientation

7. Once you have measured 9 points, the variation of the 9 points is displayed.



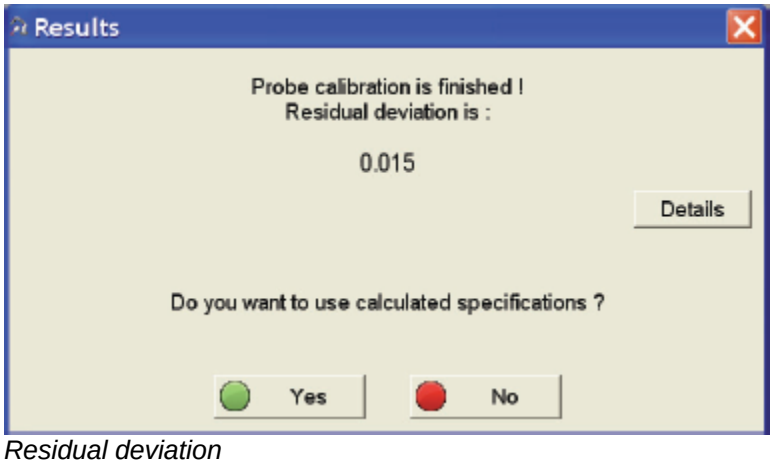
Calibration results

8. If there are errors or the result is not acceptable according the guidelines provided here, then click **Reexecute step**. If the results of the calibration are acceptable, then click **Validate**. You will be prompted with "**Please insert probe to calibrate**".
    - **Incidental errors** - A few measurements significantly higher or lower than the rest, could be the result of improper probing: i.e. trigger pressed when probe was not in contact with the calibration sphere, or too much pressure applied on the probe shaft.
    - **Systematic errors -** Consistent over or under size measurement, may be a result of the wrong parameters having been entered for the calibration sphere size. Restart the Probe Calibration and confirm that the correct sphere diameter was specified.
9. Remove the Reference Probe (15mm) unless you plan to calibrate the 15mm probe.
10. Attach the probe to be calibrated and click **OK**.
11. Using the new probe, measure the pattern of nine (9) points indicated in step 6 with four (4) different probe orientations. Each orientation represents a 90 degree twist of the wrist. Please follow closely the needed wrist orientation and point location as it is displayed on the screen. A total of 36 points are needed.



Follow the wrist orientation and location for 36 measured points

12. Once you have measured all 36 points, the variation of the 36 points is displayed. Review the results based on the same guidelines used for the reference probe (step 8).
13. If there are errors or the result is not acceptable, then click **Reexecute step**. If the results of the calibration are acceptable, then click **Validate**.
14. The system will take some time to compensate for probe deviation from nominal. The residual deviation is displayed.



Residual deviation

15. If the result is acceptable, click **Yes**. The calculated specifications are saved to the MultiGage machine's memory.

[Content for this topic comes from the MultiGage Portable CMM Setup Guide]

Appendix B: RDS Control Panel

The **RDS Control Panel** allows you view and configure settings for your MultiGage machine. When changes are made, the **Save** button appears, allowing you to modify your settings. Click **Quit** to exit without saving. The following pages are used to configure your machine:

- **Summary** - Displays machine information that has been read from your MultiGage machine.
- **Connection** - For the MultiGage machine you should use these default settings:

    - **Arm Type** - NCA
    - **Link Type** - USB
- **General Parameters**
    - **Global volume** - Adjust the volume for sounds from the machine.
    - **Language** - Will change the language for the RDS Control Panel application.
- **Probe -** You must be connected to the MultiGage machine to get this information. This page displays specific data for the currently attached probe. You can also specify:
    - **Automatic probe detection** - RDS will automatically determine which probe is connected.
    - **Force probe selection** - With this option selected, you can chose the needed probe from the list.
- **Features -** Displays available features that you can enable or disable as needed. This might include things such as WiFi, etc.
- **Advanced Settings**
    - **Enable Stops -** When this option is enabled, rotating a rotary axis past the mechanical stop will produce an error.
    - **Manage long press on arm buttons** - This option must be enabled to scan multiple hits or erase multiple hits when the associated button is held for more than one second. See "Arm Buttons".
    - **Remote mouse available** - This enables the use of 'Mouse Mode' with your machine. See "Mouse Mode".
    - **Auto suppress remote mouse if no movement** - 'Mouse Mode' will be disabled if there is no movement of your machine.
    - **Arm Button (0-3) function** - Allows you assign arm buttons to standard mouse mode buttons while in 'Mouse Mode'.
    - **Restore default parameters** - Will reset all options back to the default settings.
- **About** - Displays the current version for the RDS Control Panel

Glossary

C

Centroid: A single measured point that is part of a measured feature.

Characteristic: Each feature type has certain characteristics that define the feature's nominals and form. These characteristics are used when calculating dimensions and constructing points. A point, line or plane characteristic can be derived from a feature's measured position.

D

DATUM: A reference feature from which measurements are made. The DATUM location is known and reliable.

I

Intersection: The location where input features cross for the creation of a related feature. For example, two points might intersect to create a point.

R

Reference Plane: Provides the plane that coincides with the feature's nominal values.

S

Step: Part of a task that requires either measurement or other input.

T

Task: Includes a number of steps that need to be completed to accomplish the creation of a feature.

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