MICROSCOPY + METROLOGY SERVICES
Swisse made

RYF AG
Bettlachstrasse 2
CH-2540 Grenchen
Tel +41 32 654 21 00

RYF SA Succursale
Route de Genève 9c
CH-1291 Commugny
Tel +41 22 776 82 28

RYF AG (Zürich)
Bahnhofplatz 17
CH-8400 Winterthur
Tel +41 52 560 22 25

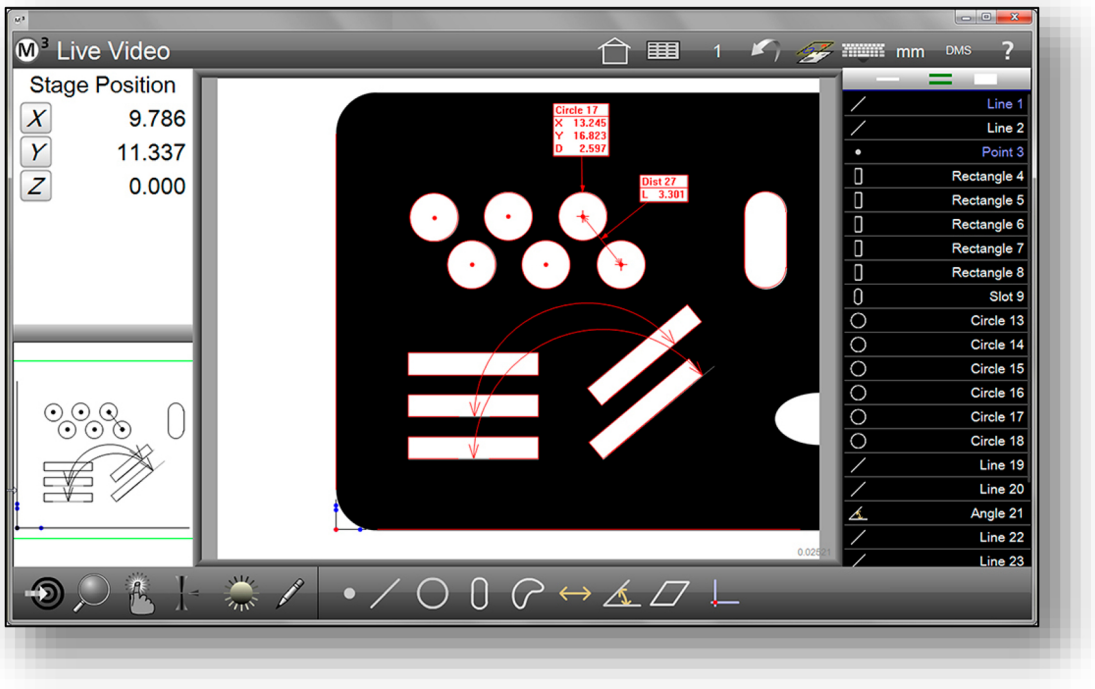
ryfag@ryfag.ch

ryf ag
Wir machen Qualität sichtbar
Nous rendons la qualité visible
Making quality visible

www.ryfag.ch

M3

Version 3



Video and FOV Inspection Software

User's Guide

October, 2018 PN# 15008-00 Revision 5

M3 Version 3
Video Inspection Software
User's Guide

Crosshair, Video Edge Detection and Field of View System

ryf ag

Wir machen Qualität sichtbar

ryfag.ch

M3 Version 3 Crosshair, Video Edge Detection and Field of View System User's Guide

Published by:
Metlogix Incorporated
175 Canal Street
Manchester, NH 03101 USA

User's Guide part number: 15008-00 Revision 5
Publishing date: October, 2018
Printed in United States of America
Copyright © 2014 by Metlogix, Incorporated., Manchester, New Hampshire

M3 software version: 3.00.00

All information set forth in this document, all rights to such information, any and all inventions disclosed herein and any patents that might be granted by employing the materials, methods, techniques or apparatus described herein are the exclusive property of Metlogix, Incorporated., Manchester, New Hampshire.

Terms, conditions and features referenced in this document are subject to change without notice.

No part of this document may be reproduced, stored in a retrieval system, or transmitted in any form or by any means, electronic, mechanical, photocopying, recording, or otherwise, without prior written permission of Metlogix, Inc.. Requests to Metlogix, Inc. for permission should be addressed to the Customer Services Department, Metlogix, Incorporated. 175 Canal Street, Manchester, New Hampshire 03101.

Limit of liability and disclaimer of warranty

While this guide was prepared with great care, Metlogix makes no representations or warranties with respect to the accuracy or completeness of the contents of this book and specifically disclaims any implied warranties of merchantability or fitness for a particular purpose. The advice, methods and instructions contained in this book might not be suitable for your situation. When in doubt regarding suitability, you are encouraged to consult with a professional where appropriate. Metlogix shall not be liable for any loss of profit or any damages, including but not limited to special, incidental, consequential or other damages.

Trademarks

Metlogix, is a registered trademark of Metlogix, Incorporated. Windows is a registered trademark of the Microsoft Corporation in the United States and other countries.

Introduction

The M3 Version 3 Video Inspection Software User Guide describes the operation of the M3 precision measuring software for software versions 3.00.00 and above. The encoder-based systems support manual part positioning and feature measurement under user control, and CNC part positioning and automated measurement under program control. Field of view systems do not include encoders or support CNC operation, all measurements are performed in the video field of view. While many M3 functions are common to encoder-based and field of view systems, there are some important functions that are unique to one or the other of the two systems. The organization of information in this user guide is shown below. Touch probe support is provided as an option in some CNC enabled systems.

User guide organization

Section	Content	Systems
1	Overview	Encoder and Field of View
2	User Interface	Encoder and Field of View
3	Probes	Encoder and Field of View
4	Measuring, Constructing and Creating Features	Encoder and Field of View
5	Image Stitching	***Encoder systems only***
6	Feature List and Part View	Encoder and Field of View
7	Reference Frames	Encoder and Field of View
8	Tolerancing	Encoder and Field of View
9	Reporting Data	Encoder and Field of View
10	Exporting Data	Encoder and Field of View
11	Programming	***Encoder (includes CNC) and Field of View (no CNC)***
12	Field of View Functions	***Field of View Systems Only***

While it is likely that this Guide includes some material that doesn't apply to your specific M3 system, most concepts explained here can easily be applied to all M3 systems.

The concepts explained in this Guide are necessary for the operation of any M3 system. Operators and supervisors will find the contents invaluable in conducting measurements, programming automatic functions and reporting results. A few user setup functions are also described that will help users and supervisors customize the M3 measurement tools, user interface screens and report formats.

Prerequisites

Operators and supervisors are assumed to understand dimensional metrology theory and practice, and have a good working knowledge of the Microsoft Windows operating system use and conventions.

Conventions & Terms

The term "M3 system" refers to the M3 software and the measuring hardware connected to it.

Icons

This guide uses the following icons to highlight information:

WARNINGS

The raised hand icon warns of a situation or condition that can lead to personal injury or death. Do not proceed until the warning is read and thoroughly understood.

CAUTIONS & IMPORTANT INFORMATION

The exclamation point icon indicates important information regarding equipment operation or maintenance, or a situation or condition that can lead to equipment malfunction or damage. Do not proceed until the information is read and thoroughly understood.

NOTES

The question mark icon indicates important supplementary information about an activity or concept.

Menu path navigation

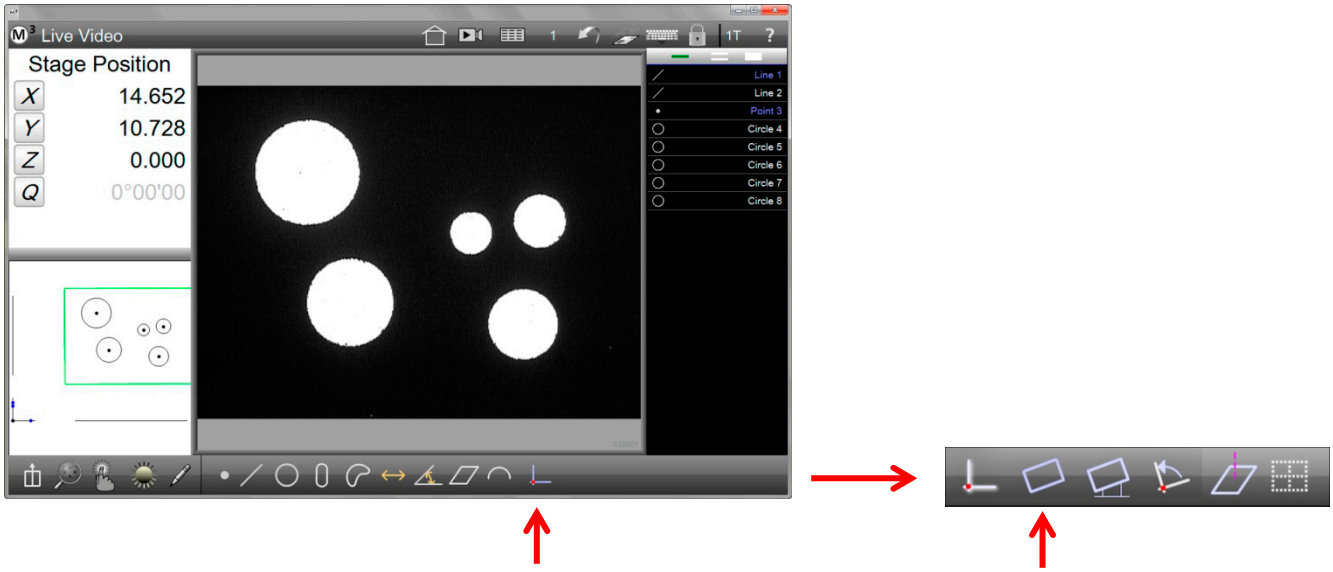
Many references are made in this guide to measurement tools that are displayed by touching or clicking the mouse cursor on a series of toolbar buttons and menu items. This kind of navigation path and the shorthand used to describe it is demonstrated in this example of displaying the Measure Skew button of the Datum menu, accessed through the Measure toolbar. The long form instruction to perform this operation is:

Press or click the Datum button of the Measure toolbar and then press or click the Skew button.

The shorthand that will be used in instructions to describe this operation is:

Measure toolbar > Datum button > Skew

A graphic representation of the operation is shown below.



NOTE

Toolbar buttons can be touched with a fingertip or clicked with a mouse cursor in M3 touchscreen systems. Instructions in this guide will use the term "press" to indicate a finger press or mouse click.

Getting Help

Help is available in this guide, in the electronic version of this guide accessed from the Help menu (question mark icon) of the System toolbar, from your Metlogix distributor or system provider and from our product support page at www.metlogix.com.

Should it become necessary to contact your Metlogix distributor or system provider, be prepared to supply the following information:

- The M3 software version number can be found in the About Settings screen accessed through the M3 System menu (M3 logo in the System toolbar).
  M3 System menu > Settings > About
- Information describing the measuring system hardware can be found in the Hardware Settings screen also accessed from the About Settings screen.
  M3 System menu > Settings > About
- A detailed description of the problem and steps that you've taken to resolve it

Contents

System Overview 1 1

User Interface 2 3

User Interaction 3

Viewports and Screen Organization 3

Screen Viewport Navigation 3

Live Video Screen 4

Measure Feature Screen 4

Part View Screen 5

Feature Detail Screen 6

Report (Run Results) Screen 6

Toolbars, Menus and Function Buttons 7

Measure Toolbar 7

Feature Measurement Menus 7

Change Feature Type Button 10

Part View Markup Menu 10

Part Lighting Controls 10

Autofocus Mode Button 10

Probe Menu 11

Zoom Menus 11

Goto Move Button 12

Extra Menu 12

System Toolbar 12

System menu 13

Main viewport indication 13

Home View Button 14

Freeze Live Video Images Button 14

Image Stitching Button 14

Report Button 15

Video Magnification Indication 15

Undo Button 15
Help Button 15
Report Toolbar 15
Edit Cells Button 15
Report Format Menu 15
Hide Row Button 15
Export Data Menu 16
Report Filters Menu 16
Import DXF Nominals 16
Statistics 16
Pivot View 16
Delete Run Results 16
Program Edit Toolbar 17
Play Button 17
Play From Here Button 17
Record Into Button 17
Re-measure Feature Button 17
Step and Repeat Button 17
Ask Button 17
Delete Program Step Button 17
Feature List 18

Probes 3 19

Crosshair Probes 19
Multipoint Video Probes 19
Touch Probes 19
Points Required for Single Point Probing 20
Points 20
Lines 20
Circles and Arcs 20
Slots and Rectangles 20
Blobs 20
Distances 20
Angles 20
Planes and Datum Planes 20

Spheres 21
Cylinders 21
Cones 21
Datum Point Features 21
Skew Features 21
Using Video Probes 21
Video Probe Scanning 21
Standard Crosshair Probe 22
Active Crosshair Probe 22
Vtouch Probe 22
MeasureLogic Probe 22
EyeMeasure Probe 23
Auto Point Entry 25
Auto Focus Mode 25
Auto Focus States 26
Focus Assist 26
Auto Focus Measurements 28
Setting the Number of Points Captured by Multipoint Probes 33
Using Video Probes Effectively 33
Video Probe Terms and Functions 34
Advanced Edge Teach 35
Inspection Example Images 38
Low Edge Contrast and Uniform Edge Form 38
Probe behavior in program playback 40
Pattern Recognition 41
Teaching a pattern 41
Datum Pattern Measurement 41
Standard Pattern Measurement 42
Pattern Recognition in Program Playback 42
Viewing the Image Source for Pattern Features 42
Using Touch Probes 43
Straight and Star Probes 43
Making Probe Contact 43
Touch Probe Setup Operations 43
Calibrating a Probe Tip 44
Calibrating using the Contact Probe Settings Screen 44

Calibrating using the Probe Teach Function 45
Adding a Probe Tip 45
Removing a Probe Tip 46
Editing a Probe Tip 46
Changing the probe Tip Name 47
Assigning a Tip Type 47
Designating the Probe as a Reference 47
Changing the Current Probe Tip 47
Configuring Probe Path Data 48

Measuring, Constructing, Creating Features 4.....49

Explicit and Automatic Measurement 49
Feature Constructions 49
Feature Creation 49
Preparing for Measurement Activities 50
Lighting Adjustment 50
Calibrating a Touch Probe Tip 50
Configuring (Touch) Probe Path Data 50
Teaching Video Edge Recognition 50
Establishing a Reference Frame 50
Part Leveling 51
Skew Alignment 51
Part Datum 52
Probing a Part Datum Point 52
Constructing a Skew and Part Datum Point from Parent Features 52
Skew 52
Datum Point 52
Creating a Datum by Manually Zeroing the X and Y Coefficients of a Feature 53
Probing and Measuring Features 54
Single Point Probing 54
Multipoint Probing 54
Probing Features 55
Probing Features with the Crosshair Probes 55
Probing Features with a Touch Probe 55
Probing Features with the Vtouch probe 55
Probing Features with the MeasureLogic Probe 56
Probing Features with the EyeMeasure Probe 56

Probing and Measuring Blobs 57
Repeat Mode Feature Measurements 58
Probing Features that Extend Beyond the Field of View 59
Multi-Measure Tool(Feature Decom)..... 60
Pattern Recognition 63
Teaching a pattern 63
Registration Pattern Measurement 64
Standard Pattern Measurement 64
Pattern Program Playback 65
Viewing the Image Source for Pattern Features 65
Goto Move Function 66
Enabling the Goto Move Function 66
Goto Position (Absolute/Relative) 66
Goto Feature 66
Goto Part View location 67
Constructing Features 68
Point Constructions 68
Circle Constructions 69
Arc Constructions 69
Line Constructions 69
Angle Constructions 70
Distance Constructions 70
Plane Constructions 70
Sphere Constructions 70
Cylinder and Cone Constructions 70
Feature Construction Procedures 70
Construction Procedures for Gauge Circle and Gauge Line 71
Gauge circle between two lines 71
Gauge circle between two arcs 71
Gauge line between two lines 71
Construction Procedures for all Other Features 72
Changing Constructed Feature Type 72
Creating Features 74
The Calc Feature(Formula Entry)..... 75

Image Stitching 5..... 78
Stitching a Super Image 78
Enabling Image stitching 78

Rectangular Image Stitching 78
Free Form Image Stitching 79
Preset Stage Stitching 80
Opening a DXF Overlay over a Stitched Image 81
Stitching a Super image Within a Part Program 82

Feature List, Feature Detail, Part View 6 83

The Feature List 83
Displaying Part Name and Run Numbers 84
Displaying Features 84
Scrolling the Feature List 84
Selecting features 84
Feature Finder 84
Deleting Features from the Feature List 86
Feature Detail screen 86
Part View screen 87
Accessing the Part View screen 88
Selecting Features in the Part View screen 88
Zooming and Panning 89
Adding Feature Annotations 90
Adding Box Markups 90
Adding Text Markups 90
Enhanced Data Cloud Display 91
3D Viewing 95
Displaying 3D Part Views 95
Rotation and elevation view controls 95
Preset 3D Views 96
The Part View Overlay 97

Reference Frames 7 99

Establishing a Reference Frame 99
Part Leveling 100
Skew alignment 100
Datum Creation 101
Probing a Part Datum Point 101
Constructing a Part Datum Point from one or more Parent Features 101

Creating a Datum by Manually Zeroing the X and Y Coefficients of a Feature 101
Rotating and Offsetting a Reference Frame 102
Rotating the Measurement Reference Frame 102
Offsetting the Skew Alignment 103
Managing Multiple Reference Frames 104
Unsaved Reference Frames 104
Saving Reference Frames 104
Locking and Unlocking Features from their Reference Frames 105

Tolerancing 8 107
Supported Tolerances 107
Applying Tolerances to a Single Feature 108
Keypad Entry of Nominal Values 109
Select Feature and go to the Nominal Feature Detail screen 109
Enter Nominal Values using the Keypad 109
Go to the Tolerance Screen, Select and Enter Tolerance 110
Go to the Deviation Screen to Observe Results 110
Entering Nominal Values using Auto-Round 111
Entering Nominal and Tolerance Values using Place Tolerancing 111
Importing Nominal Values from a DXF Drawing File 113
Applying Tolerances to a Group of Features 115
Quick Tolerance Functions 116
Configuring Preset Tolerance Values 116
Accessing the Quick Tolerance Functions 117
Interactive Mode 117
Program Edit Mode 117
Applying Quick Tolerances 118
Applying Nominal Values and Tolerances to Individual Coefficients 118
Applying Nominal Values and Tolerances to Multiple Feature Coefficients 119

Reporting Data and the Results Screen 9 121
Report Template Formats 121
Selecting a Report Format 121
Editing Report Templates 122
Accessing Report Template Editing Functions 123
..... 123

Displaying Report Edit Functions 124
Adding or Deleting Template Categories (Columns) 124
Changing Column Order 125
Changing Column Width 125
Increasing or Decreasing Horizontal Lines 125
Increasing or Decreasing Vertical Lines 126
Changing the Number of Feature Names for European Templates 126
Changing the Row Height 127
Resetting Template Modifications to the Factory Default 127
Discarding Changes Before Saving 127
Saving Template Changes as the New Default 127
User Defined Report Templates 127
Editing Report Cells 129
Hiding Report Data 130
Customizing Printed Reports 131
Specifying Printout Settings 131
Results View Visual Review Tools 132
Pivot view 133
Pinning Coefficient Data 133
Statistics 135
Results View Statistics 135
Pivot View Statistics 136
Run Zero Tab 136
Delete Tab 136
Statistics Result Feature 136

Exporting Data 10 140
Specifying Export Settings 140
Exporting a Data File 141
Direct Data Transfer to Excel 141

Programming 11 145
Part Fixturing Modes 146
No Part Fixturing 146
Temporary Part Fixturing 146
Permanent Part Fixturing 146

Creating a Part Program 147
Adjusting Probe Path Data 147
Inserting Goto Steps 148
Saving a Part Program 148
Loading a Program for Playback 149
Playing a Program 150
Playing Programs with Permanent Fixtures (CNC only) 150
Playing Programs with a Temporary or no Fixture 151
Editing programs 152
Initiate Edit Mode 152
Change Program Properties 153
Change CNC Probe Path Data 153
Edit Goto Moves 154
Change Lighting 154
Change Magnification 154
Change Feature Tolerances 154
Play Program 154
Play from Here 155
Add Program Steps 155
Enable or Disable CNC Clearance Moves 155
Re-measure Feature Steps 156
Add CNC Step and Repeat steps 156
Polar Mode Step and Repeat 156
Adding a User Message or Request Data 161
Adding User Messages and Images 161
Requesting User Data (Variables) 162
Deleting Program Steps 164
DXF Import and Auto Programming 164
Pallet Program Creation 165
Pallet Programming with SuperImages 166

Field of View (FOV) 12 171

M3 Measurement and Other Functions 171
Functions unique to field of view systems 171
Auto Play 171
Program Finder 171
Auto Play Use of the Program Finder 171

Verifying Auto Play Use of Program Finder 172
Motion Monitor 172
Operation Example 173
Measuring and Saving A Part Program 173
Playing a Part Program without Program Finder 174
Adding a Part to the Program Finder 175
Adding a Part Program with Auto Play 175
Adding a Part Program without Auto Play 176
Playing Programs using Program Finder and Auto Play 178
Playing a Program Without Auto Play 178
Playing a Program with Auto Play 179
Playing Programs to Measure Multiple Identical Parts 179
Editing Program Properties to Measure Multiple Parts 179
Playing the Program with Multiple Parts 181

Section 1: System Overview

The M3 video inspection software is advanced metrology software for performing two and three axis measurements in encoder-based systems and field of view systems at very high levels of precision and accuracy. Measurements can be performed using video comparators, measuring microscopes and a variety of video and touch probe measurement systems. Component inspection can be performed as part of prototype development, in-line production or final quality inspection. Part measuring programs are created automatically during the measurement process. Programs can be saved when measurements have been completed and played back later to measure identical parts. Measurement results from program runs can be displayed in a variety of customized report template views with a separate tab for each run. Result review tools display measurement data and statistics for each run or compare selected data across multiple runs. Feature detail windows display drawings that include point cloud distribution, data fit type, actual values, nominal values, deviations and tolerance results.

NOTE

M3 systems are provided as either encoder-based or field of view measuring systems. Only encoder-based systems support CNC functions. The touch probe option is available only for systems that include CNC support.

User interaction with the software can be through a conventional mouse interface or through the M3 MultiTouch Logic. The user interface is controlled by clicking items with a mouse or by touching the screen. MultiTouch Logic greatly increases ease of use by allowing pan and zoom of the live video and Part View displays with simple finger pinch, swipe and press actions.

During part feature measurement, feature points are entered manually or automatically using an advanced collection of video probes or touch probes. Probes include:

- **EyeMeasure:** This custom probe is created as the user's finger draws a path over an edge of the video part image on the screen. The EyeMeasure probe ensures the capture of points along complex part edges.
- **MeasureLogic probe:** The MeasureLogic probe incorporates pattern recognition to determine feature type and collect data points automatically with a single mouse click or finger press on a feature edge.
- **Vtouch probe:** The Vtouch probe moves the active crosshair to a clicked or pressed point on the live video window to collect a data point located on a part edge using video edge detection.
- **Pattern probe (optional):** The pattern probe teaches specific image patterns to the system for automatic detection during program playback.
- **Active crosshair:** The active crosshair collects a single point on a part edge at the center of the live video window or at a point clicked by the user using video edge detection.
- **Standard crosshair**: The standard crosshair collects a point at the intersection of the crosshair lines. The standard crosshair can be translated or rotated in the Field of View By pressing and dragging.
- **Touch probe (optional):** Touch probes capture a single point when contact is made with a part surface. Straight down and star probes can be used. While video probes typically probe part features in 2D space, touch probes probe features in 3D space. Touch probe support is available as an option for CNC enabled systems.

CNC-enabled systems include DXF auto-programming functions that convert DXF drawing files to part measurement programs automatically. Part pallet programs can be generated quickly by simply specifying pallet rows and columns. Step and repeat programs can be generated to measure a matrix of identical features on a single part.

CNC-enabled systems and Manual systems that include the Digital Comparator option and field of view systems can also be taught image patterns for automatic pattern recognition during program playback. Added support for rotated

pattern recognition results in more consistent playback in random fixturing scenarios, and speeds playback through the soft-fixture part positioning.

CNC focus mode measurements can be performed using any of the measuring probes in encoder-based systems equipped with the Auto Focus option. The focus mode is enabled, the measurement is initiated, the system focuses and measures automatically during program playback. Hardware frame latching and a digital camera interface provide extremely accurate and repeatable autofocus results.

Image stitching is included in CNC systems to scale and fit large parts within the field or view.

Industry-standard methods of stage and camera calibration are supported, including LEC, SLEC, NLEC, orthogonality, pixel size, camera skew and field of view.

Microsoft Windows 7, 8, or 10 provides the operating system basis of the functionality, performance and reliability necessary to address the wide variety of application requirements such as exporting data, printing reports and interfacing to third-party applications.

The software can be in one of four operating modes:

- Interactive: The software has just been launched and no program exists yet, or new a program is being recorded as measurements are made.
- Program: A complete program is loaded and features are displayed in the Feature list. If the program has not yet collected feature data in the playback mode, the Report screen displays the feature data from the original measuring/recording sequence. After the program has been played back and gathered new feature data, the measurement results are shown in numbered run tabs. The program can be played back again or edited by pressing tabs at the top of the Feature list.
- Playback: The part program is executed to perform repeated measurements of part features on identical parts. The various measurement, tolerance and other program activities are shown as the program is executed. Upon program completion, a new run tab is created in the result database showing the most recent measurement results and the software returns to the program mode. On-screen guidance is provided during playback that helps the user move from one feature to the next in non-CNC systems.
- Program Edit: The program edit mode is used to modify steps of a part program file. In the edit mode, program steps can be modified, deleted or added. Functions including re-measuring features, appending new measurement steps and changing system parameters such as light levels, camera magnifications and probe path data can be performed.
- Exit Part: The part file is exited to perform measurement, tolerance and other activities that have no impact on the part file. The part file is retained in memory and the playback mode can be reinitiated by simply pressing the play button.

Section 2: User Interface

The software user interface consists of menus, submenus and view ports that display different measurement windows and buttons that work in cooperation to perform and display measurements on a color LED monitor or touchscreen. The software interface is organized into basic functional areas that include:

- Three context-sensitive viewports
- The Feature List
- The System toolbar
- The Measure toolbar
- The Report toolbar
- The Program Edit toolbar

The software interface changes to add or delete items in support of specific system activities.

The viewports include one large viewport in the center of the screen and two smaller viewports on the left side of the screen. Viewport window contents change to support measurement and reporting activities.

The Feature List displays and provides access to all part features that have been measured, constructed or created.

The System, Measure, Report and Program Edit toolbars contain buttons that control overall measurement, reporting and the program edit functions. Toolbar buttons also provide access to submenus that initiate specific actions.

User Interaction

The user interface can be navigated and controlled by clicking a mouse cursor on the desired screen location or by pressing the desired location with a fingertip. Additional functions are accessed by long-clicking or long-pressing a screen location. Throughout the remainder of this guide, instructions and examples will use the term "press" to indicate either a mouse click or a finger press on the screen.

Viewports and Screen Organization

The organization of viewports and toolbars changes to support measurement activities that include measuring part features, constructing and creating features, applying tolerances, preparing and sending reports, exporting data and saving, playing or editing part program files. Some screens and toolbars are slightly different in encoder-based and field of view systems. A representative collection of screen organizations is shown in the following pages. Each example of screen organization shows a slightly different set of measurement and reporting functions. These functions will be described in detail in subsequent pages. Field of view functions are discussed in detail in Chapter 13: Field of View Functions.

Screen Viewport Navigation

Viewport or screen navigation is very straightforward and intuitive. Generally, screen navigation simply follows the usual workflow of measurement, application of tolerances, reporting or exporting results or editing a part program. The following screen images show six basic viewport configurations. The six screens include:

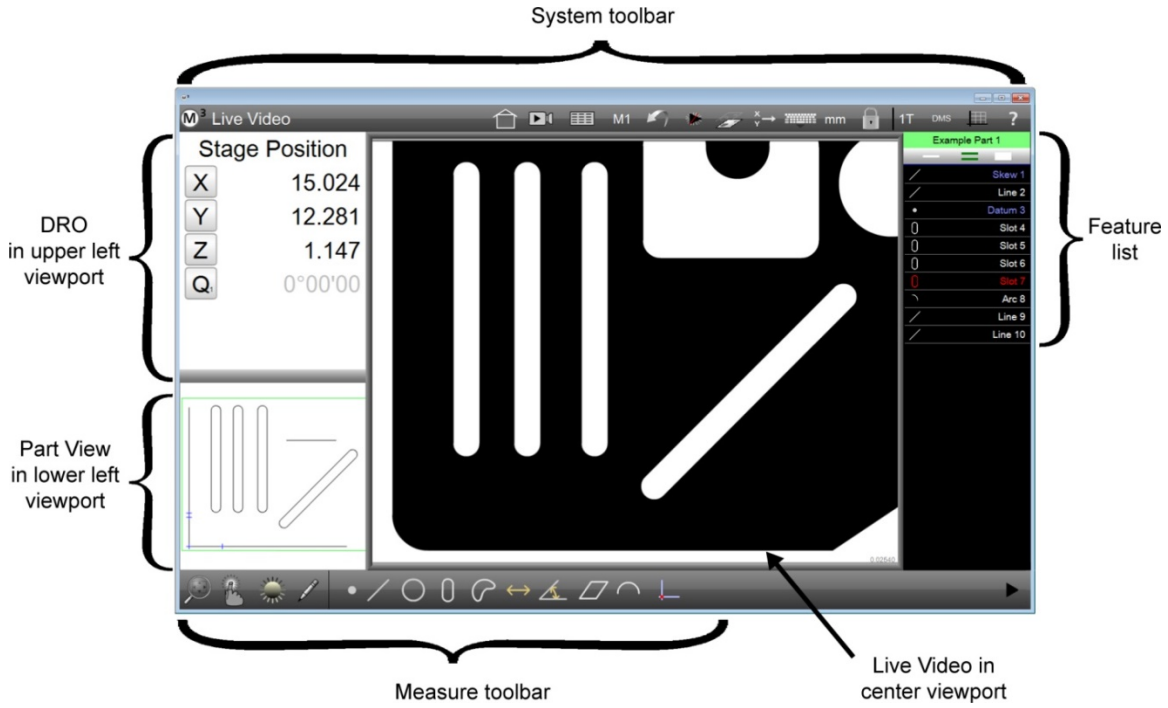
- Live Video (Home) screen
- Measure screen
- Part View screen with DRO in upper left viewport
- Part View screen with feature details in upper left viewport
- Part Detail screen
- Report screen

NOTE

Field of view functions are discussed in detail in Chapter 13: Field View Functions.

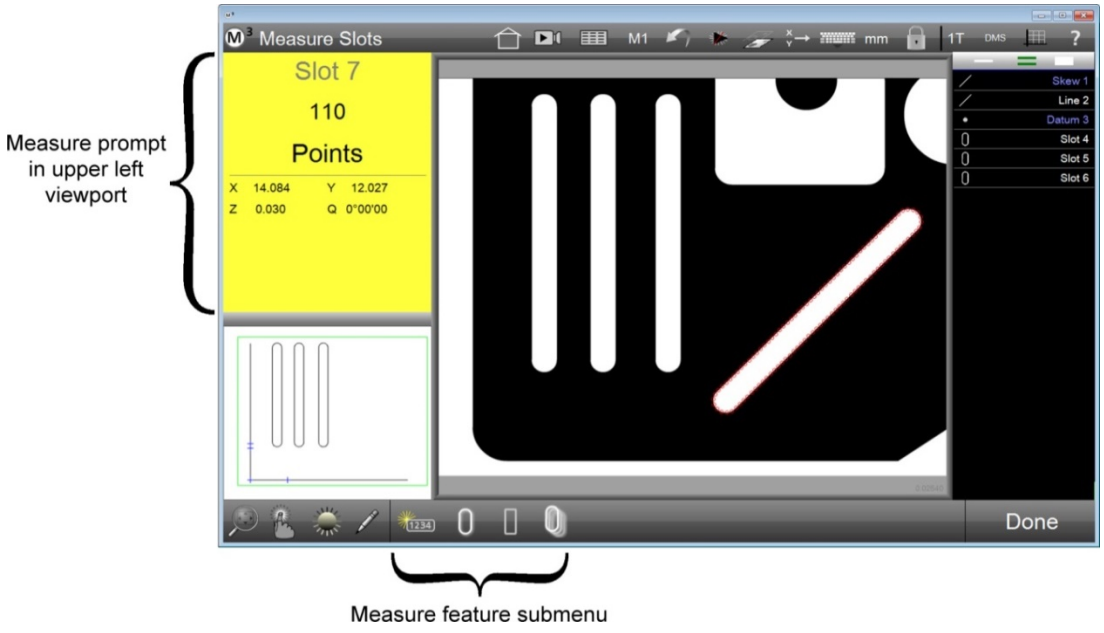
Live Video Screen

The Live Video screen displays the live video feed in the center viewport. Press the Home button to display the Live Video screen. This is the default Home screen.



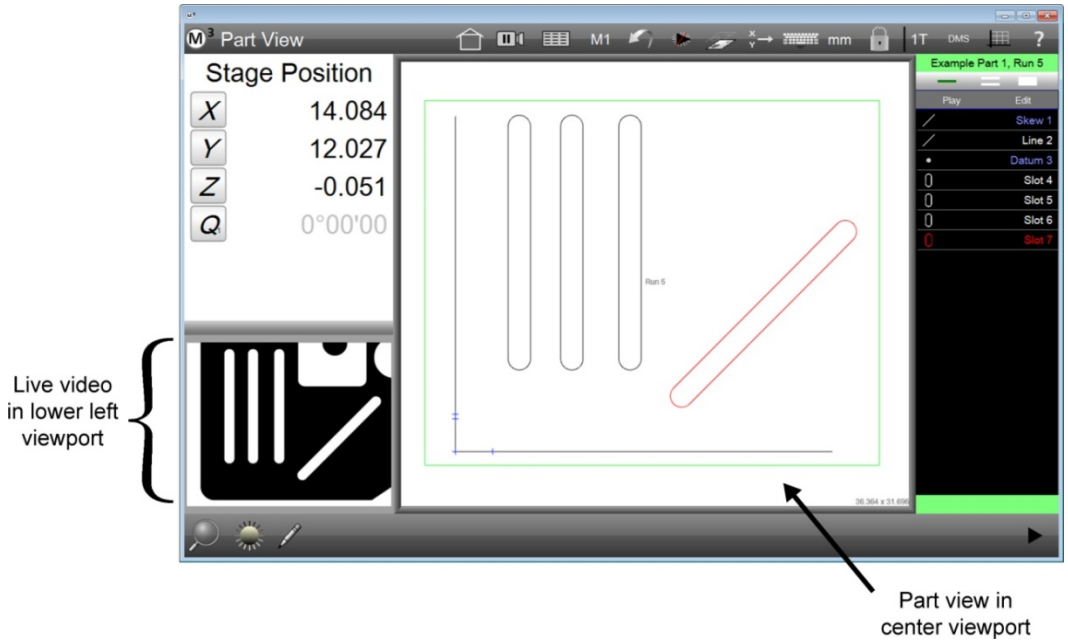
Measure Feature Screen

The Measure Feature screen is displayed as measurements are being performed. Acquired points are shown with DRO coordinates in the upper left viewport. Acquired points are shown in the Live Video image. Press any Measure Feature button in the Measure toolbar to display the Measure Feature screen. The Measure feature submenu for the selected feature measurement will be shown at the bottom of the screen.

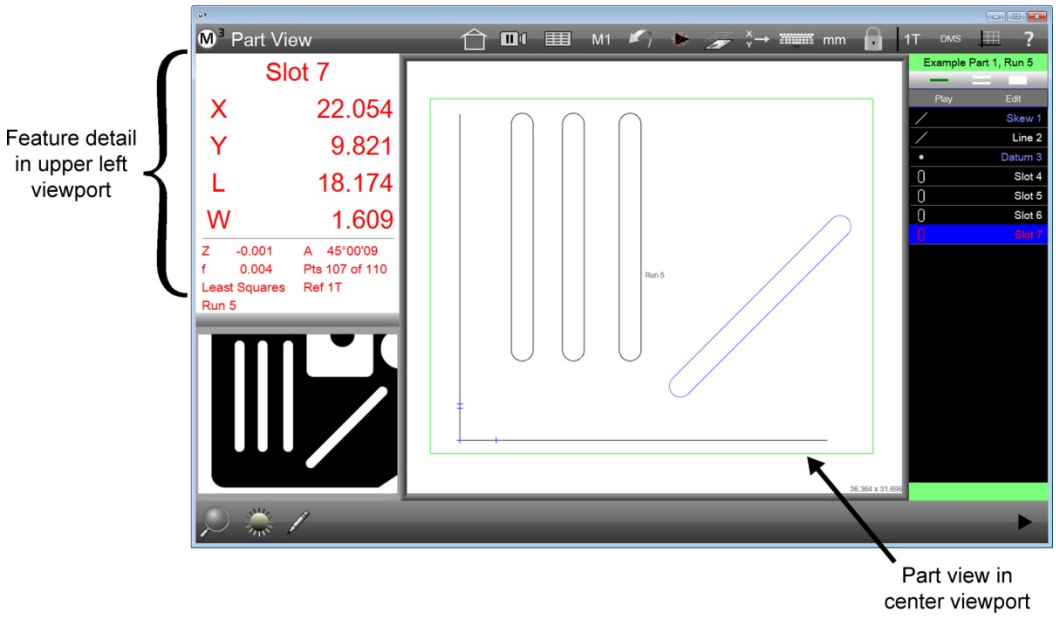


Part View Screen

The Part View screen displays a drawing of probed, constructed or created features and the measurement datum. The video field of view is highlighted by a green outline. Press in the small Part View image in the lower left viewport to display the Part View screen. The Live Video image will be moved to the lower left viewport.

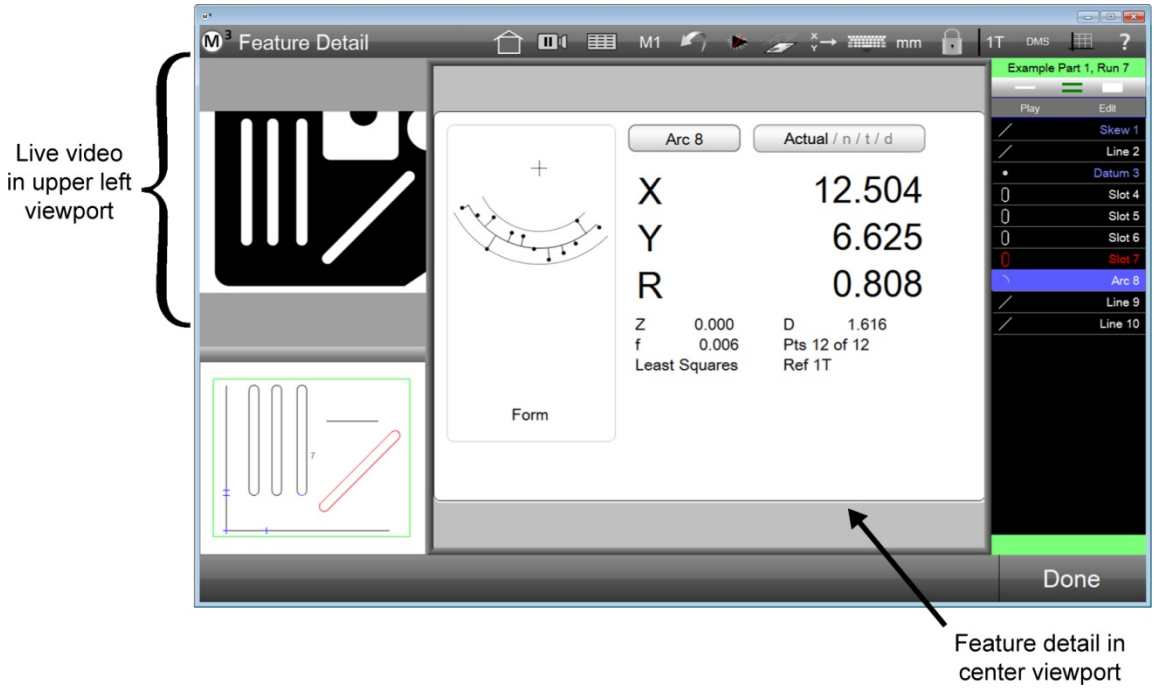


Feature detail information can be displayed in the upper left viewport of the Part View screen by selecting (pressing) a feature in the Feature List or on the Part View image. The selected feature will be shown by pulsating lines and highlighted in blue in the Feature List. Feature detail information includes position coordinates, feature geometric parameters such as diameter, form, number of probed points, the geometric fit type and the reference frame.



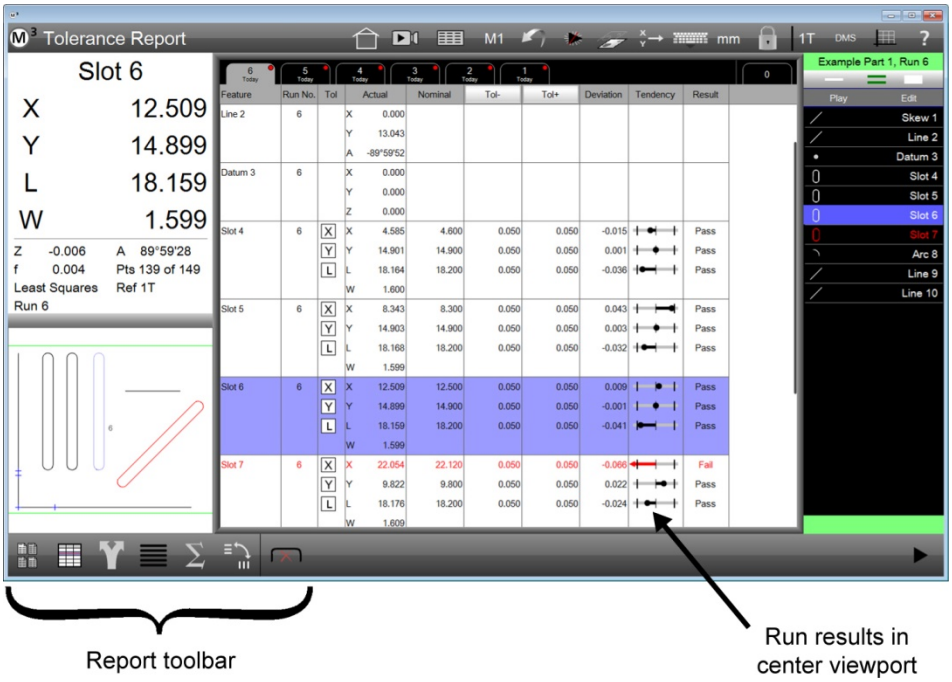
Feature Detail Screen

The Feature Detail screen displays the feature with probed points that passed and failed any assigned tolerances, and feature detail information including position coordinates, feature geometric parameters such as diameter, form, number of probed points, the geometric fit type and the reference frame. The geometric fit type can be changed and tolerances can be assigned, edited or deleted using tools provided by this screen. Press in the feature detail displayed in the upper left viewport (shown on the previous page) to display the Feature Detail screen.



Report (Run Results) Screen

The Report screen is used to format, edit and display reports of feature measurements and tolerance results. The Measure toolbar will be replaced by the Report toolbar. Program run data are shown in separate tabs. Press the Report button to display the Tolerance Report screen.



Toolbars, Menus and Function Buttons

Toolbars contain button icons that either perform an operation directly or display a menu of buttons that belong to a specific category of operation. The basic toolbars include:

- Measure toolbar
- System toolbar
- Report toolbar
- Program Edit toolbar

Measure Toolbar

The measure toolbar contains buttons that include:

1 Feature Measurement menus
2 Change Feature Type button
3 Part View Markup menu
4 Part Lighting controls
5 Autofocus Mode button (CNC enabled systems)
6 Probe menu
7 Zoom menu
8 Goto Move button (CNC enabled systems)
9 Extra menu

Feature Measurement Menus

Pressing any of the feature measurement menu buttons initiates a single feature measurement and displays a submenu of related feature measurement function buttons. Feature measurements will be discussed in detail in Section 4: Measuring, Constructing and Creating Features. Feature measurement menus include (from left to right):

- Points
- Lines
- Circles and Arcs
- Slots and Rectangles
- Blobs
- Distances
- Angles
- Planes
- Profiles (optional)
- Patterns
- Datum elements necessary to construct a measurement reference frame

Point menu

Press the Measure Point button to initiate a single point measurement and display the submenu of point measurement function buttons.

Point measurement function buttons include:

- Create point by entering data
- Probe a single point
- Probe a series of points

Line Menu

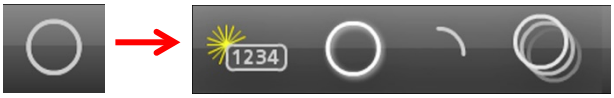
Press the Measure Line button to initiate a single line measurement and display the submenu of line measurement function buttons.

Line measurement function buttons include:

- Create Line by entering data
- Probe a single line
- Probe a series of lines

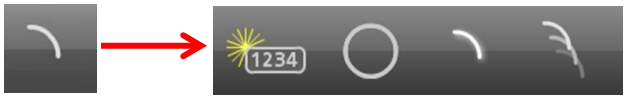
Circle and Arc Menu

Press the Measure Circle button to initiate a single circle measurement and display the submenu of circle and arc measurement function buttons.



Circle and Arc measurement function buttons include:

- Create circle by entering data
- Probe a single circle
- Initiate a single arc measurement and display arc measurement function buttons
- Probe a series of circles



Arc measurement function buttons include:

- Create arc by entering data
- Return to circle function buttons and measure circle
- Probe a single arc
- Probe a series of arcs

Slot and Rectangle Menu

Press the Measure Slot button to initiate a single slot measurement and display the menu of slot and rectangle measurement function buttons.

Slot and rectangle measurement function buttons include:

- Create slot by entering data
- Probe a single slot
- Initiate a single rectangle measurement and display rectangle measurement function buttons
- Probe a series of slots

Rectangle measurement function buttons include:

- Create rectangle by entering data
- Return to slot function buttons and measure slot
- Probe a single rectangle
- Probe a series of rectangles

Blob Menu

Press the Measure Blob button to initiate a single blob measurement and display the menu of Blob measurement function buttons. The Blob function must be enabled in the System Settings menu to be available. Please contact your Metlogix Representative for assistance if necessary.

Blob measurement function buttons include:

- Probe a single blob
- Probe a series of blobs

Distance Menu

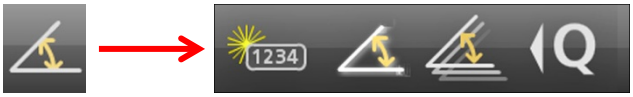
Press the Measure Distance button to initiate a single distance measurement and display the menu of Distance measurement function buttons.

Distance measurement function buttons include:

- Create Distance by entering data
- Probe a single distance
- Probe a series of distances

Angle Menu

Press the Measure Angle button to initiate a single angle measurement and display the menu of angle measurement function buttons.



Angle measurement function buttons include:

- Create angle by entering data
- Probe a single angle
- Probe a series of angles
- Measure a rotated crosshair

Plane Menu

Press the Measure Plane button to initiate a single plane measurement.

Profile Features Menu

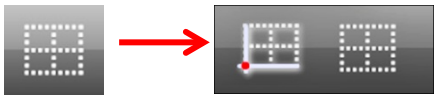
Press the Measure Profile Features button in systems that are equipped with the Digital Comparator option to construct profile features for digital overlay creation.

Datum Menu and Pattern Probe

Press the Measure Datum button to initiate a datum measurement and display the menu of Datum measurement function buttons used to create a measurement reference frame. Reference frames will be discussed in detail in Section 7: Reference Frames.

Datum measurement function buttons include:

- Probe or construct a datum point
- Probe or construct a part skew
- Create an offset skew
- Rotate the coordinate system around the datum point
- Probe or construct a datum plane
- Initiate a pattern teach and display the Pattern function buttons



Pattern function buttons include:

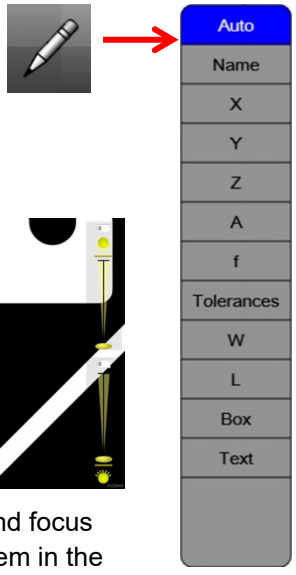
- Teach a datum pattern
- Teach a pattern

Change Feature Type Button

The Change Feature Type button changes the feature type of a feature selected in the Feature List or Part View when the feature can be defined by identical parameters. For example, a circle feature can be changed to an arc. The Change Feature button is not shown when there are no alternative feature types for the selected feature.

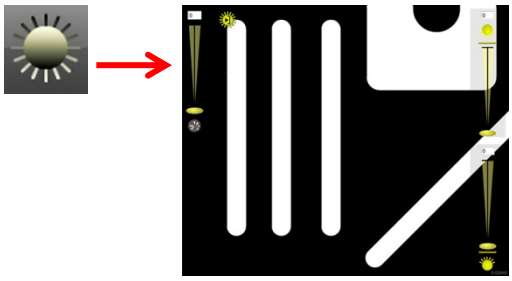
Part View Markup Menu

Press the Markup button to display the menu of Part View Markup tools. Markup tools are used to show feature parameters on the Part View drawing of a selected feature. Markups can also include squares and rectangles of selected colors and user notes. Markup functions will be discussed in detail in Section 6: Feature List and Part View Functions.



Part Lighting Controls

For systems that support Lighting Controls, press the Light Control button to superimpose the part lighting controls over the Live Video part image. Press again to toggle the display off.



Autofocus Mode Button

For systems that support Autofocus, press the Autofocus button to enable the focus assist and focus measure modes for a single measurement. Long-press the Autofocus button to lock the system in the autofocus mode. Press the button again to unlock the autofocus mode.

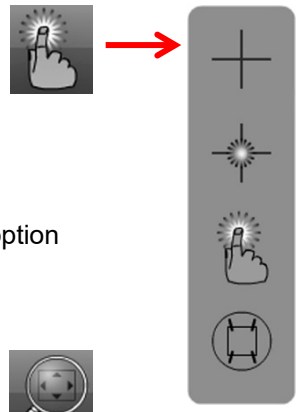
Autofocus OFF

Autofocus ON

Autofocus Locked

The Autofocus mode will be discussed in detail in Section 3: Probes.

Probe Menu



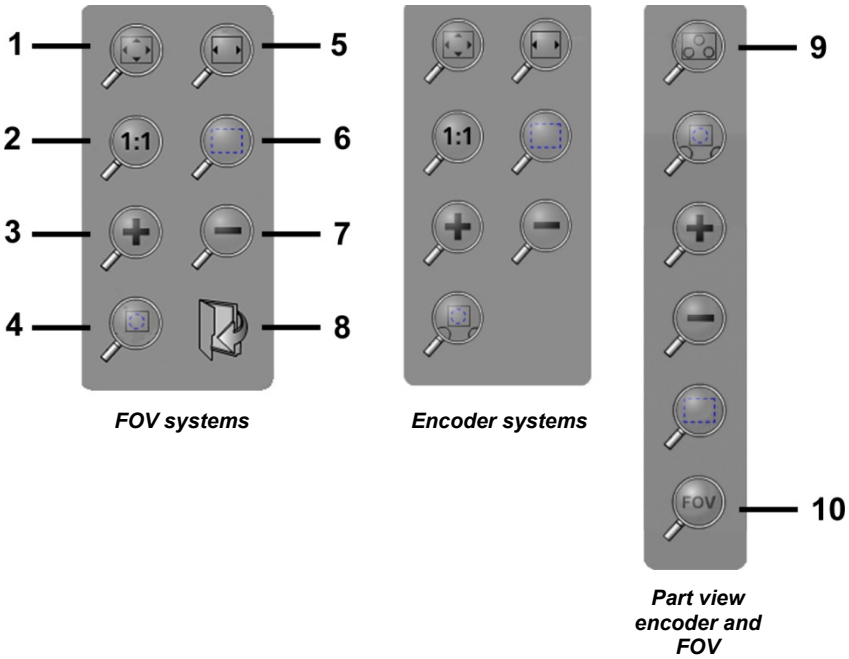
Press the Probe button to display a menu of measurement probes. Probe types will be discussed in detail in Section 3: Probes. Probe types include:

- Standard crosshair
- Active crosshair and Vtouch probe
- MeasureLogic probe and EyeMeasure probe
- DXF functions for systems that include the digital comparator option

Zoom Menus

Press the Zoom button to display the zoom magnification functions available for the Live Video or Part View in encoder systems and FOV systems. Zoom menu functions can include:

1 Zoom Fill: fits the video image horizontally and vertically
2 Zoom Fit: fits the video image horizontally
3 Zoom 1:1: sets one video pixel equal to one camera pixel
4 Zoom Marquee: fills video with user selected (press and drag) area
5 Zoom +: zooms in to increase magnification
6 Zoom -: zooms out to decrease magnification
7 Zoom Features: zooms to all features selected in the Feature List
8 Load part image into field of view (FOV systems)
9 Zoom all features: zooms to include all features
10 Zoom FOV: zooms the part view to the current field of view



FOV systems *Encoder systems* *Part view encoder and FOV*

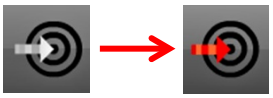
NOTE

Zoom magnifications can be changed using the mouse by clicking the mouse cursor in the live video window and moving the mouse scroll wheel forward or backward to increase or decrease the zoom magnifications.

Goto Move Button

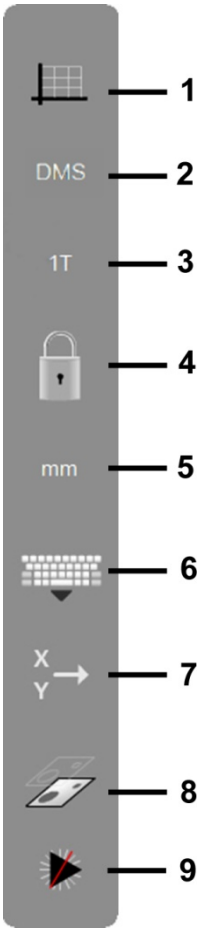
The Goto function prepares encoder-based CNC enabled systems to perform one of three CNC Goto moves including:

- Goto position
- Goto feature
- Goto Part View location



Press the Goto button to enable CNC Goto moves. The button arrow will turn red and the M3 screen will display green control borders around the interface areas where Goto commands can be executed. The Goto function is described in detail in Section 4: Measuring, Constructing and Creating Features.

Extra Menu



Press the Preferences button to display the utility functions specified as "Extra" in the Desktop Settings screen accessed from the System Menu. Utility functions can be hidden, displayed in the System toolbar or contained in the Extra menu. When no utilities are specified as "Extra" in the Desktop Settings screen, the Extra menu button is not shown in the Measure toolbar. All available utility functions are shown here (from top to bottom) in the Extra menu:

1 Toggles the displayed coordinate system between Polar and Cartesian
2 Toggles the display of angular units of measure between Degrees/Minutes/Seconds and Decimal Degrees
3 Selects the displayed reference frame
4 Locks or unlocks features to their measured reference frames
5 Toggles the display of linear units of measure between millimeters and inches
6 Displays the on-screen keyboard
7 Displays the Send menu for selecting the data to be transmitted to an RS232 device. The data choices change to support the selected feature. The Send functions are described in detail in Section 9: Reporting Data
8 Toggles the feature overlay function on or off. The feature overlay function is discussed in detail in Section 6: Feature List and Part View Functions
9 Toggles the Auto Play function on or off for FOV systems. The Auto Play function is discussed in detail in Section 12: Field of View Functions

System Toolbar

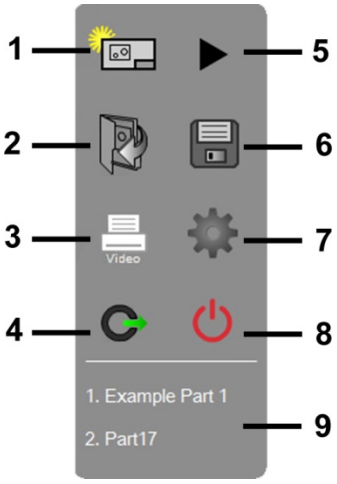
The System toolbar can be configured in the Desktop Settings screen to contain any or all of the utility functions described above in the ***Extra menu***. The minimum buttons contained in the System menu include:

1 System menu
2 Main viewport content indication
3 Home View button: The default home screen displays the Live video in the main viewport.
4 Freeze Live Video Images button (field of view enabled systems)
5 Image Stitching button
6 Report View button
7 Video magnification indication
8 Undo button
9 Help button

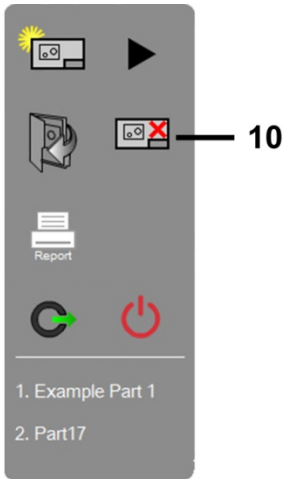
System menu

Press the System Menu button to display the system functions in the Interactive, Program playback and program edit modes. System functions can include:

1 New Part: clears the current part Feature List and part data in preparation for new part measurements
2 Open part file: display an open file dialog for opening a previously saved part file
3 Print: prints the contents of the main viewport (live video, part view, report)
4 User login: used to switch to a different user login account. User Account passwords and privileges are setup in the Security Settings screen.
5 Play program: initiates the playback of the current or loaded part program
6 Save program: displays a save part dialog for saving the current part file
7 Settings screens: provides access to the M3 settings screen menu
8 Exit M3: closes the M3 program
9 List of recent programs run
10 Exit Part: Closes program for non-program activities such as measurement or tolerance checks



Interactive mode



Results view in program playback mode

Program playback program editing

NOTE

The M3 does not save the current part upon closing. Be careful to save your current part file before exiting the software if you wish to retain your measurement session activities.

Main viewport indication

The indication will display the name of the current main viewport contents and may also provide useful information during program playback such as "Measure Circle":

- Live video
- Part View
- Report (Results view)
- Stage position

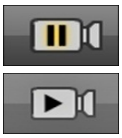
Home View Button

Press the Home View button to quickly display the home view screen. The Home View is typically the most used arrangement of viewports. The Home View is defined by specifying the main viewport contents in the Display Settings screen. The default home view is the live video. Display Settings screen content selections include:

- Video (live video)
- Part View
- Data (report)
- DRO (stage position)

Freeze Live Video Images Button

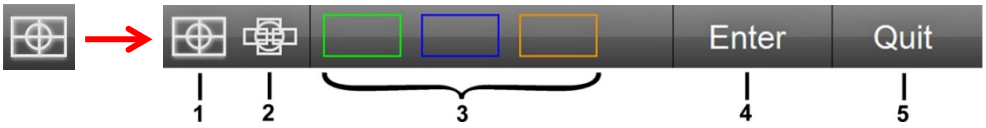
The digital comparator option provides the capability to quickly freeze and release a live video image. All features and functions remain available whether the camera image is static or live. To freeze a live video image:



- Press the pause image button to freeze the current Live Video window image in the field of view.
- Press the Play Image button to resume the live image feed.

Image Stitching Button

Press the Image Stitching button to display the Image Stitching menu. Image stitching will be discussed in detail in Section 5: Image Stitching.



The Image Stitching toolbar contains:

1 Rectangular Stitch
2 Free Form Stitch
3 Stage Preset Area
4 Enter Position
5 Quit Stitching Function

Rectangular Stitch Button

Press the Rectangular Stitch button to begin the image stitching process based on a predefined rectangular matrix of image areas.

Free Form Stitch Button

Press the Free Form Stitch button to begin the image stitching process based on arbitrary user selections of image areas.

Stage Preset Area Button

Press the Stage Preset Area button to begin the image stitching process based on arbitrary user selections of stage locations.

Enter Position Button

Press the Enter Position button to enter a stitch boundary position.

Quit Stitching Function Button

Press the Quit Stitching Function button to discard any stitching data and quit the image stitching function.

Report Button

Press the Report button to display the Results view of feature measurement and tolerance data in the report format selected in the Report Format menu of the Report toolbar. The Results view and Report toolbar will be discussed in detail in Section 9: Reporting Data and the Results Screen and after this discussion of the System toolbar.

Video Magnification Indication

The current system video magnification is displayed. This is also a menu that provides access to the other defined magnifications in the system. This indication is only displayed when more than one magnification is configured in the system.

Undo Button

Press the Undo button to erase the last software operation.

Help Button

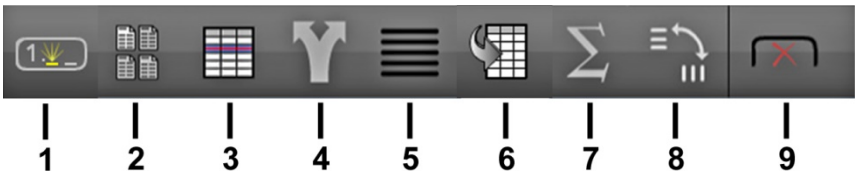
Press the Help button to display a hyperlinked version of this M3 User's Guide.

Report Toolbar

The Report toolbar is displayed when the Report button is pressed in the System toolbar to display the feature measurement Results View in the main viewport. The buttons displayed will depend on the current state of the measuring system. Reporting functions will be discussed in detail in Section 9: Reporting Data and the Results Screen.

The Report toolbar contains:

1. Edit Cell button
2. Report Format menu
3. Hide Row button
4. Export Data menu
5. Report Filters menu
6. Import DXF Nominals
7. Pivot view Statistics button
8. Pivot view button
9. Delete Run Results Tab button



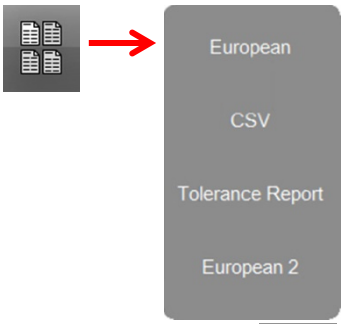
Edit Cells Button

Press the Edit Cells button to enable the alphanumeric editing of report cells. Editable cells will become highlighted in green.

Report Format Menu

Press the Report Format button to display the menu of report format types. Report formats include:

- European
- CSV
- Tolerance Report
- European 2



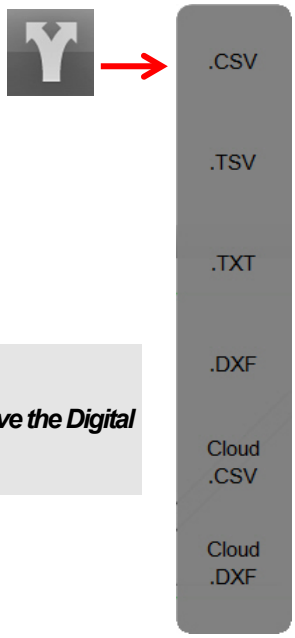
Hide Row Button

Press the Hide Row button to hide selected report rows in printed or exported report data. Long press the hide row button to hide all rows that are not currently selected.

Export Data Menu

Press the Export Data button to select an export format from the menu of export formats and export the report data. Data formats include:

- .CSV
- .TSV
- .TXT
- .DXF
- Cloud .CSV: includes all data points collected for the selected feature(s)
- Cloud .DXF (includes all data points collected for the selected feature(s)



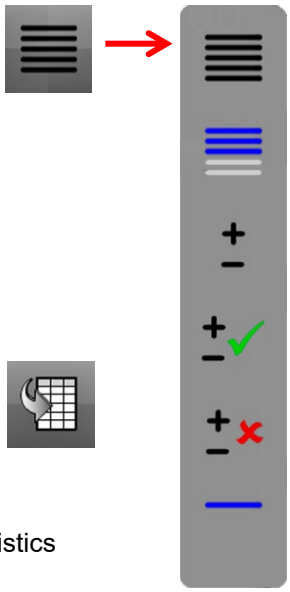
NOTE

The DXF Export option is only available on CNC systems or on manual systems that have the Digital Comparator option enabled.

Report Filters Menu

Press the Report Filters button to select a filter from Report Filters menu. Printed and exported reports will include only the filtered data. Filters include (from top to bottom):

- Filter All
- Filter Selected
- Filter Toleranced
- Filter Passed Tolerance
- Filter Failed Tolerance
- Filter Feature Type



Import DXF Nominals

Press the Import DXF Nominals button to display a file import dialog and specify a DXF drawing file from which part feature nominal tolerance values will be extracted. Importing DXF nominal values will be discussed in detail in Section 8: Tolerancing.

Statistics

Press the Statistics button to display the Statistics menu of standard statistics functions. Statistics functions will be discussed in detail in Section 9: Reporting Data and the Results Screen.

Pivot View

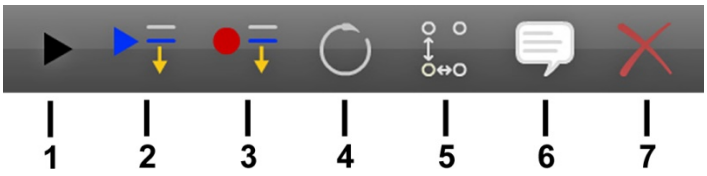
Press the Pivot button to display the selected features in a single row per run orientation. The Pivot function will be discussed in detail in Section 9: Reporting Data and the Results Screen.

Delete Run Results

Press the Delete Run Results button to delete the selected run data(tab) from the runs database.

Program Edit Toolbar

The Program Edit toolbar is displayed when an M3 program is placed in the edit mode. Program recording, playback and editing are discussed in detail in Section 11: Programming.



The Program Edit toolbar contains:

1 Play button
2 Play From Here button
3 Record Into button
4 Re-measure Feature button
5 Step and repeat button
6 Ask button
7 Delete Program Step button

Play Button

Press the Play button to reinitiate program playback after the program has been paused.

Play From Here Button

Press the Play From Here button to reinitiate program playback from the current program line location.

Record Into Button

Press the Record Into button to insert a new program step after the current location.

Re-measure Feature Button

Press the Re-measure Feature button to replace the feature measurement at the current location with new feature measurement.

Step and Repeat Button

Press the Step and Repeat button to specify a grid of repeat measurements or constructions for the selected feature in the edit mode.

Ask Button

Press the Ask button to insert a user-defined message, image or request variable data before the current program location.

Delete Program Step Button

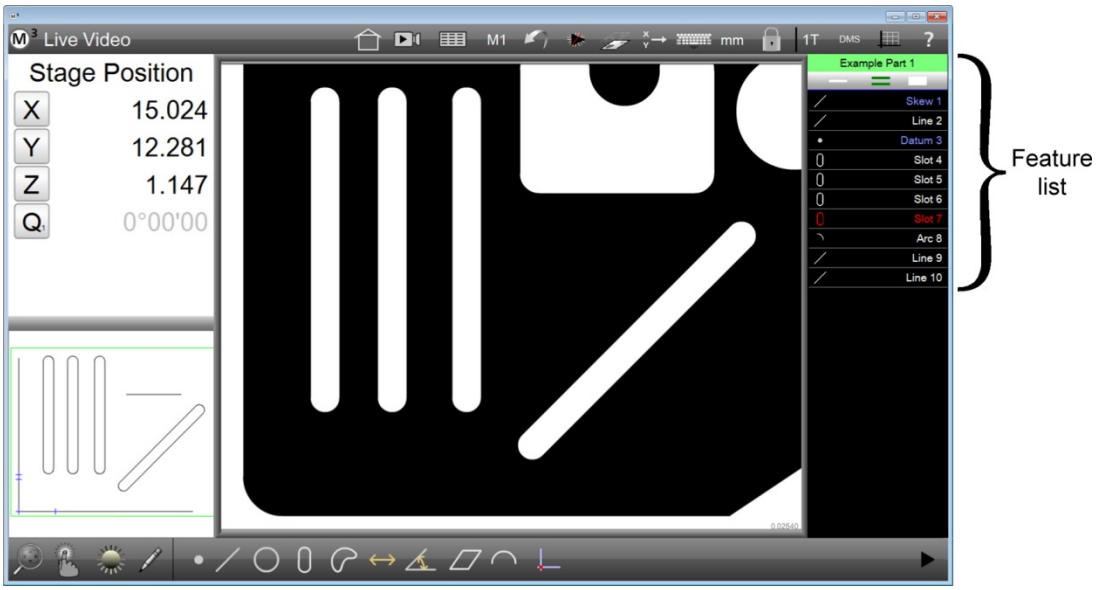
Press the Delete Feature Step button to delete the feature step at the current location.

NOTE

Various program steps can be deleted, such as feature measurements, light control, and magnification steps. However, program steps that are part of the datum creation cannot be deleted. Additionally, datum features cannot be re-measured or edited into a program with the Record Into function

Feature List

The Feature List is discussed in detail in Section 6: Feature List, feature Detail and Part View Functions.



Section 3: Probes

Features are measured by probing points along the edge of the feature within the live video window. Points are probed as part of either an explicit or automatic feature measurement using one of the video or touch probes. There are two main categories of video probes and two of touch probes:

- Crosshair video probes
- Multipoint video probes
- Straight down touch probes
- Star touch probes

NOTE
Touch probes are included only as an option on CNC systems.

Crosshair Probes

Crosshair probes capture a single point at a time. The simple crosshair probe requires that the operator manually line up the crosshair center with a point to be probed on the feature edge.

The active crosshair must still be aligned with the part by the user but the probe incorporates a relatively large area of edge detection and will detect the feature edge automatically, thereby increasing speed and accuracy.

Multipoint Video Probes

Two multipoint probes capture several points along an edge in a single probe firing. The MeasureLogic probe measures features with a single press of the fingertip or click of the mouse cursor on a feature edge. The MeasureLogic probe can automatically determine the feature type being measured eliminating the need to identify a feature type prior to probing. The EyeMeasure probe is drawn directly on an edge by the operator's fingertip or the mouse cursor and facilitates probing complex or partially obscured part edges. The number of points captured by multi-point probes can be set by the user and is discussed later in this section.

NOTE
Some M3 Systems support the Auto Focus mode which performs z-axis focus position calculations. Single point probes can be used in non-CNC manual zoom systems, and both the multipoint and single point probes can be used in motorized zoom CNC systems.

Touch Probes

Touch probes capture a single point at a time and are placed in contact with the part features manually as programs are recorded and then automatically as the programs are played back. While video probes typically probe part features in XY space, touch probes probe three-dimensional features in XYZ space.

Points Required for Single Point Probing

While multipoint probes collect many points each time they are fired, crosshairs and touch probes probe a single point at a time. The minimum number of points required for each feature type and recommendations regarding single point and multipoint probes are provided below.

Points

Require one or more points. When more than one point is probed, a least squares fit is applied to the data cloud.

Lines

Require two or more points. When more than two points are probed, the data fit type specified in the Feature Detail screen is applied to the feature's data cloud. Points should be evenly distributed along the line.

Circles and Arcs

Require three or more points. When more than three points are probed, the data fit type specified in the Feature Detail screen is applied to the feature's data cloud. Circle points should be evenly distributed around the circle. Arc points should begin at one end of the arc and finish at the other, with intermediate points evenly distributed along the arc.

Slots and Rectangles

Require five or more points. When more than five points are probed, the data fit type specified in the Feature Detail screen is applied to the feature's data cloud. Single point and multipoint probes may be used.

Blobs

Blobs are defined as irregular shapes, often characterized by a complex perimeter. The point density necessary to accurately measure a blob is much higher than the point density of the probes available in the probe menu and therefore blob measurements utilize a special tool to acquire points around the perimeter. The measurement of blobs is discussed in detail in Section 4: Measuring, Constructing and Creating Features.

Distances

Require two points.

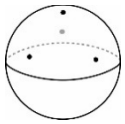
Angles

Require two or more points on each leg of the angle. Points are probed first on one leg and then on the other leg of the angle.

Planes and Datum Planes

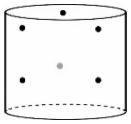
Require three or more points. When more than three points are probed, the data fit type specified in the Feature Detail screen is applied to the feature's data cloud. Probed points on a plane feature should be evenly distributed across the plane surface. Measurements of planes with greater than 15 degrees tilt are not permitted.

Spheres



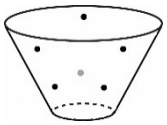
Require a minimum of four points evenly distributed around the sphere. One good pattern is three near the diameter and one at the top.

Cylinders



Require the first three evenly distributed around one Z elevation and the second three evenly distributed around a second Z elevation. Any number of points can be probed anywhere on the cylinder after the first six points have been probed.

Cones



Require the first three evenly distributed around one Z elevation and the second three evenly distributed around a second Z elevation. Any number of points can be probed anywhere on the cylinder after the first six points have been probed.

Datum Point Features

Require a minimum of one point. When more than one point is probed, a least squares fit is applied to the data cloud.

Skew Features

Require a minimum of two points. The skew feature is of fundamental importance to the creation of a reference frame, therefore many points should be probed along the skew line to ensure skew accuracy. The EyeMeasure probe or MeasureLogic probe is recommended for this measurement. When more than two points are probed, a least squares fit is applied to the data cloud.

Using Video Probes

Five video probe types are available for creating reference skews and datums and for measuring regular geometric features including points, lines, circles, arcs, slots, rectangles, blobs, distances, angles and planes. The five video probe types include:

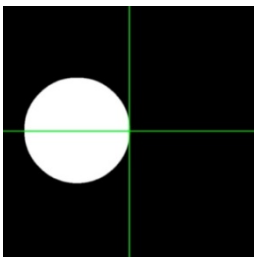
- Standard crosshair
- Simple Active crosshair
- Vtouch
- MeasureLogic
- EyeMeasure

Automatic point entry can be enabled for the standard and active crosshairs. Automatic point entry is discussed in detail later in this section.

Video Probe Scanning

When a video probe is fired one or more orthogonal scan lines cross the selected part edge bounded by light and dark areas of the screen image. Optimum scanning direction and color sensitivity for contrast recognition are set by the system for the active crosshair, Vtouch, and MeasureLogic probes. Optimum scanning direction and color sensitivity are also set by default for the EyeMeasure and Simple Active crosshair probes but can be changed if required. This topic is discussed in detail in the description of the EyeMeasure and Simple Active crosshair probes later in this section.

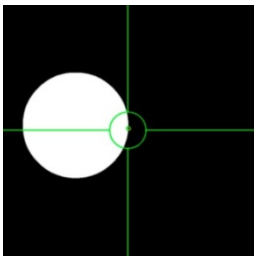
Standard Crosshair Probe



The standard crosshair consists of two lines crossed at 90 degrees. Single points are acquired manually or automatically (in CNC enabled systems) by positioning the crosshair intersection over a feature. To use the standard crosshair probe:

1. Select the standard crosshair probe from the Probe menu
   Measure toolbar > Probe menu > Standard crosshair probe
2. Position the crosshair probe over the part
3. Press the Enter button or press in the yellow measure prompt to fire the probe. A single point will be probed.
   Measure toolbar > Enter button
4. Probe the required number of points and then press the Done button to complete a feature measurement.
   Measure toolbar > Done button

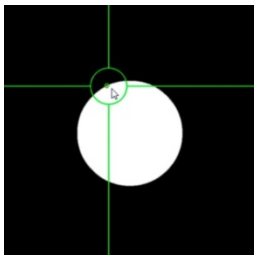
Simple Active Crosshair Probe



The simple active crosshair consists of a standard crosshair with a small circle at the center for positioning the edge detector. Single points are acquired manually or automatically (in CNC enabled systems) by positioning the crosshair circle over a feature. To use the active crosshair probe:

1. Select the active crosshair probe from the Probe menu.
   Measure toolbar > Probe menu > Active crosshair probe
2. Position the crosshair probe over the part .
3. Press the Enter button or press in the yellow measure prompt to fire the probe. A single point will be probed.
   Measure toolbar > Enter button
4. Probe the required number of points and then press the Done button to complete a feature measurement.
   Measure toolbar > Done button

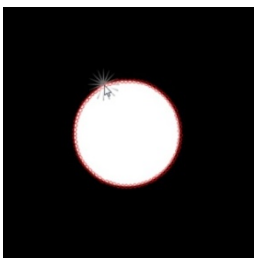
Vtouch Probe



The Vtouch probe is a special application of the active crosshair probe. When the active crosshair probe is selected, a single point is acquired on an edge that is pressed at a location on the live video window. To use the Vtouch probe:

1. Select the active crosshair probe from the Probe menu.
   Measure toolbar > Probe menu > Active crosshair probe
2. Move the stage to show the part feature of interest.
3. Press or click an edge of the part feature on the live video window screen. The probe will move momentarily to the touched point and a single point will be probed.
4. Probe the required number of points and then press the Done button to complete a feature measurement.
   Measure toolbar > Done button

MeasureLogic Probe



The MeasureLogic probe acquires many points when fired and automatically determines the feature type being measured eliminating the need to identify a feature type prior to probing. To use the MeasureLogic probe:

1. Select the MeasureLogic probe from the Probe menu.
   Measure toolbar > Probe menu > MeasureLogic probe
2. Move the stage to show the part feature of interest.
3. Press or click an edge of the desired part feature on the live video window screen to fire the probe.
4. Many points will be probed and the feature measurement will be completed automatically.

EyeMeasure Probe

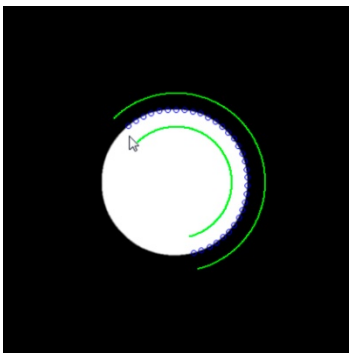
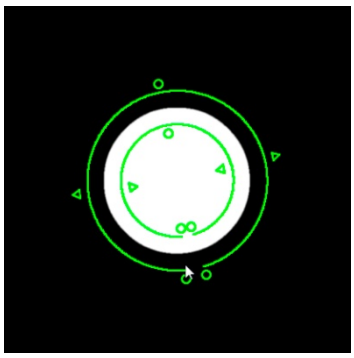
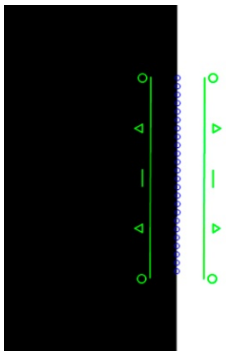
The EyeMeasure probe is a special application of the MeasureLogic probe. The EyeMeasure probe is drawn directly over an edge by the operator's fingertip or the mouse cursor and facilitates probing complex or partially obscured part edges. The EyeMeasure probe acquires many points when fired and automatically determines the feature type being measured eliminating the need to specify a feature type prior to probing. To use the EyeMeasure probe:

1 Select the MeasureLogic probe from the Probe menu.

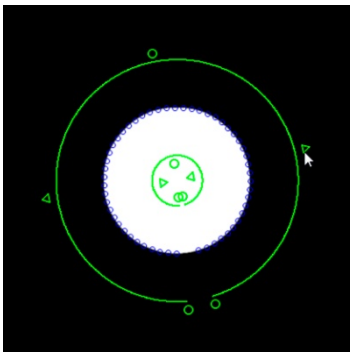
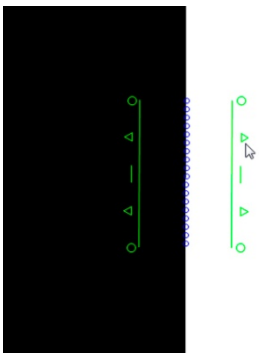
Measure toolbar > Probe menu > MeasureLogic probe

2 Move the stage to show the part feature of interest.

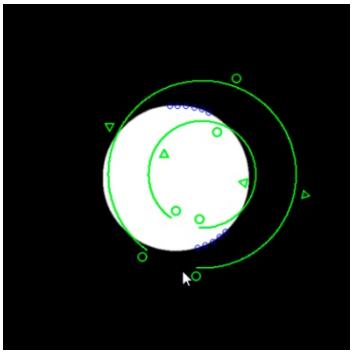
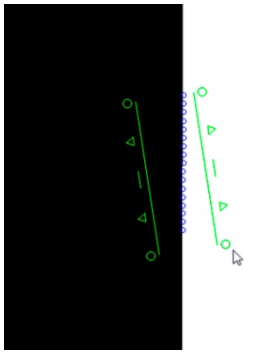
3 Press and drag along an edge of the desired part feature location on the live video window screen. In these examples, a circle and line are being probed.

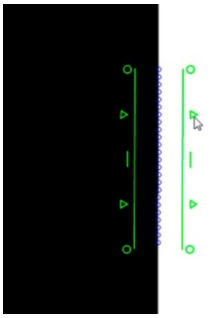
When the feature is adequately bounded by the probe, the probe size can be changed by pressing and dragging the probe triangle handles. In this example the edge sensing area of the probe is increased in size.

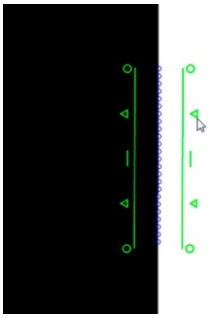



The probe can also be rotated about its initial starting point by pressing and dragging the end circle handle.

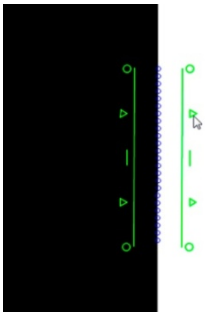



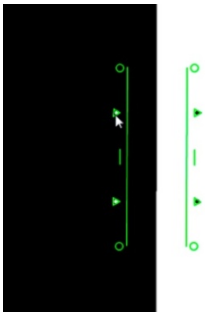
The scanning direction can be changed by pressing a triangle handle as shown in this example of the line probe. In the default orientation of the triangle handles, the triangles face away from each other indicating that the system will choose the optimum scan direction. In these examples, a triangle handle is pressed to set the scan direction from black to white, then the triangle handle is pressed again to set the scan direction from white to black. A third press on the triangle handle would return the scan direction to the default optimum.





The scan color contrast sensitivity can be changed by dragging one probe boundary across and over the other reversing the two boundary lines as shown in this example. The right boundary line of the probe was dragged across and over the left boundary line changing the color sensitivity as indicated by the white and black dots in the triangle handles.

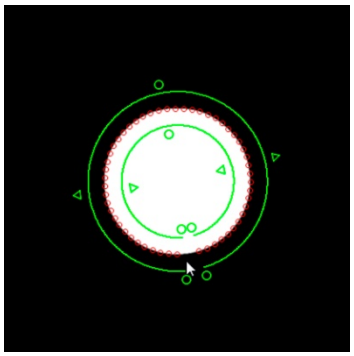


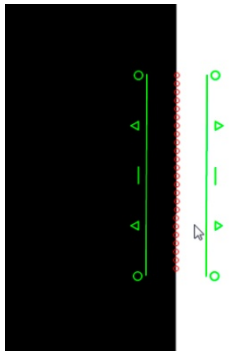


CAUTION

As mentioned earlier, the default scanning direction and color sensitivity of the EyeMeasure probe are optimum for feature measurements. Therefore, any changes in scanning direction or color sensitivity should be made only if absolutely necessary and by qualified personnel.

4 Press inside the probe boundaries to fire the probe. Many points will be probed and the measurement will be completed automatically.





NOTE

Pressing within the tool space will enter points for this probe.

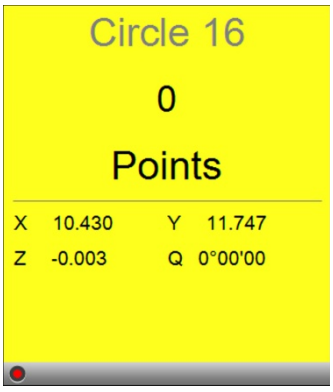
Auto Point Entry

Auto point entry can be enabled for the standard and active crosshair probes in the interactive mode and the program playback mode for non-CNC systems. When the auto point entry function is enabled, a point is collected by the system each time the stage is moved to position the part under a probe. The automatic point entry is delayed until the stage has been motionless for a short time to ensure that the probe is located where the user intended. The timer that controls this delay can be configured in the Auto Enter fields of the Measure Settings screen. The default value will provide adequate delay for most applications. To use the auto point entry function:

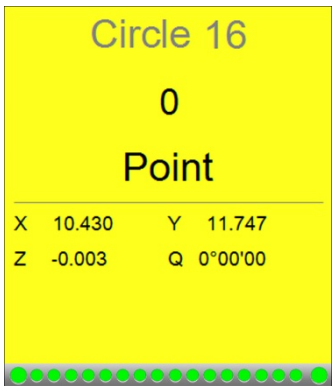
1. Select a crosshair probe from the probe menu.
   Measure toolbar > Probe menu > Standard or active crosshair probe
2. Select the desired feature to measure.
   Measure toolbar > Desired Measure Feature button
3. A small red dot will appear in the horizontal divider bar between the Part View and the measure prompt viewports. This red dot is the activation button for the auto point entry timer. Press the red dot to enable the timer.
4. Move the stage to position the feature under the probe. As you move, the dots will turn green indicating that the timer is armed.
5. When you stop moving the stage, the dots will turn yellow and begin to disappear to the left. Shortly before the point is probed the dots will turn red. A point will automatically be probed at the current crosshair position. When the point is entered, the dots will become black. When the stage is moved again, the timer will restart and green dots will be shown again.

NOTE

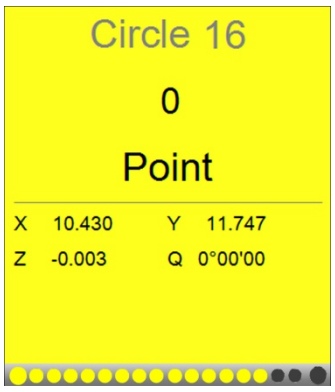
The stage can be moved to reposition the point to be probed while the timer is counting down and the dots are either green or yellow. When the dots are red, moving the stage will not restart the timer. If you accidentally enter an incorrect point position, simply press the Undo button in the system menu to remove the point.



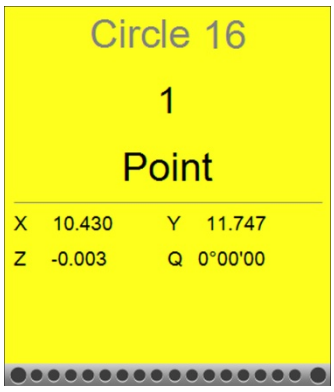
Activation red dot



Stage moving, timer armed



Stage stopped



Point entered

Auto Focus Mode

The Auto Focus mode is optional and not available in all M3 systems.

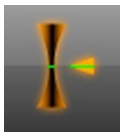
NOTE
If your system supports the Auto Focus mode, the Auto Focus button will be located in the left side of the measure toolbar.

Auto Focus States

The Auto Focus mode can be in one of three states:

- Disabled
- Enabled
- Enabled and locked

Disabled


Enabled

Enabled and locked

Press the Auto Focus button to toggle the function between disabled and enabled. Long-press the Auto Focus button to lock the function in the enabled state. Unlock and disable the Auto Focus mode with a single press.

The Auto Focus mode uses contrast sharpness recognition to determine the optimum focus as the z-axis is raised and lowered to sweep back and forth through the focus range of the video camera. The Auto Focus mode is used in manual and CNC enabled systems to perform two functions:

- Focus assist
- Focused feature measurement

Focus Assist

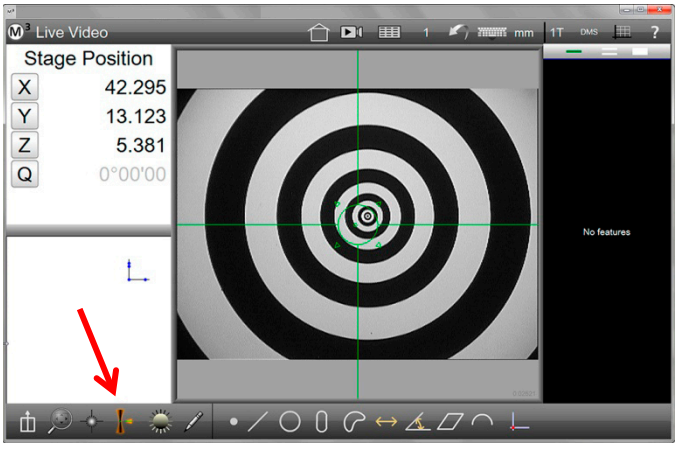
Focus assist determines the z-axis position of optimum part focus as the z-axis is moved up and down and the video camera system is swept back and forth through the focus region.

In manual systems, the user raises and lowers the z-axis. When the optimum point of focus has been determined by the system, the Stage Position indication in the measure prompt is changed to show only the z-axis focus target. The user then raises or lowers the z-axis to reduce the target value to zero at optimum focus. In CNC enabled systems, the system drives the z-axis to find and set the optimum focus point automatically.

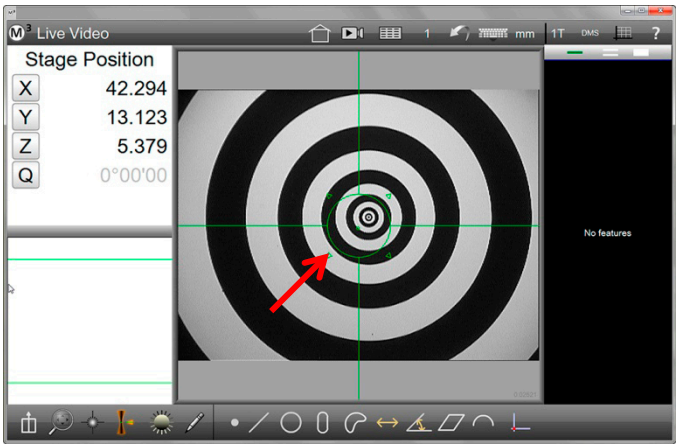
Focus Assist in Manual Systems

To use focus assist in a manual system:

1 Position the standard or active crosshair over the desired part edge.
2 Enable the Auto Focus mode.
Measure toolbar > Focus Mode button
3 Press and drag the triangle handles of the region of interest to change its size if desired.



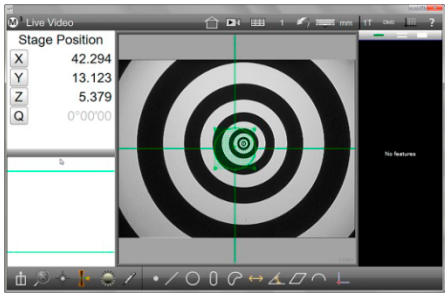
Enable Auto Focus



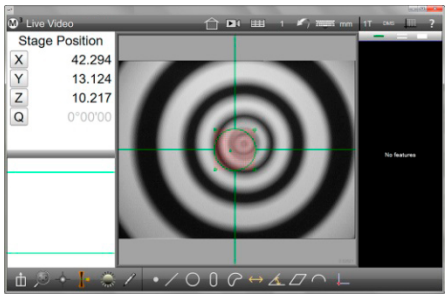
Change size if desired

4 Press in the region of interest to initiate auto focus

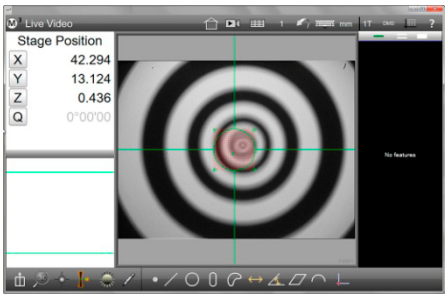
assist. A grid will appear.



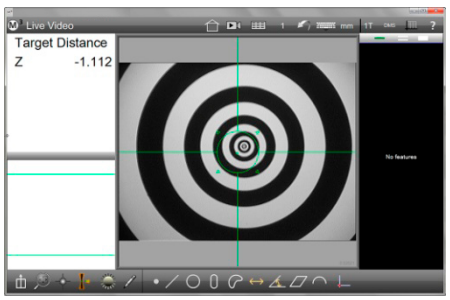
5 Raise and lower the z-axis through the range of focus until the Stage Position indicated in the measure prompt changes to indicate a Target position. The grid will be red when the system is out of focus, and green when the system is in focus.



Out of focus high

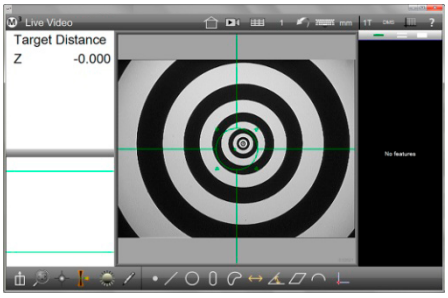


Out of focus low



Focus point found

6 Raise or lower the z-axis to reduce the Target value to zero at the optimum focus position.



Focus Assist in CNC Enabled Systems

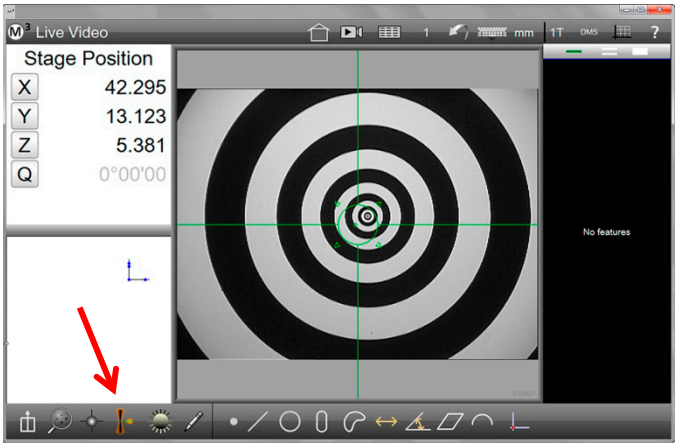
To use the focus assist in a CNC enabled system:

1 Position the standard or active crosshair over the desired part edge.

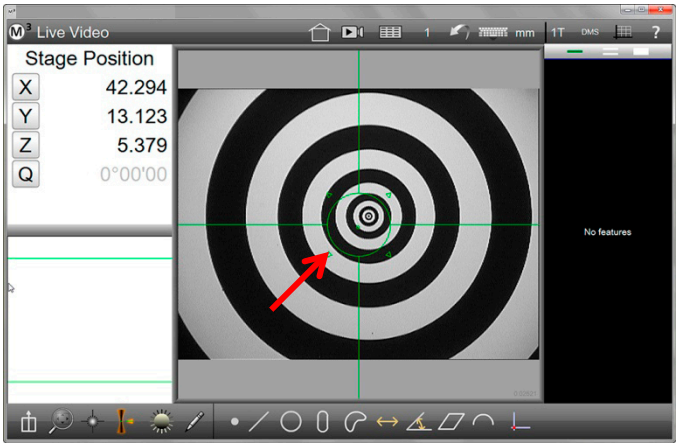
2 Enable the Auto Focus mode.

Measure toolbar > Focus Mode button

3 Press and drag the triangle handles of the region of interest to change the size if desired.



Enable Auto Focus



Change size if desired

4 Press in the region of interest to initiate auto focus assist. A grid will appear and the system will raise and lower the z-axis through the range of focus to find and set the optimum focus automatically.

Auto Focus Measurements

The Auto Focus mode can be used to conduct feature measurements in the optimum focal plane. When performing Auto Focus measurements, the system determines the z-axis position of optimum part focus before collecting feature data points.

In manual systems, the user sweeps back and forth through the range of focus by raising and lowering the z-axis. When the optimum point of focus has been found, a point at the center of the crosshair will be probed. Auto Focus points are probed using the standard or active crosshair in manual systems.

In CNC enabled systems, the system drives the z-axis to the optimum focus point automatically and any of the available probes can be used for autofocus measurements. In CNC enabled systems, and Auto Focus measurement steps are included in M3 part programs.

Auto Focus Measurements in Manual Systems

To perform Auto Focus measurements in a manual system:

1 Select the standard or active crosshair probe from the probe menu.

Measure toolbar > Probe menu > Standard or active crosshair probe

2 Enable Auto Focus (if it is not already enabled).

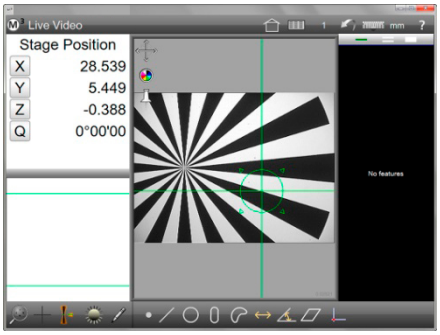
Measure toolbar > Auto Focus button

3 Position the crosshair probe over the desired feature edge and then press and drag the triangle handles of the region of interest to change its size if desired.

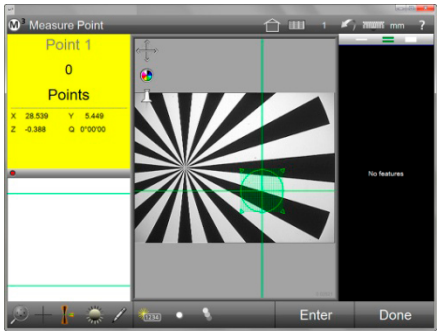
4 Press the desired feature measurement button and then press in the Measure Prompt or press the Enter button to initiate the Auto Focus measurement. The Auto Focus grid will be shown. In this example, the Measure Point button was pressed.

Measure toolbar > Measure Point button

Measure toolbar > Enter button

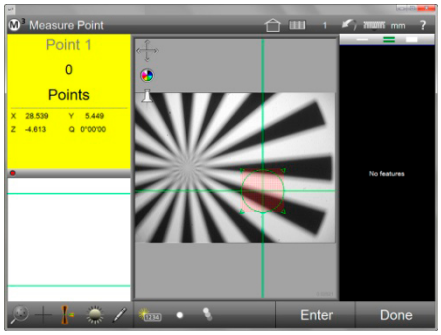


Crosshair positioned and sized

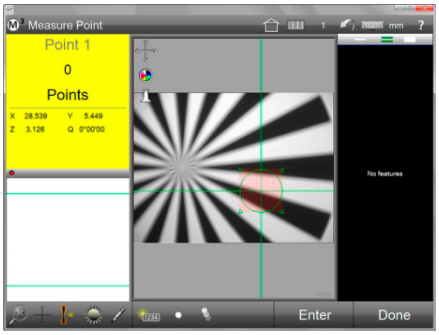


Auto Focus measurement initiated

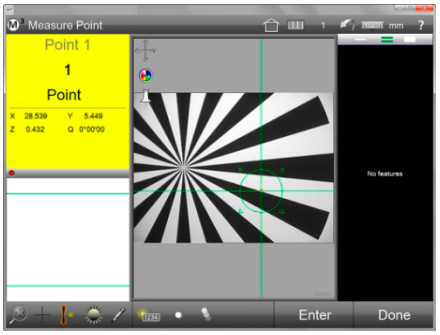
5 Raise and lower the z-axis through the range of focus until a point is probed and shown in the measure prompt viewport.



Out of focus z-axis low



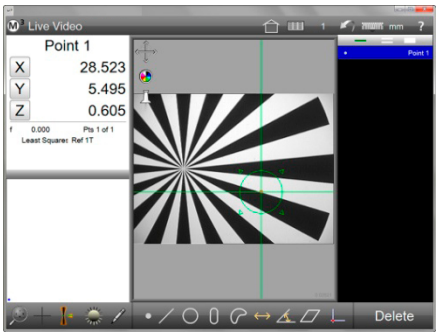
Out of focus z-axis high



In focus, point probed

6 Press the Done button to complete the measurement and enter the point into the Feature List.

Measure toolbar > Done button



Measurement completed

Auto Focus Measurements in CNC Systems Using Standard and Active Crosshairs

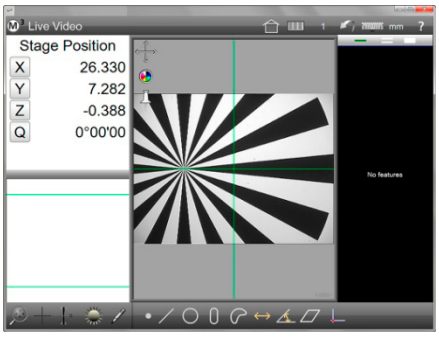
To perform Auto Focus measurements in CNC Systems using Standard and Active Crosshairs:

1 Select the standard or active crosshair probe from the probe menu.

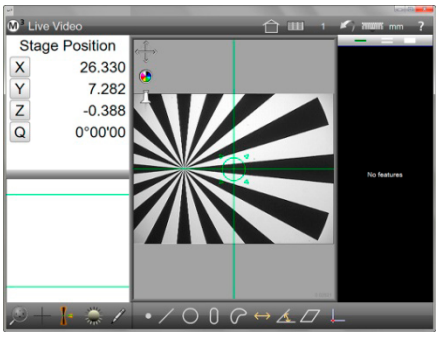
Measure toolbar > Probe menu > Standard or active crosshair probe

2 Enable Auto Focus (if it is not already enabled).

Measure toolbar > Auto Focus button



Simple crosshair selected



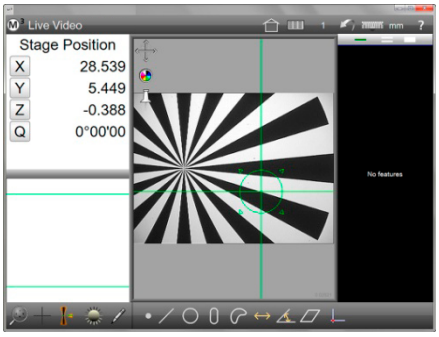
Auto focus enabled

3 Position the crosshair probe over the desired feature edge and then press and drag the triangle handles of the area of interest to change its size if desired.

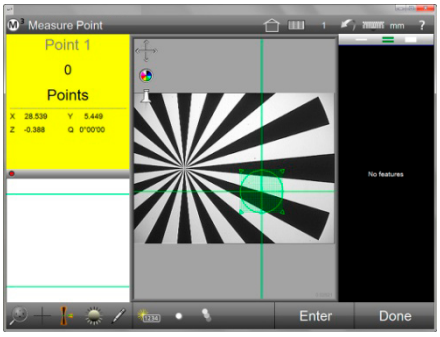
4 Press the desired feature measurement button and then press in the Measure Prompt or press the Enter button to initiate the Auto Focus measurement. The Auto Focus grid will be shown and the CNC system will raise and lower the z-axis through the range of focus to acquire a point in the optimum focal plane. In this example, the Measure Point button was pressed.

Measure toolbar > Measure Point button

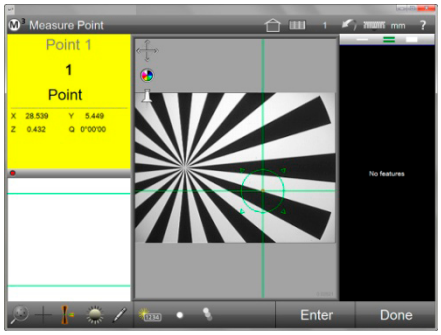
Measure toolbar > Enter button



Crosshair positioned and sized



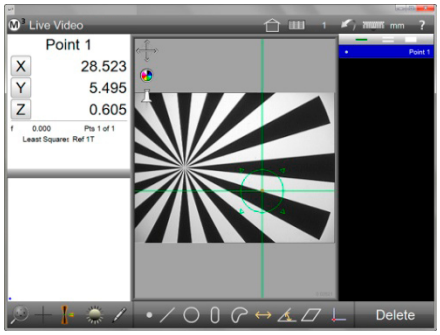
Auto Focus measurement initiated



In focus, point probed

5 Press the Done button to complete the measurement and enter the point into the Feature List.

Measure toolbar > Done button



Measurement completed

Auto Focus Measurements in CNC Systems using the Vtouch Probe

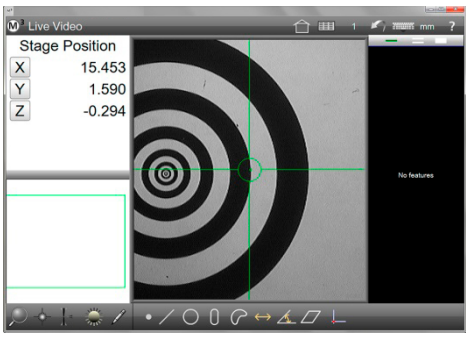
To perform Auto Focus measurements in CNC Systems using the Vtouch probe:

1 Select the Active crosshair probe from the probe menu.

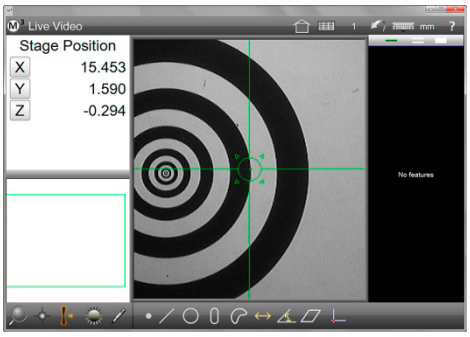
Measure toolbar > Probe menu > Active crosshair

2 Enable Auto Focus (if it is not already enabled).

Measure toolbar > Auto Focus button

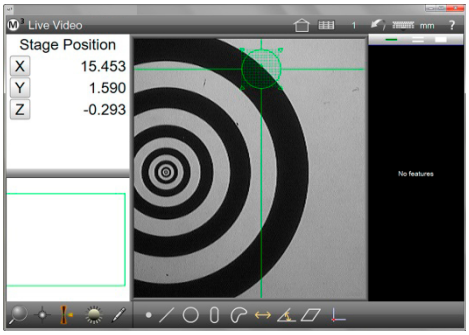


Vtouch probe selected

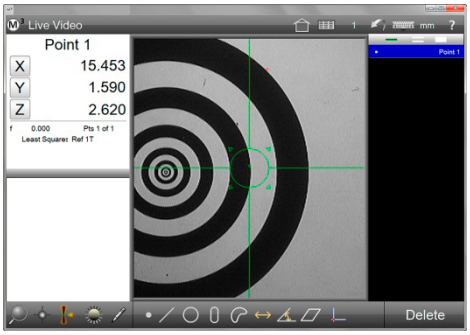


Auto focus enabled

3 Press the desired feature edge. The probe will move to the edge and the auto focus grid will appear. Then the CNC system will raise and lower the z-axis through the range of focus to acquire a point in the optimum focus plane. The point will be entered into the Feature List and the probe will return to the center of the screen.



Touch the desired edge



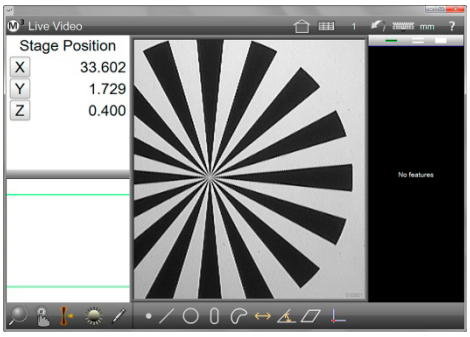
A point is automatically entered into the Feature List

Auto Focus Measurements in CNC Systems using the MeasureLogic Probe

To perform Auto Focus measurements in CNC Systems using the MeasureLogic probe:

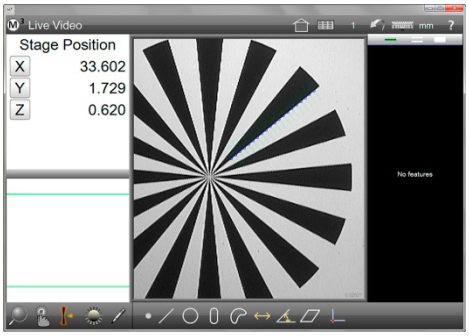
1 Select the MeasureLogic probe from the probe menu.
Measure toolbar > Probe menu > MeasureLogic probe

2 Enable Auto Focus (if it is not already enabled).
Measure toolbar > Auto Focus button

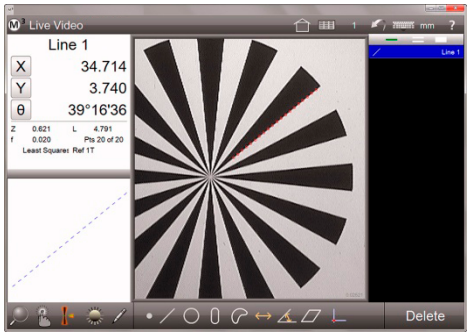


MeasureLogic probe selected and Auto focus enabled

3 Press the desired feature edge. The CNC system will raise and lower the z-axis through the range of focus to acquire points along the edge in the optimum focal plane. The points will be evaluated by the system and the appropriate feature will be entered into the Feature List.



Touch the desired edge



A feature is automatically entered into the Feature List

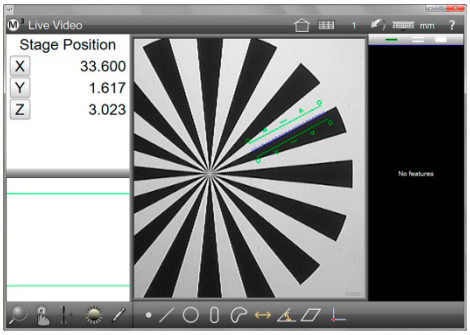
Auto Focus Measurements in CNC Systems using the EyeMeasure Probe

To perform Auto Focus measurements in CNC Systems using the EyeMeasure probe:

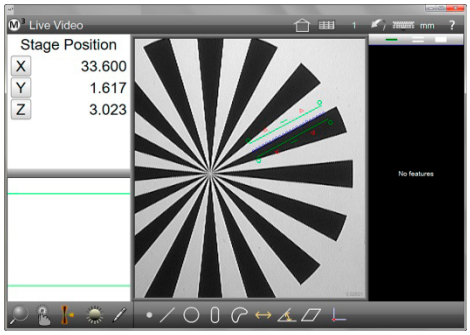
1 Select the MeasureLogic probe from the probe menu.
Measure toolbar > Probe menu > MeasureLogic probe

2 Press and drag along the desired feature edge to position the EyeMeasure probe over the edge.

3 Press and drag the probe handles to change the probe size if desired and then enable Auto Focus (if it is not already enabled).
Measure toolbar > Auto Focus button
The probe sizing handles will turn red to indicate that the Auto Focus mode is enabled.

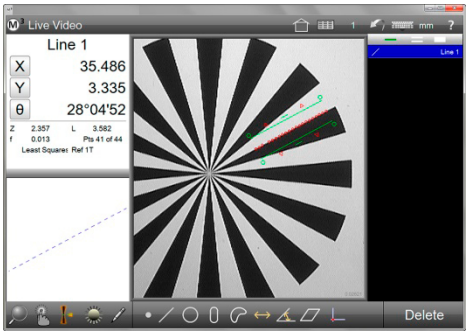


Press and drag the EyeMeasure probe along an edge



Enable Auto Focus

4 Press inside the EyeMeasure probe to initiate a measurement. The CNC system will raise and lower the z-axis through the range of focus to acquire points along the edge in the optimum focal plane. The points will be evaluated by the system and the appropriate feature will be entered into the Feature List.



A feature is automatically entered into the Feature List

Setting the Number of Points Captured by Multipoint Probes

The M3 software provides control of video edge detection point density for multipoint probes through the Stitch Spacing setting found in the Video Settings screen. Stitch spacing is defined as the number of pixels between video edge detection points collected and has a default setting of 10. Fewer pixels between points increases the number of points captured.

Adjust stitch spacing to control the number of points collected for a feature of a given size. When more points are desired for a given feature, decrease the stitch spacing. When fewer points are desired, increase the stitch spacing value. To change the value of stitch spacing, navigate to the Stitch Spacing data field in the Video Settings screen and enter the desired value.

M3 System menu > Settings menu > Video button > Stitch spacing data field

CAUTION

The default value of 10 pixels between data points is adequate for most applications. Significantly increasing the pixel space between data points can reduce the accuracy of feature measurements. Increasing or decreasing the point spacing can also affect the reliability of the measure logic probe. It is more difficult for the measure logic probe to make feature determinations when the spacing is too large, and the measure logic tool will follow an edge less effectively if the spacing is too small.

Using Video Probes Effectively

The section describes the effective use of the M3 video probes that make use of automatic edge detection. These probes include:

- Active crosshair
- Vtouch
- MeasureLogic
- EyeMeasure

Regardless of edge contrast or form characteristic, the standard crosshair can always be used to enter points by lining the crosshair intersection up with the desired edge and entering points manually.

The probe concepts discussed here apply to the use and behavior of probes in either the non-programmatic (interactive) mode, or in the program edit mode. Probe behavior in program playback will be discussed later in the section.

This probe discussion includes:

- Terms and functions that govern the performance and behavior of the M3 video probes
- Six inspection examples from typical vision inspection applications that will demonstrate the use and applicability of probe terms and functions

Video Probe Terms and Functions

The following terms define many of the functions that govern the performance and behavior of video edge detection probes:

- Vtouch active crosshair
- Auto teach MeasureLogic
- Manual teach MeasureLogic
- Buffer zone EyeMeasure

Vtouch Active Crosshair

The Vtouch probe is the active crosshair used to acquire a single point by pressing the touch screen or clicking the mouse cursor on an edge. The Vtouch probe:

- Can be used with either auto-taught or manually taught edge parameters
- Provides the ability to choose the best possible point location on an edge thereby improving accuracy and repeatability
- Requires an explicit feature measurement request, except for point features
- Is particularly useful when single point acquisition and flexibility in point placement are required

Auto Teach MeasureLogic Probe

The auto teach MeasureLogic probe is extremely robust, and is suitable for both soft and hard edges. The Auto Teach MeasureLogic probe:

- Operates once for each tool firing, capturing many data points
- Performs a statistical analysis of nearby edges and characterizes them
- Edge parameters are then used, such as threshold (edge strength), and edge width, to follow and acquire points on edges that are parametrically similar
- Is ideal for edges of good form characterization, and weak or strong edge contrast

NOTE

To improve low edge contrast point acquisition, the manual edge teach routine described below can be used in conjunction with MeasureLogic probe measurements.

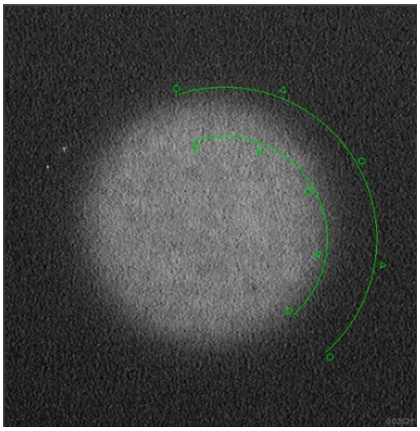
Manual Teach Procedure for the MeasureLogic Probe

The default edge contrast recognition parameters are adequate for probing under most conditions using the MeasureLogic (or EyeMeasure) probe. However, when probing in conditions that make good image contrast impossible, edge contrast recognition parameters can be changed by performing the manual teach procedure. The Manual Edge Teach procedure is used to teach the Measure Logic Probe light to dark contrast thresholds when it becomes necessary to measure in the presence of weak edge contrast, thin edges or both.

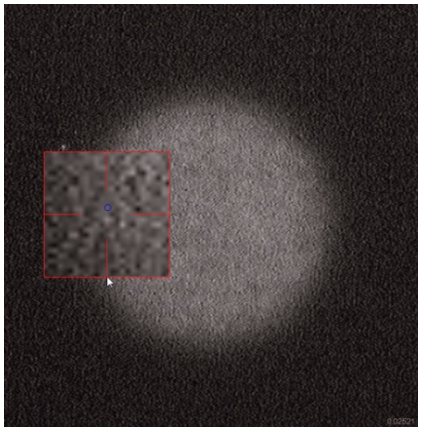
- Manual Teach function performs an extensive analysis of edge contrast and calculates several parameters used for subsequent measurements using any of the video edge detection probes
- A color change, to stake and candidate points, indicates points acquired under manually taught conditions
- The manually taught probe can be returned to the default auto teach mode by pressing the Unteach Button

To perform the manual teach procedure for the MeasureLogic probe:

1 When an edge is not recognized by the MeasureLogic probe, press and hold on the edge in the video window. A magnified image of the edge will be shown and the system will change edge contrast recognition parameters to accommodate the weakly defined edge.



Edge is not recognized by the MeasureLogic probe

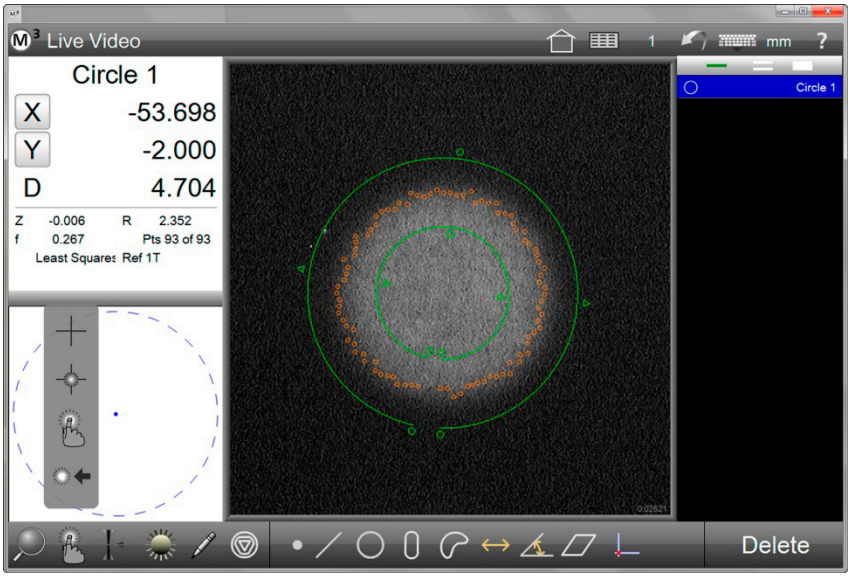


Press and hold on the edge

2 Release the edge. The MeasureLogic probe can now be used to acquire data points on the low contrast edge. In this example, a circle is probed on an extremely low contrast edge using the EyeMeasure application of the MeasureLogic probe.

3 The MeasureLogic probe edge contrast parameters can be reset to the default auto teach parameters by pressing the Unteach button.

Menu toolbar > Probe menu > Unteach



Advanced Edge Teach

-The Advanced Edge Teach(AET) mode introduces new fine tune controls to the video measurement process, improving edge collection results in complex imaging scenarios. The new Advanced Edge Teach dialog provides a more detailed look at candidate edges and provides "live" feedback of the candidate edges currently detected.

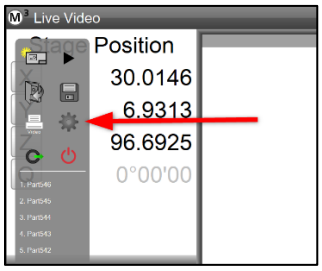
-Detection of low contrast, narrow, and difficult to image edges is improved with graph-based control of contrast, material direction, and edge width.

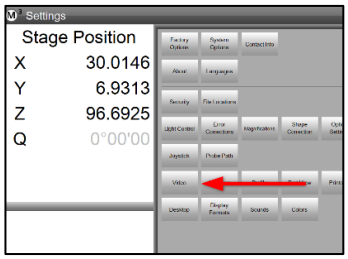
Advanced Edge Teach mode can be used in conjunction with any of the existing M3 VED probes.

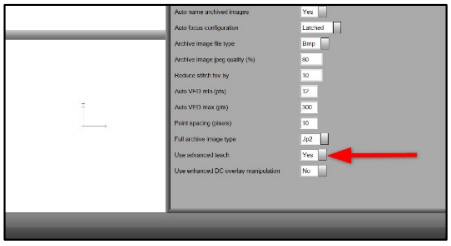
Setup:

IMPORTANT: The new Advanced Edge Teach functionality is OFF by default.

To enable the advanced edge teach functionality navigate to the Video Setup Screen and set the Advanced Edge Teach flag to "Yes".





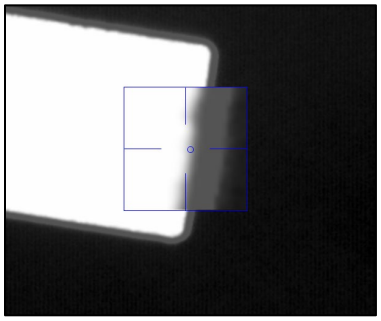


Operation:

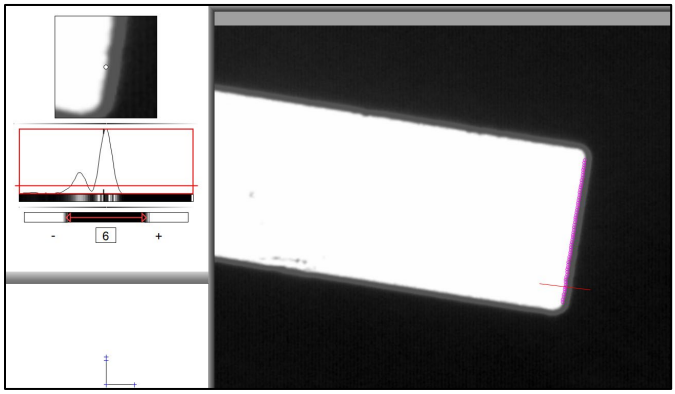
IMPORTANT: For brevity, the Advanced Edge Teach functionality will be referred to as "AET".

The AET function is initiated by long pressing on an edge in the Live Video window with any of the Video probes selected(Active Crosshair, MeasureLogic, or EyeMeasure).

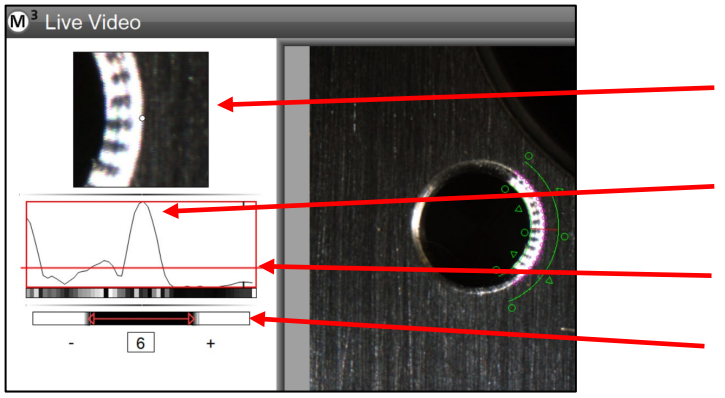
While continuing to hold down the mouse button or your finger, drag the small magnification popup window until the edge of interest is targeted by the small blue ringlet.



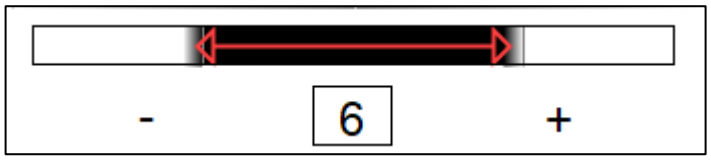
Release the mouse button or your finger to set the edge sample position, identified by a red stitch line drawn across the edge of interest. The AET dialog will also be displayed on the left side of the M3 desktop.



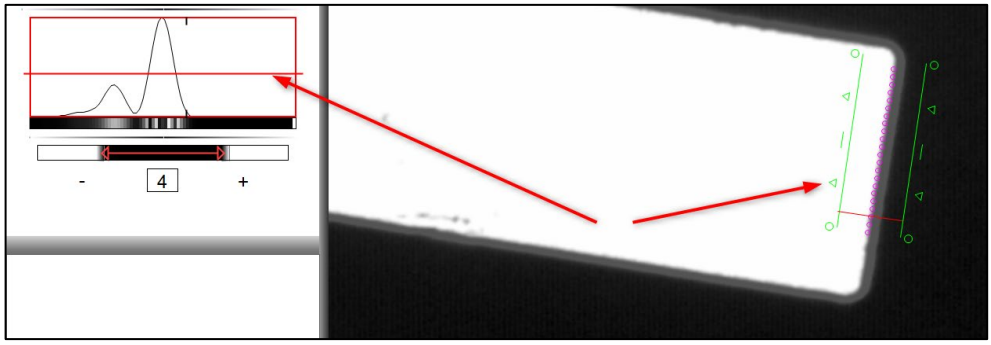
Adjust the red contrast threshold bar in the contrast graph to set the current threshold for edge detection. The points in the live video window will update in "real-time" to indicate current state of edge detection, based on the AET settings.



Adjust the Edge Thickness Filter value by pressing the (+) or (-) buttons. This filter sets the minimum number of continuous pixels of sufficient contrast required to be considered a valid edge. The unit is in pixels.

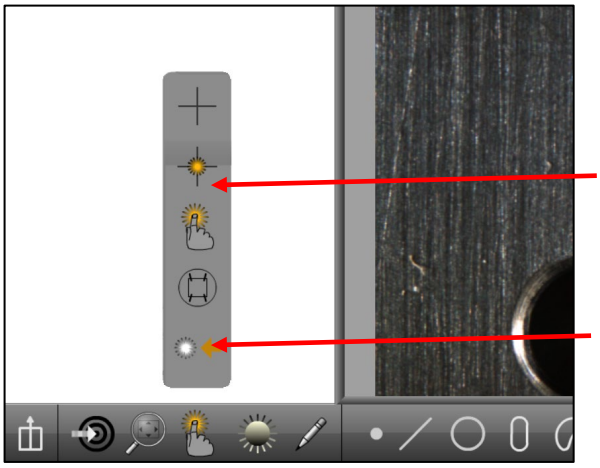


Using AET with the EyeMeasure Probe enables color sense and direction to be used along with the contrast threshold control to target specific transitions by contrast level.



Sample Procedure Steps for using Advanced Edge Teach:

1. Select, or generate the desired Video Probe.
2. Long Press on the live video window with the left mouse button, targeting the edge transition that you would like to modify edge parameters for. You will see the magnification popup window appear proving a zoom region to assist with targeting.
3. Release the left mouse button once the target is achieved.
4. Make adjustments to the AET dialog. The candidate edge points will be updated "real-time" in the live video window as adjustments are made.
5. Adjust the red Contrast Threshold Bar to set the min contrast detection level for the current edge of interest. Adjust the Minimum Edge Thickness filter(in pixels) by incrementing the value with the "+" or "-" buttons.
6. Press the "Done" button to save the current AET settings.
7. Once modified edge parameters have been saved the video probes will be displayed in the toolbar with an orange colored tint, indicating that the system is currently using modified edge parameter.



-Probes displayed with an "orange tint" indicate that custom edge parameters are currently in use.

-Press the orange "left arrow" button to discard the current AET and return to default "Auto" edge parameters.

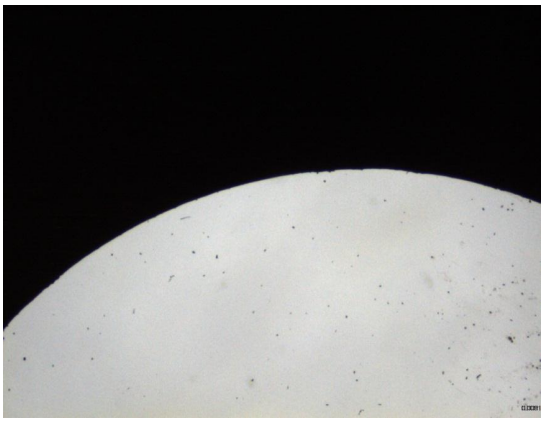
Buffer Zone EyeMeasure Probe

The EyeMeasure probe is particularly useful and robust in situations of poor edge form, or when it is required or helpful to control the exact length and relative orientation of the point acquisition path. The EyeMeasure video probe:

- Can be used with either the default auto teach or the manual teach edge parameters
- Attempts to find a continuous edge along a traced path
- Can have its shape, length and rotational orientation modified using one of the buffer handles
- Provides feature type intelligence for Lines, Circles, and Arcs, when features are not explicitly requested
- Is particularly useful for features with poor form such as having noisy edges, excess material or contamination, or interrupt features, e.g. a keyway.

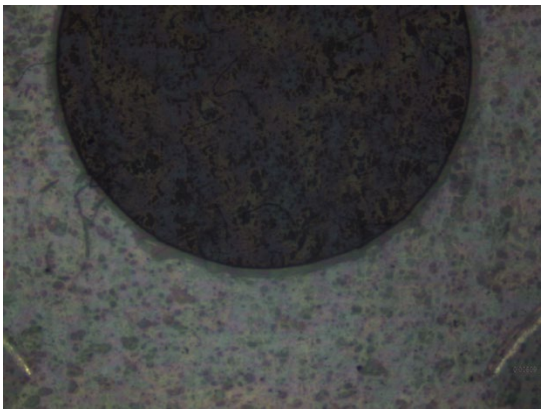
Inspection Example Images

The following six video inspection images demonstrate a variety of edge conditions that may be encountered in typical vision inspection applications. These images will demonstrate the use and applicability of the terms and functions discussed above. Each image is followed by a description of the M3 Video Probe capability or optimization that was utilized to achieve successful acquisition of points on the edge in question.



High Edge Contrast and Uniform Edge Form

The MeasureLogic probe is typically the ideal tool for high edge contrast applications. Strong edge contrast and consistent feature form are ideal for the automatic edge detection capability of this probe. The MeasureLogic probe can also be used in the alternative EyeMeasure mode. The default auto teach behavior of this probe is generally adequate for edges like the one pictured here.



Low Edge Contrast and Uniform Edge Form

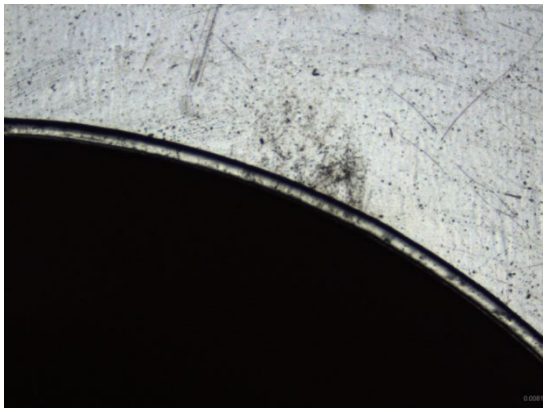
For edges with poor edge contrast, but with good edge form, the manual teach routine can be used to optimize the edge detection capability of the MeasureLogic probe. The consistent edge form

makes the MeasureLogic probe a suitable probe choice for successful point acquisition in the sample shown here. The MeasureLogic probe can also be used in the EyeMeasure mode.

NOTE
The manual teach procedure was described in detail earlier in this section.

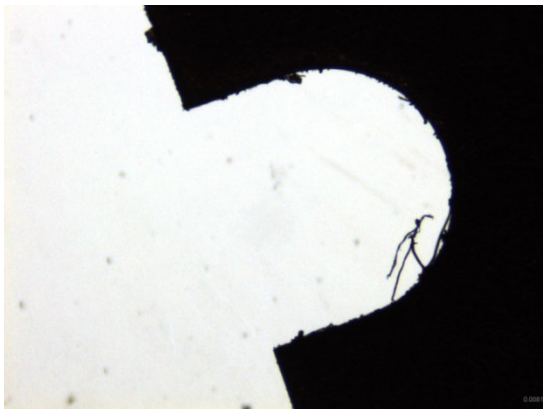
High Edge Contrast and Low Edge Thickness

In some cases thin edges will require the manual teach routine to be performed. This is especially true when the edge is located close to other edges. In these cases, the manual teach routine optimizes the MeasureLogic probe for a specific edge thickness profile. Following the manual teach routine, the Measure Logic probe will be optimized for point acquisition on thin edges.



High Edge Contrast and Irregular Edge Form

For edges that present significant edge contamination, using the EyeMeasure probe can provide enhanced point acquisition within the defined buffer zone of this probe. The EyeMeasure probe is generated by selecting the MeasureLogic tool button from probe menu, and tracing along the desired edge, the buffer zone will be drawn along the traced edge. Features can be measured explicitly, or in the case of lines and circles, automatically, using this probe. Press or click within the buffer zone to fire the probe and acquire points.

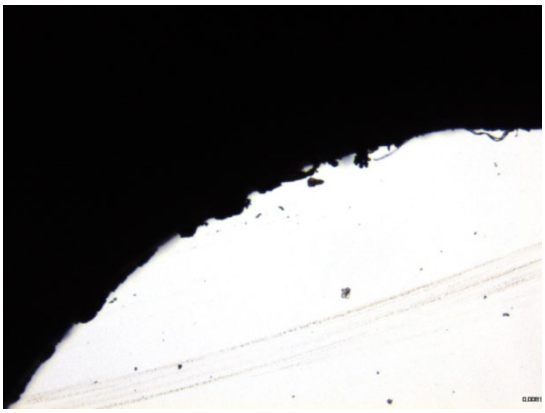


Low Edge Contrast and Irregular Edge Form

The manual teach routine can also be used in conjunction with the EyeMeasure probe. Due to the poor edge contrast, the image shown here likely requires the edge parameters to be taught prior to adequate point acquisition. The minor to moderate edge contamination exhibited here makes the EyeMeasure probe ideal for probing.

NOTE
The manual teach procedure was described in detail earlier in this section.

Edge Characteristics Requiring Single Point Acquisition



When a single point is desired at a specific location along an edge, the Vtouch probe should be used. The Vtouch probe acquires a single point at an edge location specified by the operator with a single press or click on the video window image. Typically explicit feature measurements are performed and the individual points are probed at the desired locations along the edge of a given feature.

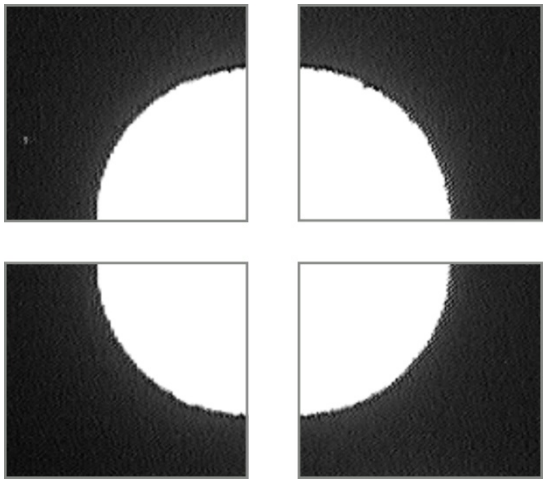
For edges that have poor edge contrast, the manual teach procedure can be performed prior to utilizing the Vtouch probe to improve the edge detection performance.

To use the Vtouch probe, select the Active crosshair in the probe menu. Probe the desired points by pressing or clicking on the edge where you would like to acquire a point.

Features that Extend Beyond the Field of View

In the M3 software, groups of acquired points are assigned to a particular feature measurement based on the probe that was used to capture them and whether or not the feature measurement was initiated explicitly by selecting a feature type prior to probing points.

The example shown here represents one of the most compelling reasons for ***explicit feature measurement***: features that do not fit within the field of view (a single video window). Because manufactured parts are never truly perfect and therefore deviate from perfect form, the M3 software might require the specification of a feature type prior to collecting groups of points. Explicit measurements will ensure that the desired feature is always returned upon completion of a measurement.



Feature distributed over four video windows

NOTE

A detailed example of measuring beyond the field of view is provided in Section 4: Measuring, Constructing and Creating Features.

Probe behavior in program playback

Part programs are recorded as measurements are performed. During program playback, the part program remembers the approximate probe position and gradient of each edge that was recorded as part of a feature measurement. As program playback proceeds, the system simply moves to the next recorded position to find an edge. When an edge is found it must be similar to the recorded edge at that position to be considered a valid point for acquisition. Thus, program playback can be viewed as a collection of stage positions and their associated candidate points based on the recorded edge parameters of measured features. The video probe used for the recorded measurement is inconsequential to the manner in which points are collected during playback.

Once part registration has been established, the system advances to the next programmed stage position with a screen press or Enter button press in manual systems, and occurs automatically in CNC systems. All valid candidate points that fall within a given field of view will be collected automatically until the feature measurement is complete.

Pattern Recognition

The pattern recognition function teaches specific image patterns to the system for automatic detection during program playback. Program playback can be improved using a soft-fixture based part alignment routine, utilizing this pattern detection mechanism.

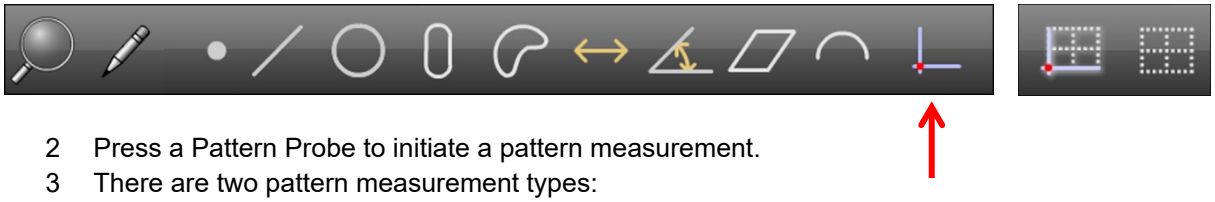
Patterns can be measured, or taught, as datum patterns where the skew and zero positions are set automatically or as standard pattern features where the part or machine coordinate pattern position and orientation are reported for the pattern feature.

Teaching a pattern

Select the desired pattern to be used for detection in playback. Ideal candidate patterns should contain image characteristics that are unique relative to the rest of the field of view in which they are taught. In addition, target pattern regions should contain reasonable light to dark pixel contrast and be relatively free of image noise and distortion.

To teach a pattern:

1. With the desired pattern in the field of view, press the Datum button in the Measure Toolbar to display the Datum Pattern Menu.

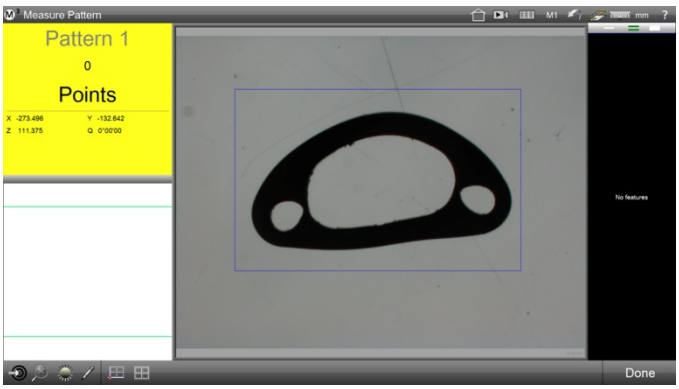


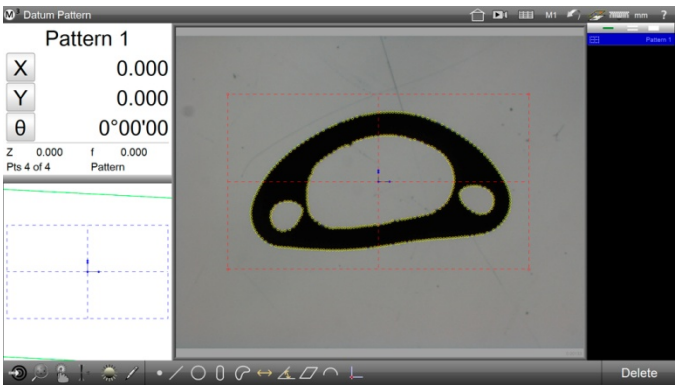
2. Press a Pattern Probe to initiate a pattern measurement.
3. There are two pattern measurement types:
   - Datum pattern measurement (default type)
   - Standard pattern measurement

Datum Pattern Measurement

This is the default pattern measurement and sets the measured pattern feature to XY=0 and angle=0. To measure (or teach) a datum pattern:

1. Select the Measure Datum Pattern tool (default), press and drag to completely enclose the desired pattern. The pattern feature will be shown in the Feature List, and the pattern position and angle will be shown in the detail view, with position and angle set to zero. The pattern will be registered for Program playback as shown in the Part View, and other features can be probed.

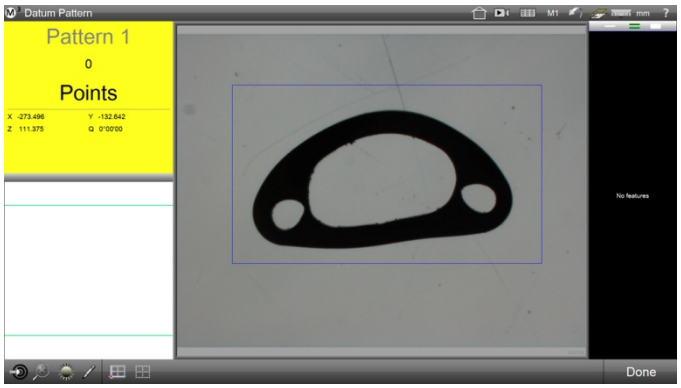


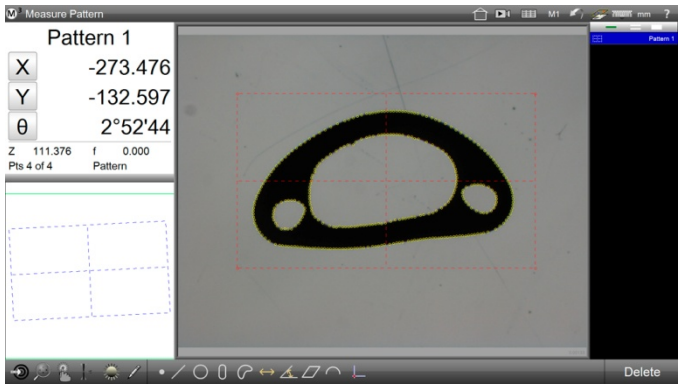


Standard Pattern Measurement

To perform a standard pattern measurement (teach):

1 Select the Standard Pattern Tool and then press and drag to completely enclose the desired pattern.





The pattern will be shown in the feature list, and the current stage position and angle will be shown in the detailed view window.

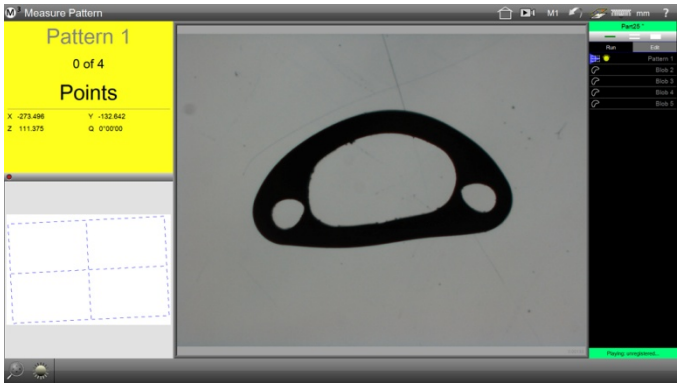
Note

Standard pattern features are typically used when a datum alignment will be constructed from two or more pattern features.

Pattern Recognition in Program Playback

Once a fully registered datum pattern has been measured (taught) and other features have been probed, the measuring program can be played back. To play the program:

1 Position the datum pattern anywhere in the field of view and then press anywhere in the Live Video window to execute the program. The program will recognize the datum pattern, register the coordinate system and measure any remaining features.



Notes

In CNC-enabled systems, a message will be displayed asking for playback confirmation prior to any potential stage movement.

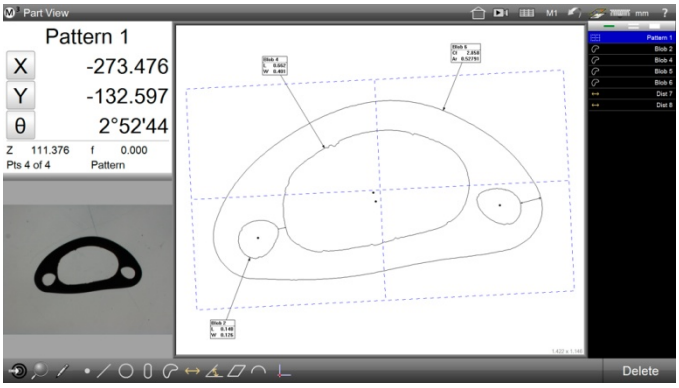
Important note for CNC systems

The angle of a given pattern is used as a temporary skew in program playback. Because the repeatability of the pattern angle is good only to 1°, subsequent features, particularly when located far from the initial pattern, may not be found automatically. To use the pattern finder most effectively in this case, choose an initial pattern that is located as close to the intended datum features as possible.

Viewing the Image Source for Pattern Features

For any measured (taught) pattern feature, the large feature detail screen can be used to view the originally sampled image of a given pattern. To view a pattern:

1 Select the pattern feature from the Feature List that you would like to view.
2 Press in the small feature detail window at the top left of your screen to access the large feature detail window. The original image sample will be displayed for the selected pattern.



Using Touch Probes

In supporting system touch probes can be used to measure reference skews and datums and 2D and 3D geometric features including points, lines, circles, arcs, slots, rectangles, distances, angles, planes spheres, cylinders and cones. Only the video Measure Blob tool is used to measure blobs.

Straight, Disc, Cylinder and Star Probes

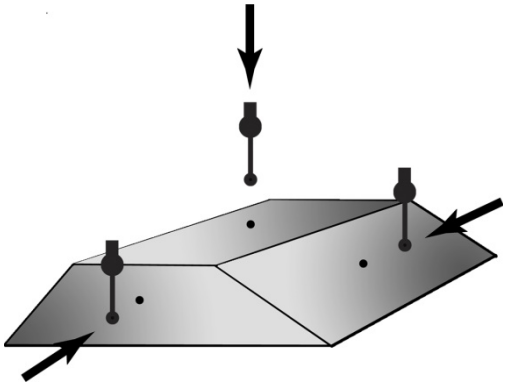
All touch probes are effectively used in the same way though proper probe selection will be important for certain application where probe tip type and probing direction is citical. To use a touch probe:

1 Select the desired touch probe from the Probe menu.
Measure toolbar > Probe menu > Touch probe
2 Move the stage to make probe contact at the desired feature location. A point will be entered upon contact.
3 Probe the required number of points and then press the Done button to complete a feature measurement.
Measure toolbar > Done button

Making Probe Contact

Probing is optimized by following some simple guidelines:

- Approach the target surface without direction changes in the last 5 to 10 mm
- Try to make the approach direction orthogonal to the target surface
- Do not drag the probe across the surface
- Do not probe sharp edges or allow the tip to slip off an edge



Touch Probe Setup Operations

Most systems that support the touch probe option are already prepared to probe feature points with a touch probe. However, in some cases it will be necessary to perform some basic probe operations before measuring features:

- Calibrate a probe tip
- Add a probe tip
- Remove a probe tip
- Edit a probe tip
- Change the current probe tip
- Configure probe path data

Calibrating a Probe Tip

Probe tip diameter and position are calibrated by measuring a certified qualification sphere artifact. Probe tips are calibrated using the:

- Contact Probe Settings screen
- Probe Teach function

NOTE

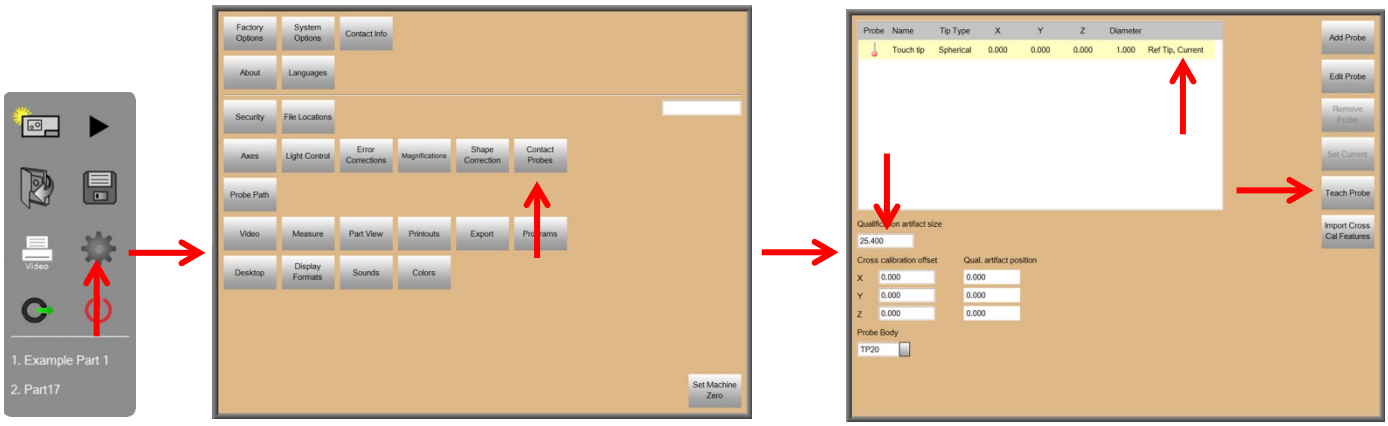
The system defaults to a sphere artifact, but can be configured by the system installer to use a certified ring gauge.

Calibrating using the Contact Probe Settings Screen

To access the Contact Probe Settings screen:

1 Press the Contact Probe Settings screen button.

M3 system menu > Settings > Contact Probes

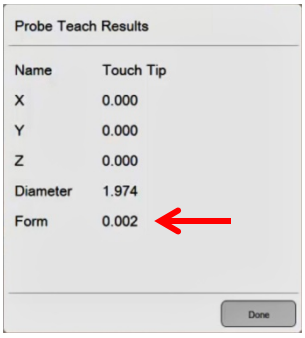


2 Confirm that the displayed qualification sphere artifact size matches your qualification sphere diameter in the current unit of measure.

3 Verify that the probe tip you wish to calibrate is set as the current probe. If it is not, select it in the probe group and press the Set Current button

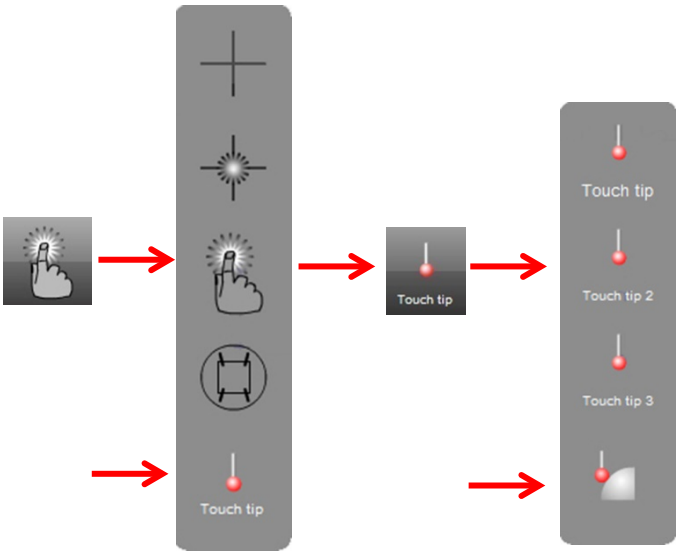
4 Press the Teach Probe button to display the Teach Probe measure screen.

5 Probe a minimum of 4 points well distributed around the qualification sphere and then press Done. The Probe Teach Results dialog will be displayed. The measurement form error should be very small. If the form error is outside acceptable limits, recalibrate the tip. If an unacceptable form error persists, seek technical assistance.

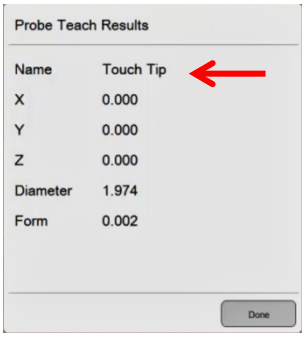


Calibrating using the Probe Teach Function

When you are certain that the qualification sphere used in your system is the correct diameter, the Probe Teach function accessed from the Probe Menu may be used to calibrate the probe tip. To initiate the Probe Teach function:



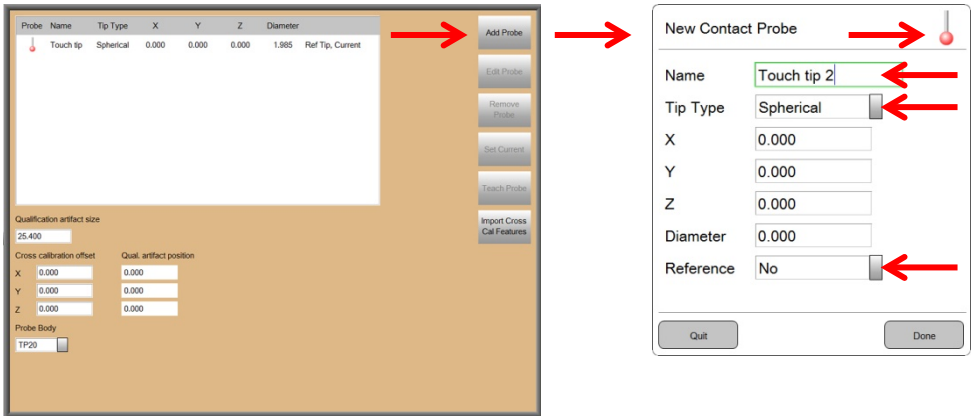
1. Press the Probe Menu button and then press the Touch Probe button to select touch probes (if necessary).
2. Long-press the Touch Probe button to display the Touch Probe menu, and then select the probe that you wish to calibrate (if necessary).
3. Long-press the Touch Probe button to display the Touch Probe menu, and then press the Probe Teach button. The Teach Probe measure screen will be displayed.
4. Probe a minimum of 4 points well distributed around the qualification sphere and then press Done. The Probe Teach Results dialog will be displayed. The measurement form error should be very small. If the form error is outside acceptable limits, recalibrate the tip. If an unacceptable form error persists, seek technical assistance.



Adding a Probe Tip

To add a probe tip:

1. Press the Contact Probes Settings screen button to access the Contact Probe Settings screen.
   M3 system menu > Settings > Contact Probes
2. Press the Add Probe button to display the New Contact Probe dialog.
3. Press the tip graphic in the upper-right corner of the dialogue to cycle through and select the desired tip orientation. Tip orientations include:
   - Down
   - Left
   - Right
   - Front
   - Back

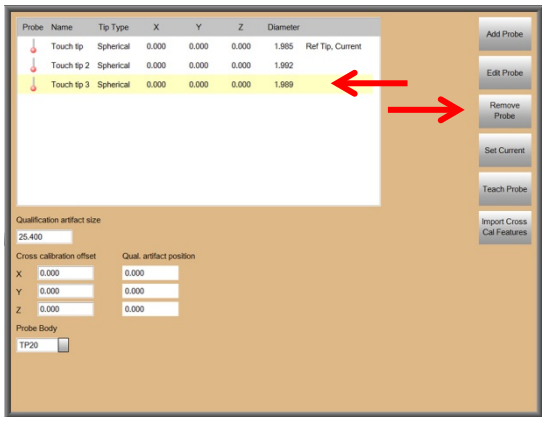


4. Enter a tip name into the Name field.
5. Press the Tip Type drop-down button to select a type. Types include:
   - Spherical
   - Disk
   - Cylindrical
6. Press the Reference drop-down button to designate the probe as the reference if desired.
7. Press Done. The new probe tip will be displayed in the probe group and must now be calibrated. Select the new tip, press the Set Current button and then press the Teach Probe button to calibrate the tip as explained earlier.

Removing a Probe Tip

To remove a probe tip:

1. Press the Contact Probes Settings screen button to access the Contact Probe Settings screen.
   M3 system menu > Settings > Contact Probes
2. Select the desired probe in the probe group and press the Remove Probe button.



CAUTION

When multiple probes are shown in the probe group, it is possible to remove the reference probe. It may be necessary to remove the reference probe if the tip becomes damaged, or there may be instances when the reference probe tip needs to be replaced to address the requirements of a specific application. When the reference probe tip is replaced, the current calibrations of all other tips in the group become invalidated and after the calibration of the new reference tip, new calibrations of all other probes will need to be performed.

NOTE

When the current probe is removed, but is not the reference, the system sets the reference probe as current.

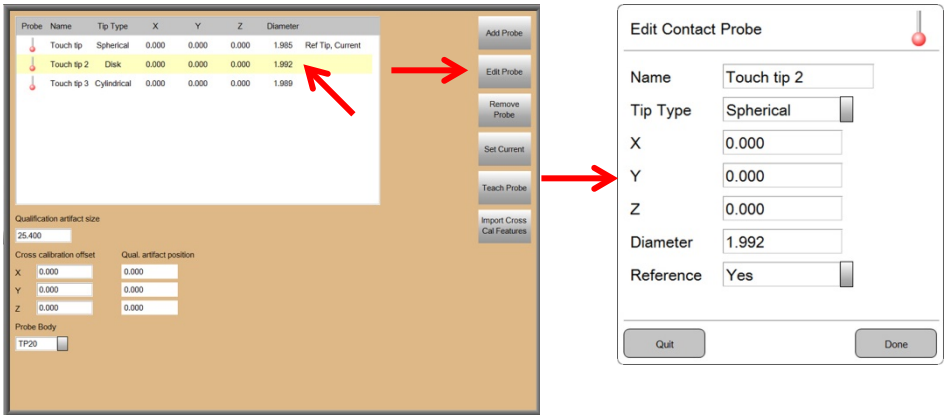
Editing a Probe Tip

Probe tips can be edited to:

- Change the name
- Assign a tip type
- Designate the probe as the reference

All probe editing is performed in the Edit Contact Probe dialog of the Contact Probe Settings screen. To display the Edit Contact Probe dialog for the probe to be edited:

1. Press the Contact Probes Settings screen button to access the Contact Probe Settings screen.
   M3 system menu > Settings > Contact Probes
2. Select the probe tip to be edited from the probe group.
3. Press the Edit button to display the Edit Contact Probe dialog.



Changing the probe Tip Name

To change the name:

1 Enter the new name into the Name field and press Done.

Assigning a Tip Type

To assign a tip type:

1 Press the Tip Type drop-down box button, select the desired type and press Done. Available tip type assignments include:

- Spherical
- Disk
- Cylindrical

Designating the Probe as a Reference

Any probe in the probe group can be designated as the reference.

CAUTION

When the reference probe tip is changed, the current calibrations of all other tips in the group become invalidated and after the calibration of the new reference tip, new calibrations of all other probes will need to be performed.

To designate a probe as the reference:

1 Press the Reference drop-down box, select Yes and press Done.

Changing the Current Probe Tip

To change the current probe tip:

1 Press the Contact Probes Settings screen button to access the Contact Probe Settings screen.
M3 system menu > Settings > Contact Probes

2 Select the desired probe in the probe group and press the Set Current button. The selected probe will now be shown as current in the probe group.

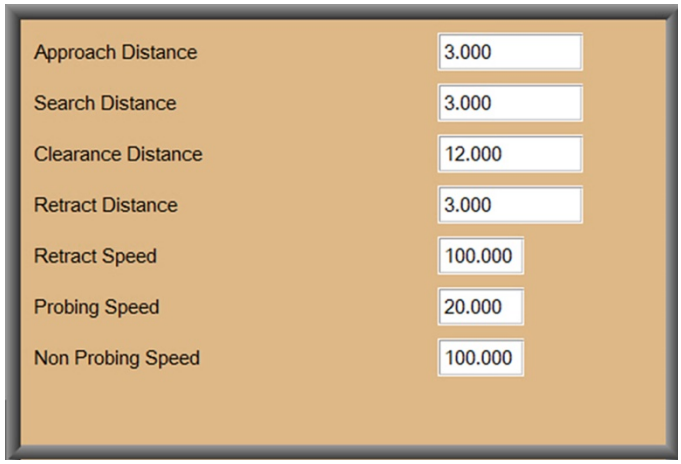
NOTE
When a probe is selected from the Probe menu of the Measure toolbar it is set as the current probe.

Configuring Probe Path Data

The probe path data settings define the default probe touch probe behavior for CNC programs. All probes used in programs initially use the default probe path data settings. These settings can be edited in program steps when the default probe behavior needs to be modified to meet the needs of a specific application. Editing probe path data steps is discussed in Section 11: Programming.

Probe path data are configured in the Probe Path Settings screen. To configure probe path data:

1 Press the Probe Path Settings screen button. The Probe Path default configuration will be displayed.
M3 System menu > Settings > Probe path



Probe path data configuration fields include:

- Approach distance: Distance from the anticipated target surface where probing speed is initiated in the current unit of measure. The approach path is calculated by the system to be approximately orthogonal to the target surface.
- Search distance: Distance added to the approach distance when surface contact is not made at the anticipated location. The measurement fails if contact is not made within the search distance.
- Clearance distance: There are two clearance distances of the same value. One clearance is toward the target and included after the non-probing speed to the target and before the approach distance. The second is away from the target and included after the retract distance and before the next non-probing speed begins to the next target. The clearance moves are intended to locate the probe so that the to-target and from-target moves will be unobstructed. The clearance directions are determined by the system based on the target feature surface being probed and generally moves along the probe tip axis
- Retract distance: Distance that the probe moves immediately after contact is made with the target surface in the current unit of measure. The retract path retraces the approach path for a small distance at higher speed.
- Retract speed: The speed in the retract region expressed in mm per second
- Probing speed: The speed in the approach and search regions expressed in mm per second
- Non-probing speed: The maximum probe speed between features

CAUTIONS
When preparing programs for parts that include complex surface variations in 3D space, care must be taken to avoid probe collisions. The clearance distance might not be adequate to eliminate collisions and might need to be replaced or supplemented with manually programmed Goto (safe) moves. Goto moves are discussed in Section 11: Programming.

The default Non-probing speed might be too high for first executions of complex part programs. Debugging complex programs at lower Non-probing speeds reduces the possibility of probe damage. The Non-probing speed can easily be changed in program steps during the program debugging process.

Section 4: Measuring, Constructing and Creating Features

Measurements can be performed on manual and CNC enabled systems using single point video, multipoint video or touch probes. Measurements can be performed explicitly using any probe or automatically using the MeasureLogic or EyeMeasure probes. Blobs must be measured explicitly using a special tool. The measurement of blobs is a special case discussed later in this section. Additional measurement tools are provided for FOV systems. FOV systems are discussed in detail in Section 12: Field of View Functions.

NOTES

Most measurement, construction and feature creation concepts discussed in this section apply equally well to encoder-based moveable stage systems and fixed stage FOV systems. However, some of the material in this section will not apply to FOV systems, such as material regarding CNC control or encoder-based positioning and location. Measurements in FOV systems occur only in a fixed position field of view and are based on video pixel resolution.

Touch probes are provided only as an option on CNC systems.

Measurement activities can include:

- Measuring features by probing
- Constructing features
- Creating new features based on data provided by the user

Users can perform quick checks of individual part features without establishing a reference frame or can establish a reference frame including a part level, part skew and zero datum in preparation for formal measurements that will be compared to part drawings and be recorded as a part program.

Explicit and Automatic Measurement

Explicit feature measurements are always performed by pressing the desired Measure Feature button and then probing at least a minimum number of required points on the feature's edge. Features measured with the MeasureLogic or EyeMeasure tool can be measured explicitly or can be measured automatically by allowing the M3 software to determine the feature type based on the data points collected. Touch probe measurements are always explicit.

Feature Constructions

Feature constructions are performed explicitly by pressing the desired Measure Feature button and then selecting the parent features from the Feature List or from the Part View that will be used for the construction. When the feature construction is complete the resulting construction will be added to the Feature List and Part View.

Feature Creation

Features are created by pressing the desired Measure Feature button, pressing the associated Create Feature button and entering the desired feature parameter data into the fields provided.

Preparing for Measurement Activities

Preparation for measurement can include the following:

- Lighting adjustment
- Teaching video edge recognition
- Calibrating a touch probe tip
- Configuring (touch) probe path data

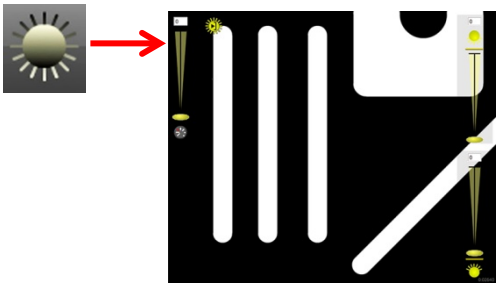
Lighting Adjustment

To make stage lighting adjustments:

1 Press the Light Control button to superimpose the light lighting controls over the Live Video part image.

2 Adjust the sliders or enter values into the data fields to obtain the desired part lighting.

Measure toolbar> Light Control button



Calibrating a Touch Probe Tip

Please refer to Touch Probe Setup Operations in Section 3: Probes.

Configuring (Touch) Probe Path Data

Please refer to Configuring Probe Path Data in Section 3: Probes.

Teaching Video Edge Recognition

Default edge detection parameters are adequate for most applications. However, poor contrast images can make multipoint probing with the edge detection probes difficult or impossible. The edge detection parameters can be changed to optimize probing. Please refer to the *Manual Teach Procedure for the MeasureLogic Probe* discussed earlier in Section 3: Probes for detailed instructions.

Establishing a Reference Frame

While a reference frame might be unnecessary for quick checks of part features in prototyping or similar environments, a reliable measurement reference must be established before part features can be measured as part of a formal manufacturing or inspection process. The M3 measurement reference frame can consist of:

- Part leveling (a primary datum plane)
- Skew alignment (secondary alignment)
- Part Datum (measurement coordinate system origin)

Part Leveling

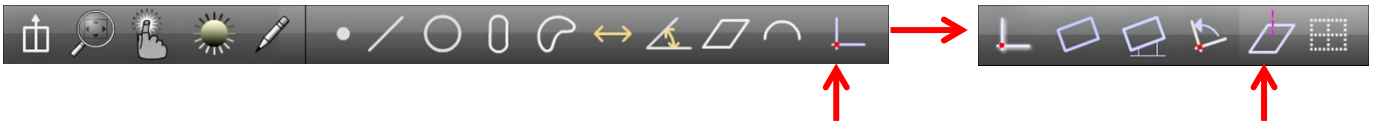
When the surface plane of a part is not orthogonal to the measuring axes, small cosine errors can be generated during the measurement process. These errors can be eliminated by leveling the part.

> ***NOTE***
> ***This might not be required in most well-maintained systems. Additionally, for VED measurements, if the depth of field of the camera system is not less than the level error, no benefit can be gained from leveling the part. Part leveling of 15° or more is not permitted.***

To measure a primary datum plane:

1 Select the desired probe from the probe menu.
Measure toolbar > Probe menu > Desired probe

2 Press the Datum Plane button of the Datum menu.
Measure toolbar > Datum menu > Datum plane button



3 Probe a minimum of three points evenly distributed across the surface of the part. Probing more than three points increases the accuracy of the measurement.

4 Press the Done button to complete the measurement and enter the datum plane into the Feature List.
Measure toolbar > Done button

> ***NOTE***
> ***Previously measured or constructed plane features can also be leveled by manually zeroing the theta angle coefficient and the Z position coefficient. Pressing the label buttons to the left of these coefficients in either the feature detail screen or the DRO viewports will set the corresponding values to zero.***

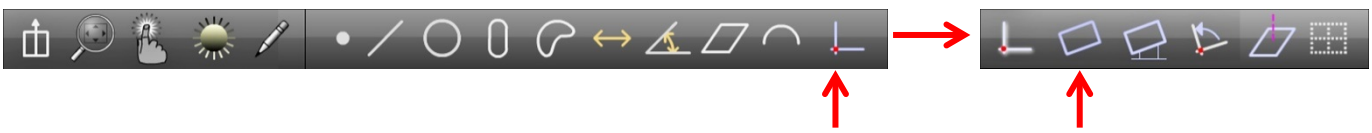
Skew Alignment

When a part is misaligned (twisted) in the XY plane, small cosine errors can be generated during the measurement process. These errors can be eliminated by creating a skew alignment for the part. The skew alignment performs a precise measurement of the part of misalignment in the XY plane. Once the misalignment is known to the system, subsequent feature measurements are automatically compensated to eliminate cosine errors. Measurement data in the feature detail screen and DRO viewports will reflect measurements of a perfectly aligned part.

To create a skew alignment:

1 Select the desired probe from the probe menu.
Measure toolbar > Probe menu > Desired probe

2 Press the Skew button of the Datum menu.
Measure toolbar > Datum menu > Skew button



3 Probe a minimum of two points well distributed along the entire length of the desired part reference edge.

4 Press the Done button to complete the measurement and enter the skew line into the Feature List.
Measure toolbar > Done button

Part Datum

Creating the part datum point is the final step in establishing a valid reference frame for subsequent measurements. The part datum can be created by:

- Probing a single point
- Constructing a point from one or more parent features
- Manually zeroing the X and Y coefficients of a feature

Probing a Part Datum Point

The part datum can be created directly by probing a point. To probe a part datum point:

1 Select the desired probe from the probe menu.
Measure toolbar > Probe menu > Desired probe
2 Press the Datum button.
Measure toolbar > Datum button
3 Probe the desired point and press the Enter button or in the Measure Prompt window. The point will be probed.
Measure toolbar > Enter button or Measure Prompt window
4 Press the Done button to complete the measurement and enter the part datum point into the Feature List.
Measure toolbar > Done button

Constructing a Skew and Part Datum Point from Parent Features

The skew alignment and part datum can be constructed from one or more parent features that are contained within the Feature List when simply probing a desired point or skew line is not convenient or possible. Supported point and line constructions are specified in *Constructing Features* later in this section.

Skew

To construct a skew line from a pair of positional features (two circles in this example):

1 Press the Skew button.
Measure toolbar > Datum button > Skew button
2 Select the desired parent features or features in the Feature List, such as two circles.
3 Press the Done button to complete the construction and enter the part skew alignment into the Feature List.
Measure toolbar > Done button

Datum Point

To construct a datum point from a single feature:

1 Press the Datum button.
Measure toolbar > Datum button
2 Select the desired parent feature or features in the Feature List.
3 Press the Done button to complete the construction and enter the part datum point into the Feature List.
Measure toolbar > Done button

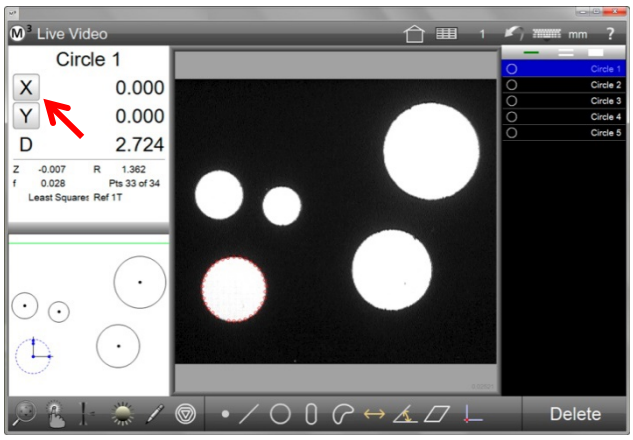
Creating a Datum by Manually Zeroing the X and Y Coefficients of a Feature

Any of the features contained in the Feature List can be used as a part datum by manually zeroing the X and Y feature coefficients. The X and Y coefficients can be zeroed in the DRO viewport or in the feature detail screen. To zero the X and Y coefficients of a feature:

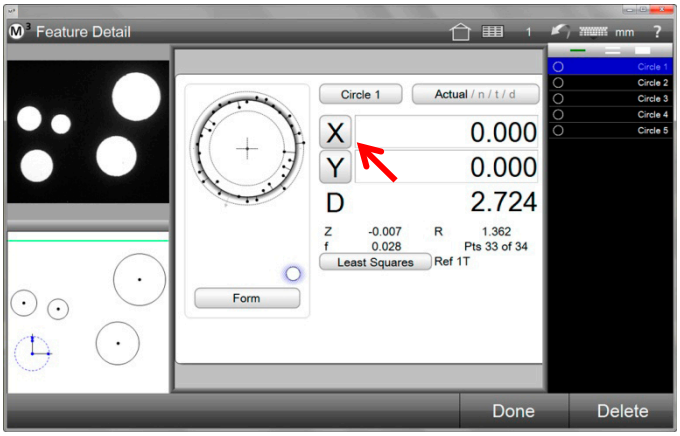
1 Select the desired feature in the Feature List.

2 Press the X coefficient button and then the Y coefficient button in the DRO viewport or in the feature detail screen. The X and Y values will become zero establishing the geometric center point of the feature as the part datum point.

In this example, the geometric center point of circle 1 was set to zero and will serve as the part datum point for subsequent measurements.



X and Y set to zero in the DRO



X and Y set to zero in the feature detail screen

Probing and Measuring Features

Part features are measured by probing and collecting points that define the feature shape and location. Features can be measured explicitly using any probe or automatically using the MeasureLogic or EyeMeasure probes. Explicit feature measurements are always performed by pressing the desired Feature Measure button and then probing at least the minimum required number of points on the feature's edge. Features measured with the MeasureLogic probe can be measured explicitly as above, or by allowing the system to determine the feature type automatically based on the data cloud of points collected. Features can be probed manually or automatically as part of a part program using video edge detection or touch probes in CNC enabled systems. Probed features are shown in the Part View and entered into the Feature List. As mentioned earlier, the measurement of blobs is a special case requiring explicit measurements using a special tool.

The following feature measurements are supported by the M3 system:

- Points
- Lines
- Circles and arcs
- Slots and rectangles
- Patterns (teaching patterns for recognition in program playback)
- Distances
- Angles
- Planes
- Blobs
- Spheres
- Cylinders
- Cones

NOTE

Pattern recognition is only supported in CNC systems or in manual systems that also have the Digital Comparator option enabled.

Occasionally the system can misinterpret probed data points, particularly when there is not an even distribution of points collected along the entire feature edge. This can lead to feature identification errors. The Change Feature Type button can be used to change the type of a feature selected in the Feature List or Part View when identification errors occur. For example, a circle feature can be changed to an arc. The Change Feature button is not shown in the Measure toolbar when there are no alternative feature types for the selected feature.

Detailed feature information including coefficients, geometric form, fit type and the data cloud of points probed is always available in the Feature Detail screen discussed in Section 6: Feature List, Feature Detail and Part View Functions. A graphic drawing of the feature and the data cloud of points probed is also available in the Feature Detail screen. Tolerances can always be applied to measured features and are discussed in Section 8: Tolerancing.

Single Point Probing

Single point probing is performed using the video crosshairs or touch probes. Features are measured by probing one point at a time as the user aligns the probe on the feature edge until at least a minimum number of required points have been probed. The number of points required for single point probing is discussed in detail in *Points Required for Single Point Probing* contained in Section 3: Probes. Feature measurements become more accurate as the number of points probed increases beyond the minimum number required.

Multipoint Probing

Multipoint probing is performed using the MeasureLogic or EyeMeasure probe. Features are measured by touching a feature edge with the MeasureLogic probe or by pressing and dragging along the edge with the EyeMeasure probe. In either case, many points will be evenly distributed on the feature edge and collected automatically by the probe. Multipoint probes are discussed in detail in Section 3: Probes.

Probing Features

All features supported by the M3 system can be probed with any of the M3 probes with the exception of the blob, which will be discussed separately in the next few pages. The process of probing a feature with a given probe is identical for all features. Since the use of probes has been discussed thoroughly in Section 3: Probes and in the beginning of this section, a procedure for probing features with each probe type is provided here and should be sufficient for a complete understanding of probing features.

NOTE

Please refer to Points Required for Single Point Probing contained in Section 3: Probes for the minimum number of points required when probing features with crosshairs and touch probes.

Probing Features with the Crosshair Probes

To probe features with the crosshair probes:

1 Select the crosshair from the probe menu.
Measure toolbar > Probe button > Crosshair

2 Press any Measure Feature button in the Measure toolbar.
Measure toolbar > Desired Measure Feature button

3 Position the center of the crosshair probe on an edge within the live video window.

4 Press Enter or in the measure prompt window button to fire the probe and capture the point.
Measure toolbar > Enter button

5 Continue probing until the required number of points has been probed and then press the Done button to complete the measurement. The feature will be shown in the Part View window and added to the Feature List.
Measure toolbar > Done button

Probing Features with a Touch Probe

Straight down and star probes are used in the same way except for application specific probe tip and probing direction. To probe features with a touch:

1 Select the desired touch probe from the Probe menu.
Measure toolbar > Probe menu > Touch probe

2 Move the stage to make probe contact at the desired feature location. A point will be entered upon contact.

3 Probe the required number of points and then press the Done button to complete a feature measurement.
Measure toolbar > Done button

Probing Features with the Simple Active Crosshair probe

The simple active crosshair probe is a special application of the active crosshair. To probe features with this probe:

1 Select the simple active crosshair probe from the probe menu.
Measure toolbar > Probe button > Simple Active crosshair probe

2 Press any Measure Feature button in the Measure toolbar.
Measure toolbar > Desired Measure Feature button

3 Press an edge of the feature to be measured within the live video window. The probe will momentarily move to the edge and acquire a point.

4 Continue probing until the required number of points has been probed and then press the Done button to complete the measurement. The feature will be shown in the Part View window and added to the Feature List.
Measure toolbar > Done button

NOTE: It is also possible to set a direction and color sense for the Simple Active crosshair probe. Simply click the sizing handle to set the color sense(dark/light)and rotate the probe to achieve the desired direction. The probe will collect points along the crosshair axis that has the arrow handles on it. This allows for collection of specific points along an axis of edge scanning.

Probing Features with the MeasureLogic Probe

To probe features with the MeasureLogic probe:

1. Select the MeasureLogic probe from the probe menu.
   Measure toolbar > Probe button > MeasureLogic probe
2. Press any Measure Feature button in the Measure toolbar.
   Measure toolbar > Desired Measure Feature button
3. Press an edge of the feature to be measured within the live video window. The MeasureLogic probe will fire many points on the feature edge.
4. Press the Done button to complete the measurement. The feature will be shown in the Part View window and added to the Feature List.
   Measure toolbar > Done button

The MeasureLogic probe can also be used without designating a feature type first. The system will then detect the feature type automatically and complete the measurement, eliminating the need for steps 2 and 4.

Probing Features with the EyeMeasure Probe

The EyeMeasure probe is a special application of the MeasureLogic probe. The EyeMeasure probe is a special video tool for measuring complex or partially obscured edges. This probe is not selected as the other video probes are. It is drawn on the screen over the desired edge by the user. To probe features with the EyeMeasure probe:

1. Select the MeasureLogic probe from the probe menu.
   Measure toolbar > Probe button > MeasureLogic probe
2. Press any Measure Feature button in the Measure toolbar.
   Measure toolbar > Desired Measure Feature button
3. Press and drag your finger or the mouse pointer along the edge until the desired tool length and orientation is achieved.
4. Lift your finger or release the mouse and the tool will remain visible on the edge. The size and orientation of the tool can now be adjusted as described in Section 3: Probes.
5. Press inside the EyeMeasure probe to fire the probe and collect many points along the feature edge.
6. Press the Done button to complete the measurement. The feature will be shown in the Part View window and added to the Feature List.
   Measure toolbar > Done button

The EyeMeasure probe can also be used without designating a feature type first. The system will then detect the feature type automatically and complete the measurement, eliminating the need for steps 2 and 6.

NOTE

Auto point entry, focus assist and focus measurements can be used with all M3 probes to speed measurements and increase accuracy as described in detail in Section 3: Probes.

Probing and Measuring Blobs

Blobs are irregular shapes with a potentially complex perimeter. The M3 software measures blobs in the field of view with a special tool that captures an extremely high density of points. Two methods are used to measure blobs using the blob tool: pressing the blob edge or tracing the blob perimeter.

Press the Blob Edge

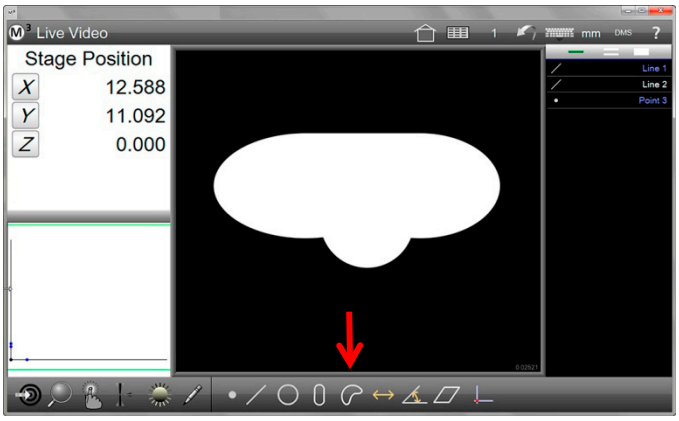
To measure a blob by pressing the edge:

1 Position the entire blob within the field of view.

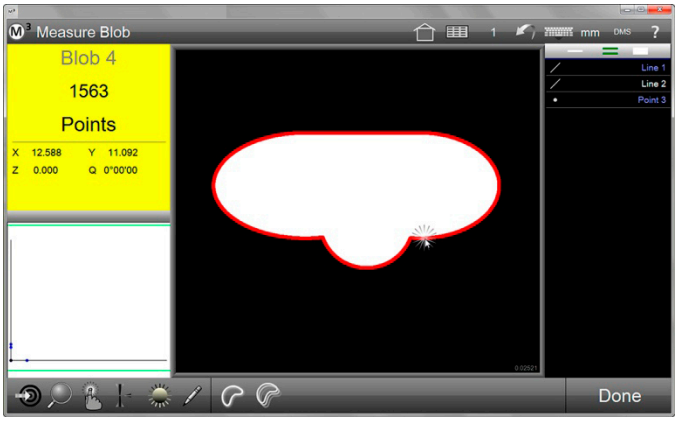
2 Press the Measure Blob button.

Measure toolbar > Measure Blob button

3 Press or click the blog edge. The blob tool will probe points on the perimeter of the blob.



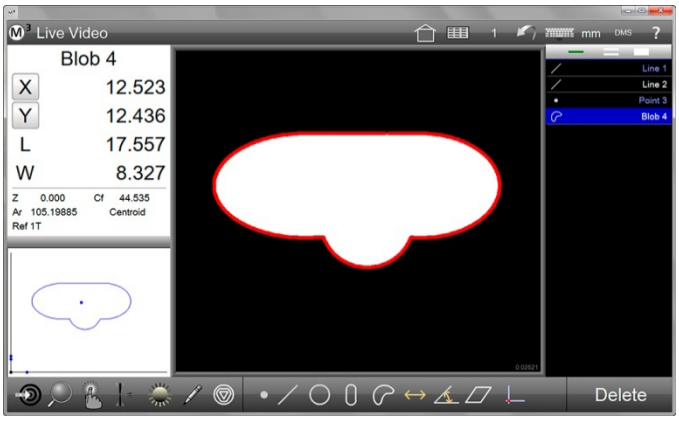
Press the Measure Blob button



Press the Blob Edge

4 Press the Done button to complete the measurement. The blob will be shown in the Part View and entered into the Feature List.

Measure toolbar > Done button



Trace the Blob Perimeter

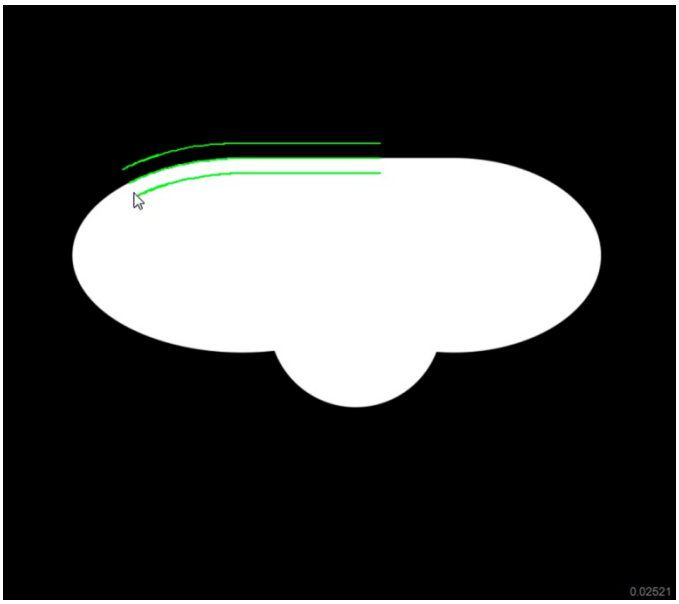
To measure a blob by tracing the perimeter:

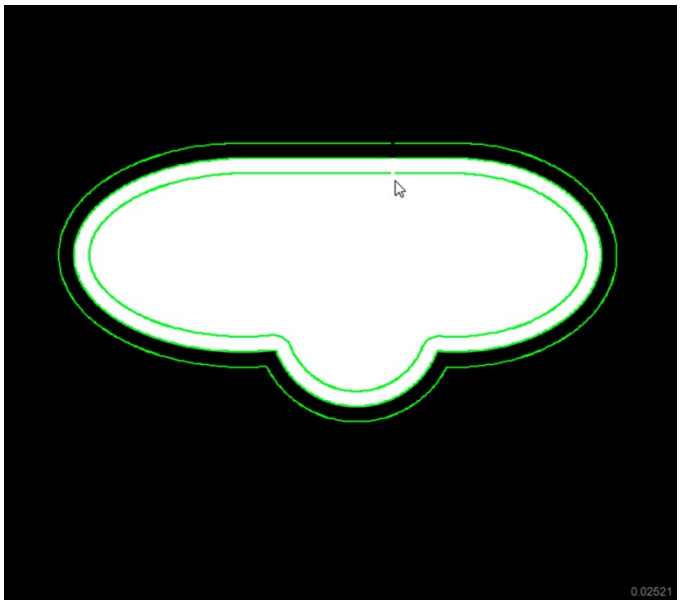
1 Position the entire blob within the field of view.

2 Press the Measure Blob button.

Measure toolbar > Measure Blob button

3 Press or click the blog edge and drag the blob probe around the entire perimeter. Be careful not to leave a gap on the perimeter where the probe beginning and end meet.

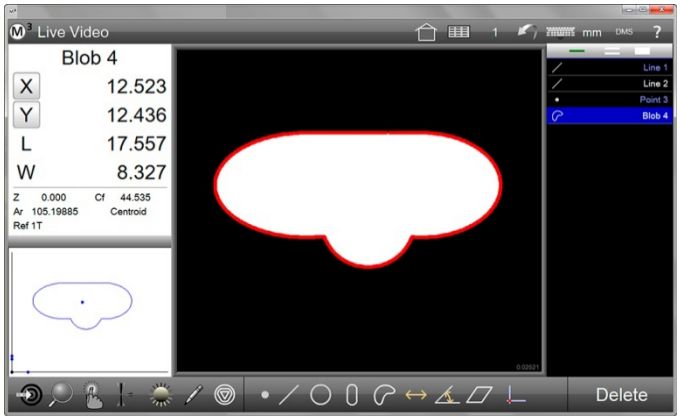




4 Press inside the probe boundaries to fire the probe and collect points.

5 Press the Done button to complete the measurement. The blob will be shown in the Part View and entered into the Feature List.

Measure toolbar > Done button



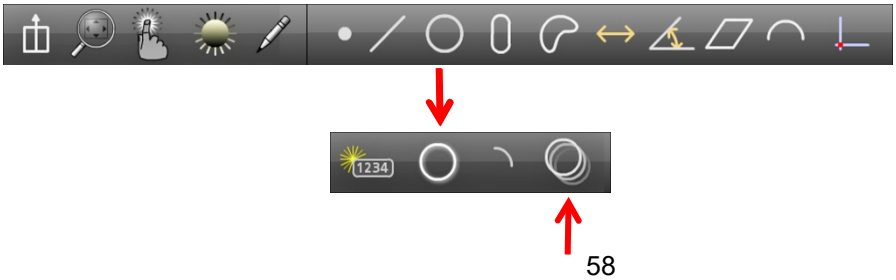
Repeat Mode Feature Measurements

When measuring in the repeat mode, the Measure Feature button does not need to be pressed between each feature measurement. The system prepares the next feature measurement automatically at the completion of the previous feature measurement. The repeat mode facilitates the measurement of a large number of like features sequentially. The repeat mode is available for all feature types using the following procedure:

1 Press the Measure Feature button for the desired feature type (a circle was selected in this example). The submenu for that feature type will be shown.

2 Press the Measure Repeat button for that feature.

Measure toolbar > Desired Measure Feature button > Repeat measure button

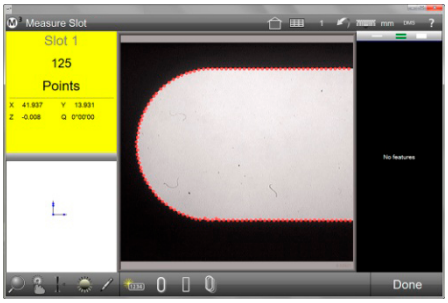


3 Select the desired video probe from the probe menu.
Measure toolbar > Probe button > Desired probe
4 Probe the first feature until the necessary points have been collected.
5 Press the Done button to conclude the first measurement. The new feature will appear in the Feature List and Part View. Continue measuring like features and then press the Done button to exit the repeat mode.
Measure toolbar > Done button

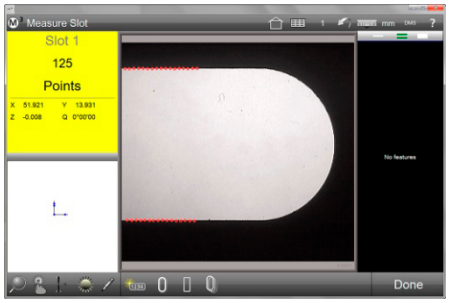
Probing Features that Extend Beyond the Field of View

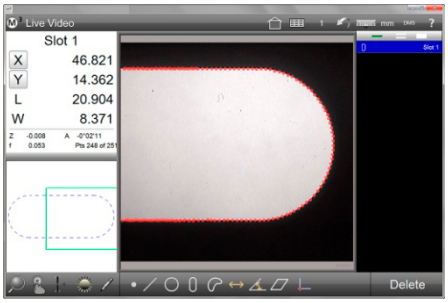
Features that extend beyond the field of view must be measured explicitly. To measure a feature that extends beyond the field of view:

1 Press the measure feature button for the desired feature type (a slot was selected in this example).
Measure toolbar > Desired Measure Feature
2 Select the desired video probe from the probe menu.
Measure toolbar > Probe button > Desired probe
3 Probe the portion of the feature that is included in the field of view.
4 Move the stage to position the remaining portion of the feature within the field of view and continue probing.
5 Press the Done button to conclude the feature measurement.
Measure toolbar > Done button



Probe the portion of the feature included in the field of view





Probe the remaining portion and press the Done button

NOTE

Features that extend beyond the field of view can also be measured using the image stitching function discussed in Section 5: Image Stitching. Image stitching is most useful when the feature extends far beyond the field of view.

Feature Decomp using the Multi-Measure tool

-Quickly measure groups of features of the same or varying type using the new Multi-Measure decomp tool.

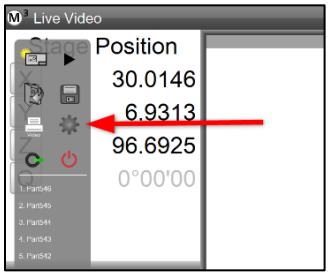
-Identify and measure features along a continuous edge contour with a single click on an edge or marquee drag a region of interest where "enclosed" features can be identified and measured.

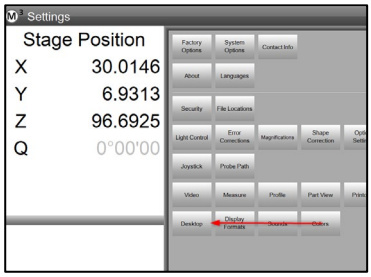
-A feature filter can be used to quickly specify which feature types should be measured, and which should be ignored.

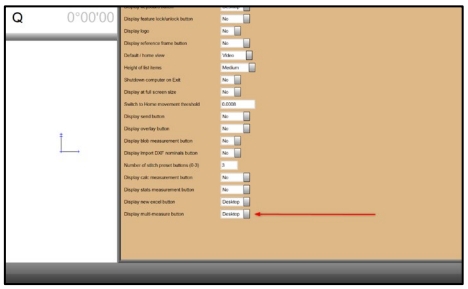
Setup:

IMPORTANT: The new Multi-Measure tool is OFF by default.

To enable the Multi-Measure tool navigate to the Desktop Settings Screen and set the "Display multi-measure button" flag to "Desktop". This will add the new Multi-Measure tool to the main measurement tool bar.





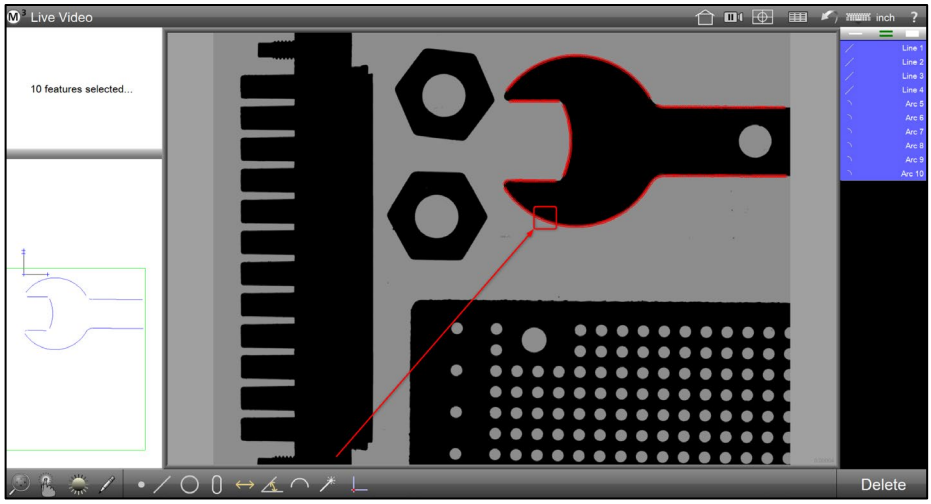


Operation:

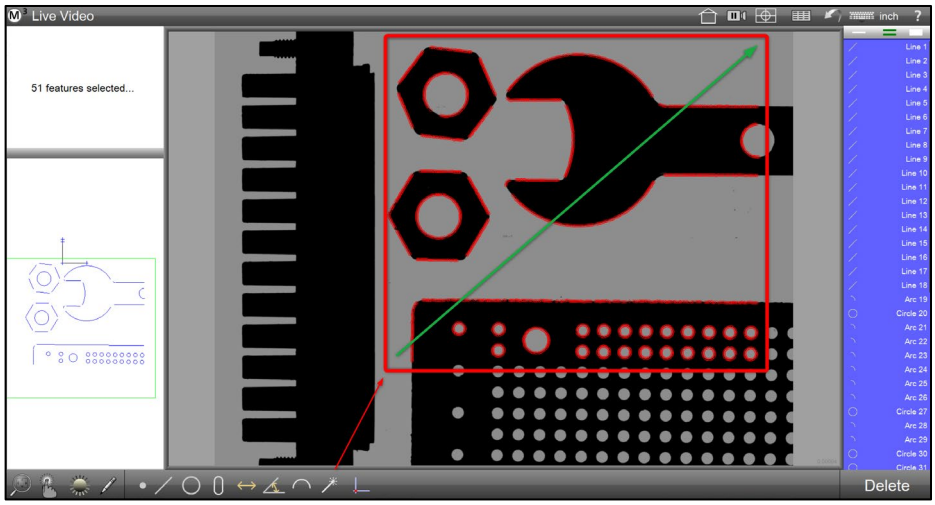
The Multi-Measure tool is a new measuring function that analyzes edges within a video image and "decomposes" edge contours into a group of standard fit feature results. The Multi-Measure tool is capable of identifying and measuring line, circle, arc, and blob features within a field of view or stitched image. Specialized algorithms within the probes firing code determine resulting feature types as well as the start and end position of feature data clouds, as one feature transitions into the next.

There are (3) different modes in which the Multi-Measure tool can be "fired":

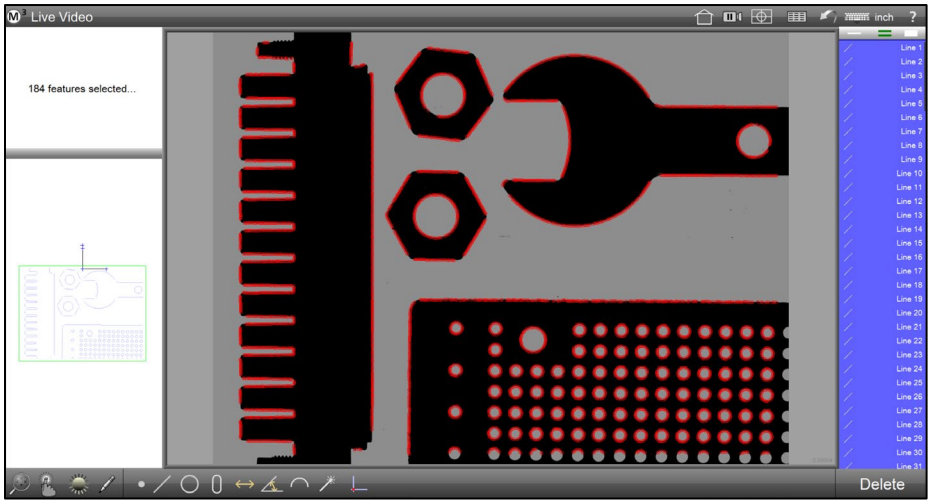
Single Edge Mode – Perform a single edge decomp by selecting the Multi-Measure tool from the bottom toolbar and then pressing on any edge in the Field of View. The tool will scan along the contour of the edge that was clicked on and return all detected features to the feature list.



Marquee Mode – Perform a marquee decomp by selecting the Multi-Measure tool from the bottom toolbar and then clicking and dragging a marquee box around a region of edges within the field of view. The tool will analyze all edges within the rectangular marquee region and return all detected features to the feature list.



Full Screen Mode – Perform a full screen decomp by selecting the Multi-Measure probe from the bottom toolbar and then pressing the Enter button, located in the bottom toolbar. The tool will analyze all edges within the current Field of View and return all detected features to the feature list.



Multi-Measure Tool Filters
In some cases when using the Multi-Measure tool it may be necessary to collect only specific feature types for a single execution of the tool. For this case the Multi-Measure tool filters can be used to specify which feature types will be measured, and which will be filtered. Filtered features are not added to the feature list.

Setting a Filter
Press the Multi-Measure tool button to expand the tools sub-menu, seen below. This menu contains "two state" toggle buttons for each feature to enable or disable filtering. When a feature button is pressed a "red slash" will appear indicating that features filter is enabled.

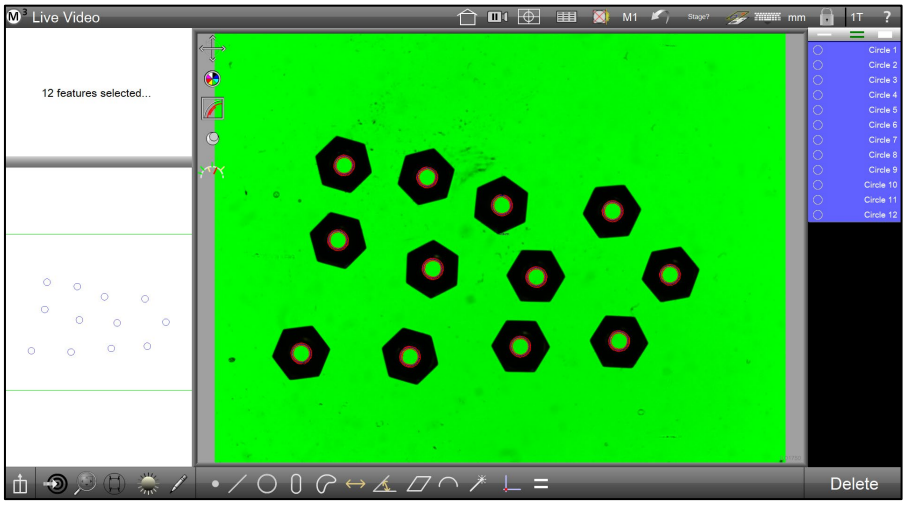
NOTE: The standard feature types(Circle, Arc, and Line) cannot be combined with Blob features when using Multi-Measure. As such the filter modes are mutually exclusive. When the Blob is un-filtered, the other feature types will automatically be filtered, and vice-versa.

Sample Procedure Steps for using the "Multi-Measure Tool" to measure multiple circles on a group of parts:

1. Press the Multi-Measure button located in the measure tool bar.

2. Press on the Filter Arc and Filter Line buttons in the Multi-Measure submenu to configure Multi-Measure to acquire circle features only.

3. Press the Enter button in the lower right toolbar to collect all of the circle features in the current image.



Pattern Recognition

The M3 pattern recognition function teaches specific image patterns to the system for automatic detection during program playback. Program playback can be improved using a soft-fixture based part alignment routine, utilizing this pattern detection mechanism. Patterns can be measured or taught as datum patterns where the skew and zero positions are set automatically, or as standard pattern features where the part or machine coordinate pattern position and orientation are reported for the pattern feature.

NOTE

Pattern recognition is supported in CNC systems or in manual systems that also have the Digital Comparator option enabled. Pattern recognition is also supported in non-CNC Field of View systems discussed in Section12: Field of View Functions.

Teaching a pattern

Select the desired pattern to be used for detection in playback. Ideal candidate patterns should contain image characteristics that are unique relative to the rest of the field of view in which they are taught. In addition, target pattern regions should contain reasonable light to dark pixel contrast and be relatively free of image noise and distortion.

To teach a pattern:

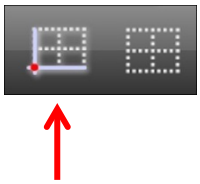
1 With the desired pattern in the field of view, press the Datum button in the Measure Toolbar to display the Datum Pattern Menu.

Measure toolbar > Datum button

2 Press a Pattern Probe to initiate one of two pattern measurements:

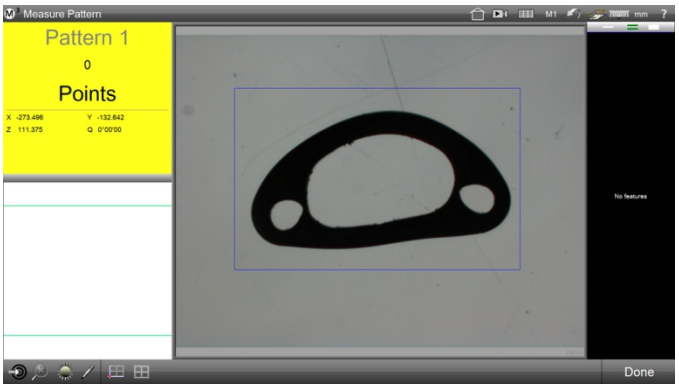
- Registration pattern measurement (default type)
- Standard pattern measurement

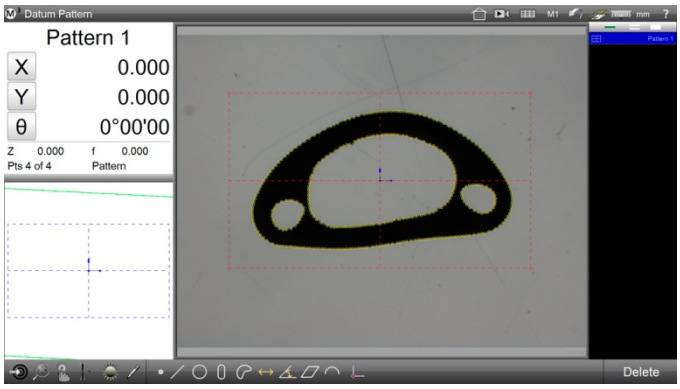
Registration Pattern Measurement



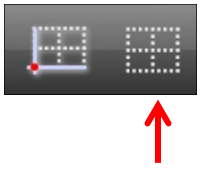
This is the default pattern measurement and is used to establish program registration for part playback. To measure (or teach) a registration pattern:

1 With the Measure Registration Pattern tool selected (default), press and drag to completely enclose the desired pattern. The pattern feature will be shown in the Feature List, and the pattern position and angle will be shown in the detail view. The pattern will be registered for Program playback as shown in the Part View, and other features can be probed.



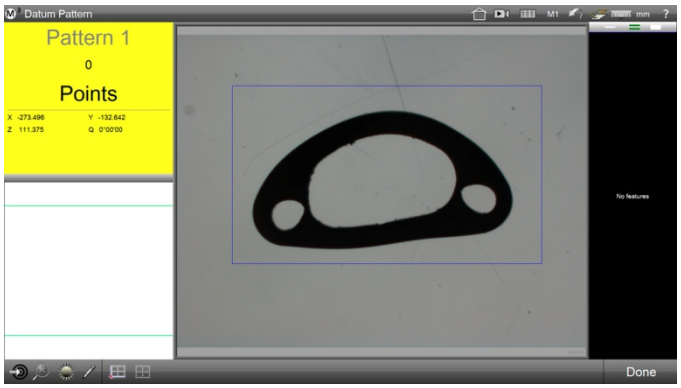


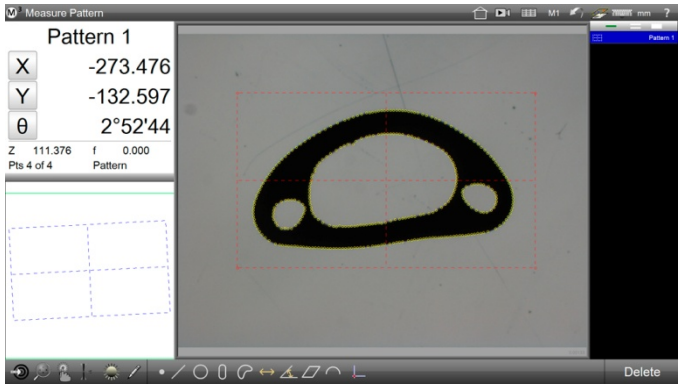
Standard Pattern Measurement



To perform a standard pattern measurement (teach):

1 Select the Standard Pattern Tool and then press and drag to completely enclose the desired pattern.





The pattern will be shown in the Feature List, and the current stage position and angle will be shown in the detailed view window.

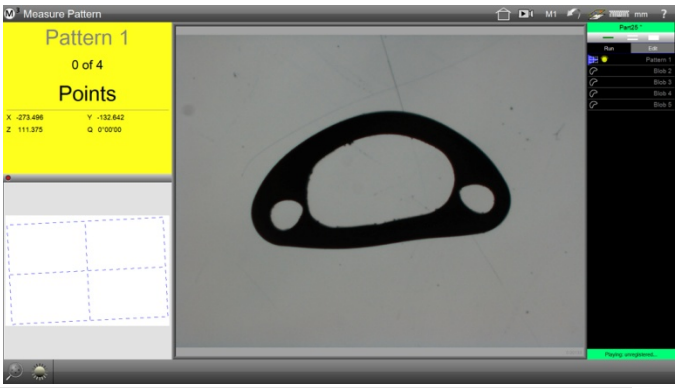
Note

Standard pattern features are typically used when a datum alignment will be constructed from two or more pattern features.

Pattern Program Playback

Once a registration pattern has been measured (taught) and other features have been probed, the measuring program can be played back. To play the program:

1 Position the pattern anywhere in the field of view, and then press anywhere in the Live Video window to execute the program. The program will recognize the registration pattern, register the coordinate system and measure any remaining features.



NOTES

In CNC-enabled systems, a message will be displayed asking for playback confirmation prior to any potential stage movement.

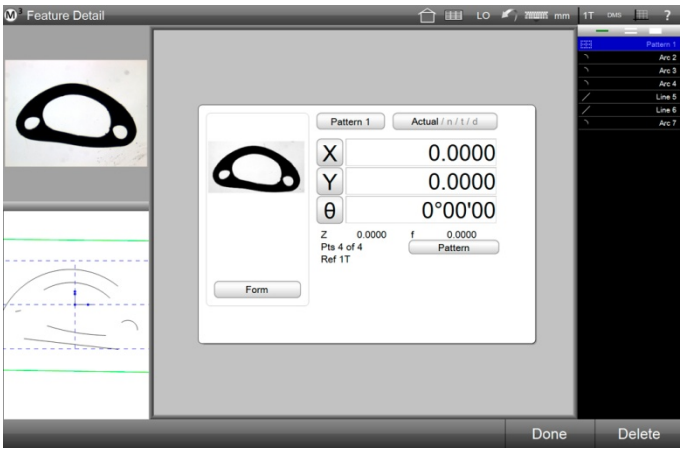
Important note for CNC-enabled systems

The angle of a given pattern is used as a temporary skew in program playback. Because the repeatability of the pattern angle is good only to 1°, subsequent features, particularly when located far from the initial pattern, may not be found automatically. To use the pattern finder most effectively in this case, choose an initial pattern that is located as close to the intended datum features as possible.

Viewing the Image Source for Pattern Features

For any measured (taught) pattern feature, the large feature detail screen can be used to view the originally sampled image of a given pattern.

Select the pattern feature from the Feature List that you would like to view, press in the small feature detail window at the top left of your screen to access the large feature detail window. The original image sample will be displayed for the selected pattern.



Goto Move Function

The Goto button prepares CNC enabled systems to perform one of three CNC Goto moves including:

- Goto position
- Goto feature
- Goto Part View location

Enabling the Goto Move Function

To enable the Goto Move function:

1 Press the Goto button. The button arrow will turn red and the M3 screen will display green control borders around the interface areas where Goto commands can be executed.

Measure toolbar > Goto button

Goto Position (Absolute/Relative)

To execute a Goto position CNC move:

1 Press the Goto button when the live video image is displayed in the large viewport.

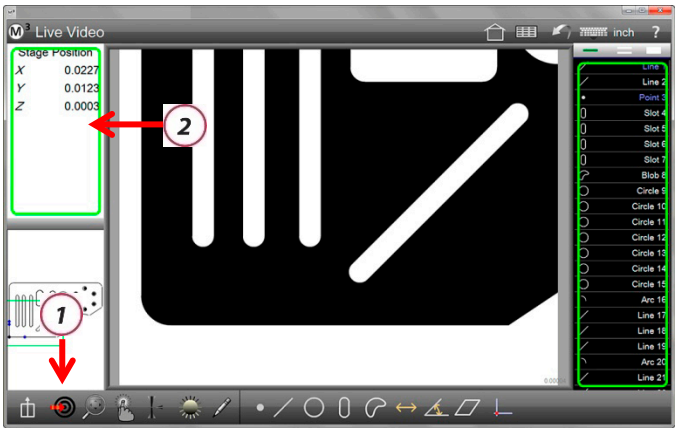
Measure toolbar > Goto button

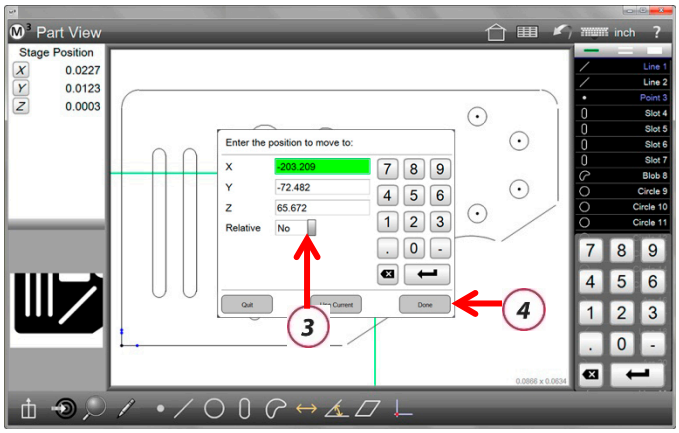
2 Press within the small DRO window, highlighted in green, to display the Goto position dialog.

3 Select the Goto type, absolute or relative, and enter the desired values for the X, Y, and Z axes.

4 Press Done to execute the Goto move.

Measure toolbar > Done button





NOTE

Goto absolute position executions are always recorded, and will be played back during program execution. These steps can be edited or deleted using the edit and delete program step functions from the program edit mode. Program editing will be discussed in detail in Section 11: Programming.

Goto Feature

1 Press the Goto button. The green Goto border will be displayed around the Feature List.

Measure toolbar > Goto button

2 Select the feature from the Feature List you would like to go to.

3 The system will execute a Goto move to the selected feature.

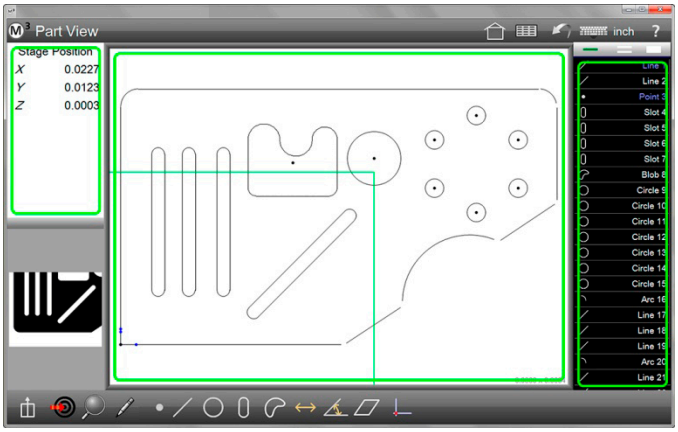
Goto Part View location

1 When the Part View is displayed in the large viewport, press the Goto button. The green Goto border will be displayed around the Part View space.

Measure toolbar > Goto button

2 Press or click within the Part View space in the desired location for the Goto click execution.

3 The stage move will be executed immediately, positioning the stage at the specified location.



Constructing Features

Part features are constructed from one or more parent features contained within the Feature List. The following feature constructions are supported by the M3 system:

- Points
- Lines
- Circles and arcs
- Distances
- Angles
- Planes
- Spheres
- Cones
- Cylinders

Features can be constructed from positional and non-positional parents. For the purposes of constructing features, the center points of positional features are used as points. The number of points required for the feature construction is identical to the numer required for feature measurement and is discussed in detail in *Points Required for Single Point Probing* contained in Section 3: Probes. Parent features include:

Positional features:

- Points
- Circles and arcs
- Slots and rectangles
- Blobs
- Patterns
- Spheres

Linear features:

- Lines
- Cylinders
- Cones

Other features:

- Planes
- Angles
- Distances

Point Constructions

Point constructions include:

- Average point from any combination of positional features
- Center point of circle or arc
- Start point, Center point, endpoint, or anchor point of a line. The anchor point is the point on the line closest to the datum.
- Start point and endpoint of a distance
- Apex point of an angle
- Midpoint between two positional features
- Intersection of two lines
- Intersection 1 or 2 between circle and circle, arc and arc or circle and arc
- Intersection 1 or 2 between line and circle or line and arc
- Point online perpendicular to a positional feature

- Tangent point 1 or 2 on circle from a point
- Centroid, +X, -X, +Y, -Y from a blob
- Center plane or sphere
- Midpoint of call or cylinder axis
- Apex of cone
- Intersection of cylinder axis and plane
- Intersection of cone axis and plane
- Perpendicular point positional feature and plane
- Intersection of three planes

Circle Constructions

Circle constructions include:

- Gauge circle between lines using parent lines and created circle (discussed later)
- Gauge circle between two arcs using parent arcs and created circle (discussed later)
- Gauge circle between three lines with circle tangent to each line
- Bolt circle from three or more circles
- Average circle from three or more circles
- Circle from three or more positional features
- Duplicate circle
- Circle from arc
- Equal circumference and equal area at Centroid, +X, -X, +Y, -Y from a blob
- Middle of cylinder or cone
- Intersection of cone and cylinder
- Intersection of cone and plane
- Intersection of cylinder and plane

Arc Constructions

Arc constructions include:

- Duplicate arc
- Arc from circle
- Arc from three or more positional features

Line Constructions

Line constructions include:

- Duplicate line
- Perpendicular line from line
- Gauge line from two parent lines and a created line (discussed later)
- Average of multiple lines
- By sector line and perpendicular by sector
- Line from 2 or more positional features
- Tangent line 1 and 2 between circles
- Tangent line 1 or 2 between point and circle
- Line from a distance
- Perpendicular line to a line through a positional feature
- Line parallel to a line through a positional feature
- Line tangent 1 or 2 to a circle and perpendicular to a line
- Line from a line and angle (rotated by the angle)
- Axis of cylinder or cone projected on any of three work planes

- Intersection of two planes projected on any of three work planes

Angle Constructions

Angle constructions include:

- Duplicate angle
- Angle of a line starting at + X direction
- Angle between lines: included, 180-, 180+ and 360-
- Taper angle of cone
- Cone Taper off to concentric circles at different heights and different radii

Distance Constructions

Distance constructions include:

- Duplicate distance
- Circumference of a circle
- Links of a line
- Distance between two angle apexes
- Minimum, center or maximum between circles or circle and arc or arcs
- Minimum, center or maximum between point and circle or point and arc
- Minimum, center or maximum between line and circle or line and arc (perpendicular distance)
- Minimum, center or maximum between lines
- Perpendicular from point to line
- Sum of two distances

Plane Constructions

Plane constructions include:

- Duplicate plane
- Least squares of best fit plane from three or more positional features
- A point aligned with XY
- Multipoint from points (fit plane)
- Multipoint from circles (fit plane)
- Two planes (mid-plane)

Sphere Constructions

- Duplicate
- Multipoint from points (fit sphere)

Cylinder and Cone Constructions

- Duplicates
- Multipoint fit from points
- Two circles define ends of cylinder or cone

Feature Construction Procedures

There are two procedures for feature construction:

- Gauge circle and gauge line constructions
- All other feature constructions

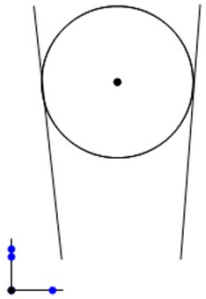
Construction Procedures for Gauge Circle and Gauge Line

Gauge circle and gauge line constructions include:

- Gauge circle between 2 lines
- Gauge circle between 2 arcs
- Gauge line between 2 lines

Gauge circle between two lines

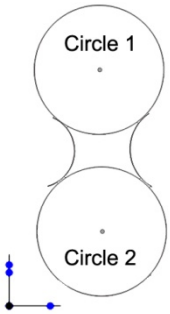
To construct a gauge circle between two lines:



1. Create a circle of the desired diameter (creating features is discussed immediately after this section). The diameter of the created circle must be larger than the minimum distance between the two lines.
2. Press the Measure Circle button.
   Measure toolbar > Measure Circle button
3. Select the desired two parent lines and the created circle from the Feature List.
4. Press the Done button to complete the construction. The gauge circle will be shown between the two parent lines in the Part View and will be added to the Feature List.
   Measure toolbar > Done button

Gauge circle between two arcs

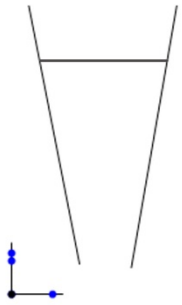
To construct a gauge circle between two arcs:



1. Create a circle of the desired diameter (creating features is discussed immediately after this section). The diameter of the created circle must be larger than the distance between the two arc perimeters.
2. Press the Measure Circle button.
   Measure toolbar > Measure Circle button
3. Select the desired two parent arcs and the created circle from the Feature List.
4. Press the Done button to complete the construction. Gauge circle 1 will be shown between the two parent arcs in the Part View and will be added to the Feature List. Press the Change Feature Type button to alternate between Gauge circle 1 and Gauge circle 2.
   Measure toolbar > Done button

Gauge line between two lines

To construct a gauge line between two lines:



1. Create a line of the desired length (creating features is discussed immediately after this section). The length of the created line must be longer than the minimum distance between the two parent lines and must be horizontal or vertical.
2. Press the Measure Line button.
   Measure toolbar > Measure Line button
3. Select the desired two parent lines and the created line from the Feature List.
4. Press the Done button to complete the construction. The gauge line will be shown between the two parent lines in the Part View and will be added to the Feature List.
   Measure toolbar > Done button

Construction Procedures for all Other Features

All feature constructions (except for gauge circles and gauge lines described earlier) use the following procedure:

1 Press the Measure Feature button of the feature type that you wish to construct.
Measure toolbar > Measure Feature button

2 Select (highlight) the required parent features from the Feature List.

3 Press the Done button to complete the construction. The constructed feature will be shown in the Part View and entered into the Feature List.
Measure toolbar > Done button

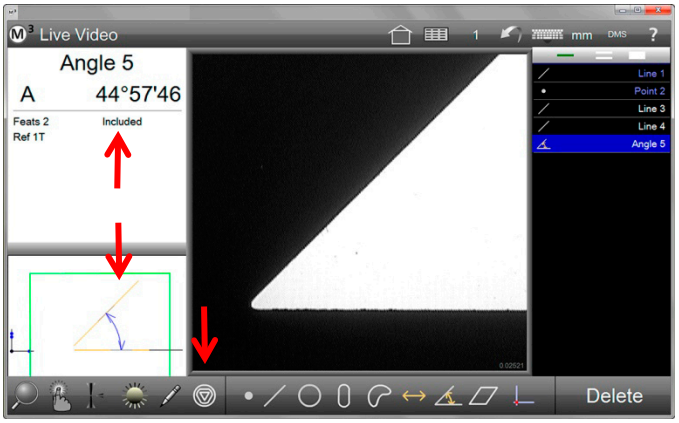
Changing Constructed Feature Type

Many constructed features can have more than one feature type. For example, an angle constructed between two lines can be any one of 4 types: included, 180-, 180+ and 360-. To change the feature type:

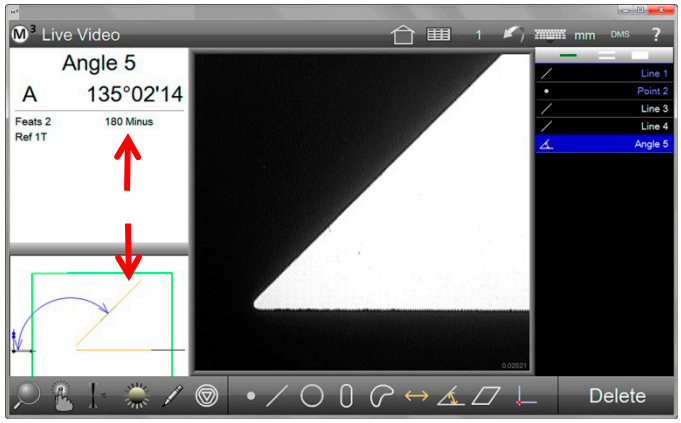
1 Select the constructed feature in the Part View or from the Feature List.

2 Press the Change Type button and note the feature type description shown in the Feature Detail viewport in the upper left portion of the screen.
Measure toolbar > Change Feature Type button

NOTE
The Change Feature Type button is shown in the Measure toolbar when feature type alternatives exist. If the button is not present in the Measure toolbar, only one type of feature exists for the selected construction.



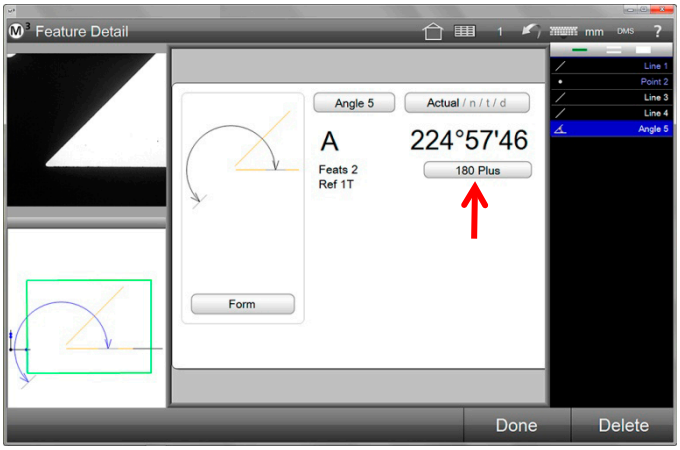
Note the feature type, then press the Change Feature Type button

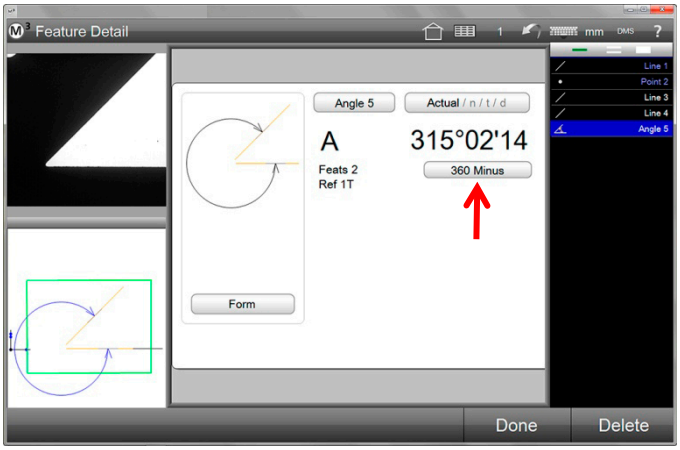


The feature type will change

3 The feature type can also be changed in the Feature Detail viewport. Press in the small feature detail viewport in the upper left portion of the screen to display the Feature Detail viewport.

4 Press the Feature Type button to change the feature type. Pressing the Feature Type button repeatedly will cycle through all of the available feature types.





Creating Features

Features can be created by entering the desired position size and other parameters that describe the feature. The M3 system supports the creation of the following features:

- Points
- Lines
- Circles
- Arcs
- Slots
- Rectangles
- Distances
- Angles
- Planes
- Spheres
- Cylinders
- Cones

The basic procedure for creating features is identical for all feature types, only the required feature parameters change depending on the feature type being created. To create a feature:

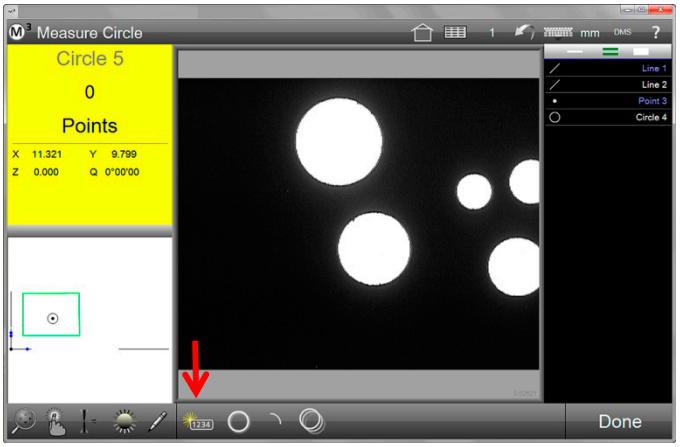
1 Press the Measure Feature button of the feature that you wish to create. The Measure Feature submenu will be displayed. In this example, a circle will be created.

Measure toolbar > Measure Feature button

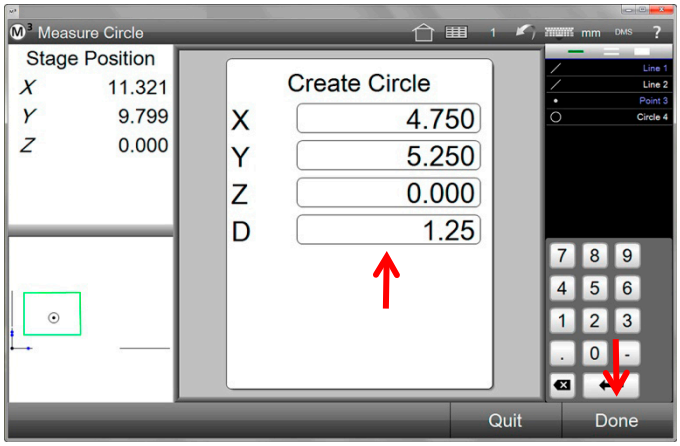
2 Press the Create Feature button. Data fields will be displayed for entering the required feature parameters.

3 Enter the required feature parameters and then press the Done button to complete the creation. The new feature will be shown in the Part View and added to the Feature List.

Measure toolbar > Done button



Press the Create Feature button in the Measure Feature submenu



Enter the required feature data and then press Done

Formula Results via Calc Feature

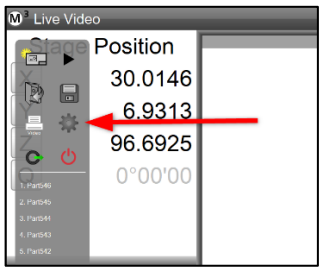
-Added new "Calc" feature type to support formula-based calculations inside the M3 software.

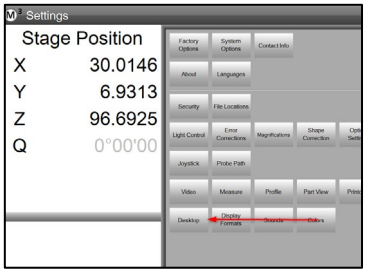
-Generate custom formula calculations utilizing measurement result data and standard mathematical operators and trigonometric functions.

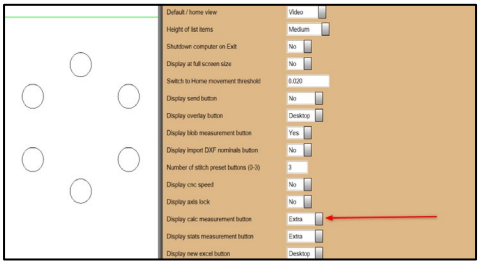
-Perform simple and complex measurement calculations directly in the M3 software.

Setup:

IMPORTANT: The new Formula Calc functionality is OFF by default.
To enable the Formula Calc functionality navigate to the Desktop Settings Screen and set the Display Calc Measurement button flag to "Desktop" or to "Extra" depending on whether you would like the new button added to the desktop toolbar or to the Extra menu.

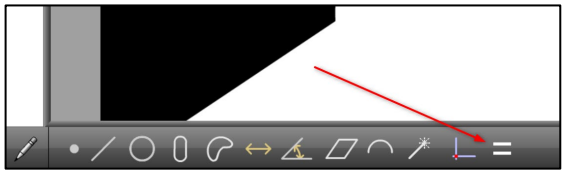






Operation:
The Calc feature is used to solve complex measurement tasks which may require combining coefficients from several different features or by extracting a feature coefficient from a feature and then applying the power of an arithmetic/geometry calculator.

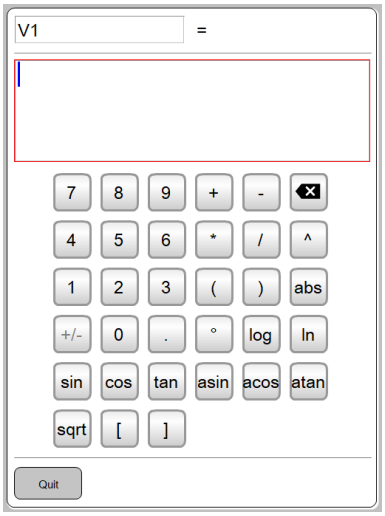
➔ When enabled, a new button will be added to the bottom toolbar(or Extra Menu) of the main M3 desktop. This button will be used to access the Formula Calc Feature.



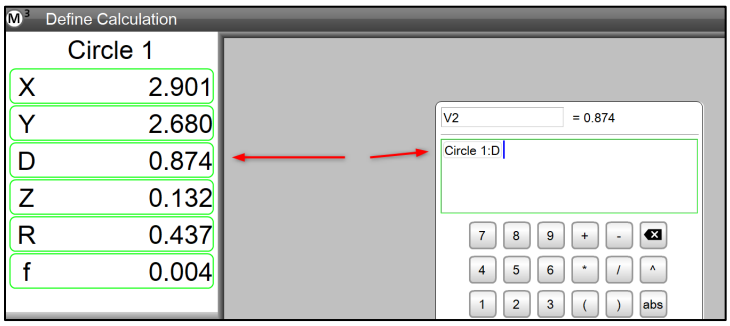
Sample Procedure Steps for the "Formula Calc" functionality:
(The formula for calculating the circumference of a circle is (Pi * Diameter)

1. Measure a circle feature.
2. Select the Calc button from the bottom toolbar.

3. The Calc feature formula entry screen will be displayed.



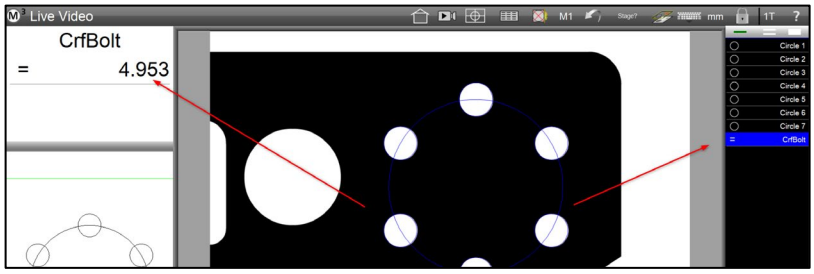
4. Select the Circle feature in the feature list. A list of selectable coefficients will appear in the upper left viewport.
5. Select Diameter coefficient from the upper left viewport.



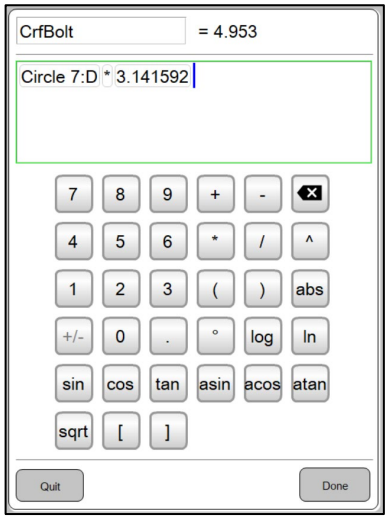
6. Press the Multiplication operator and then the Pi button from the Calc keyboard.
7. Press the Done button in the lower right toolbar to complete the formula calculation.
8. The Calculation result will be added as a calc feature to the feature list.
9. The Calc feature result can also be renamed, toleranced, and reported. See Example Procedure step #6 of the previous section for instruction on doing so.

Editing the formula for a Calc Feature Results:
→The formula used to generate a Calc feature result can always be edited in either interactive or program edit mode.

1. To edit a Calc result formula, select the calc feature in the feature list and then long press, using the left mouse button, on the calc result view port, located in the upper-left detail viewport.



2. The Calc result formula entry screen will be displayed. Modify the formula as desired and press done to finish editing the Calc feature formula.



Section 5: Image Stitching

Image stitching is a method of scaling super images that greatly extend beyond the field of view so that they fit within the field of view and can more easily be measured with pattern and other probes. Image stitching is included in manual and CNC encoder-based systems. Image stitching is used to:

- Scale and fit large images into the field of view
- Use DXF overlays on (scaled) super image parts
- Collect (scaled) super images within traditional part programs for use in conjunction with pattern recognition for locating difficult edges and features that vary greatly from part to part

Stitching a Super Image

Super images can be stitched using four methods:

- Rectangular stitching
- Free form stitching
- Preset stage stitching

Enabling Image stitching

Before using one of the image stitching methods, image stitching must be enabled. To enable image stitching:

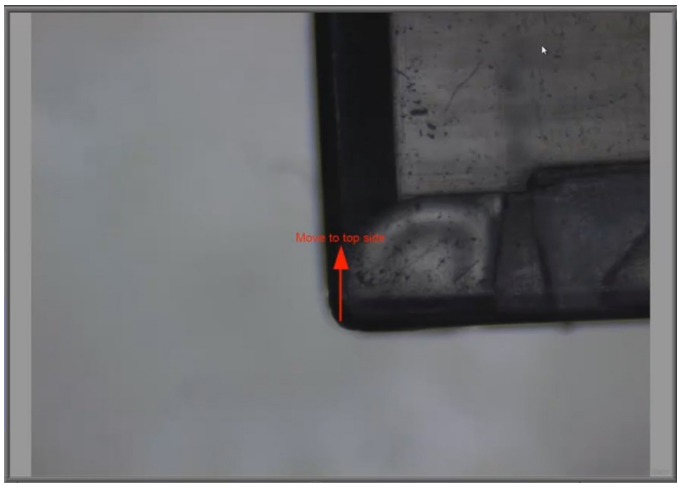
1 Click the Stitch Mode button. The Image Stitch toolbar will be displayed at the bottom of the screen. The default method of rectangular stitching will be enabled.

System toolbar > Stitch button

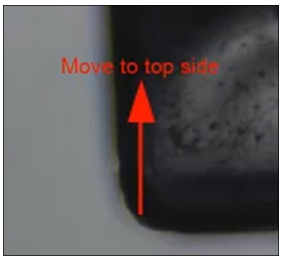
Rectangular Image Stitching

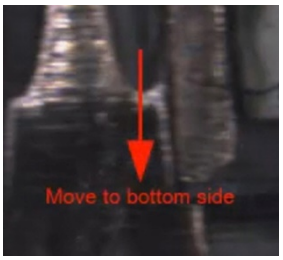
To perform rectangular stitching:

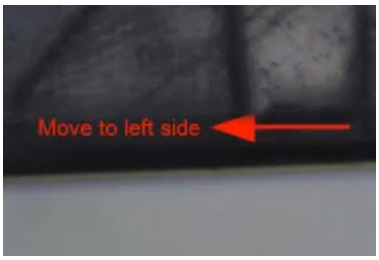
1 Press the Rectangular Stitch button to enable rectangular stitching. A red guidance arrow will be shown in the video window.



2 Move the stage as directed by the red arrow to the four bounding corners of the rectangular image area. Press the screen or Enter at each corner. The Stitch dialog will be displayed when the image is enclosed.

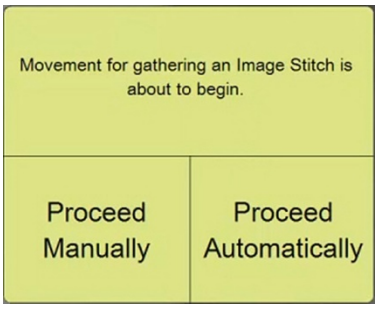


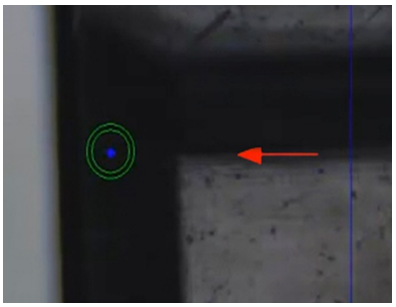




3 Press Proceed Automatically for CNC image stitching or Proceed Manual for manual stitching.

4 When manual stitching is selected, follow the user guidance arrows to store the stage locations. Move the stage to insert a red arrow into the green target circle. CNC systems will perform this task automatically.





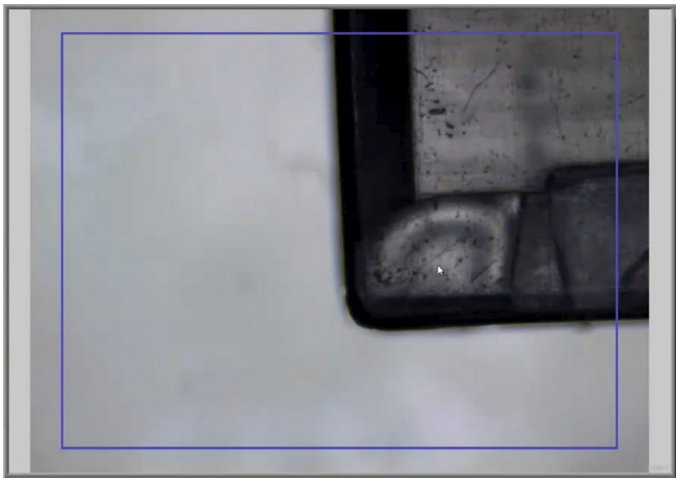
5 The final stitched image will appear in the live video and part view windows. The video image can now be probed and measured.

Free Form Image Stitching

To perform free form stitching:

1 Press the Stitch Free button

2 Move the stage to position the stitch frames (shown in blue) at the desired locations selecting all or only portions of the image you wish to stitch. Press the screen or Enter button at each frame location. The frames are not required to be adjacent. Frames need only enclose areas that you wish to measure. When the frames enclose the image, the Stitch Dialog will be displayed as in rectangular stitching described above.



3 Press Automatically for CNC image stitching or Manual for manual stitching.

4 When manual stitching is selected, follow the user guidance arrow to store the stage locations. Move the stage to insert a red arrow into the green target circle. CNC systems will perform this task automatically.

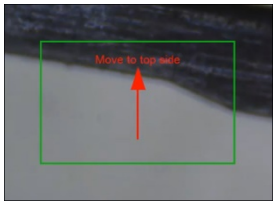
5 The final stitched image will appear in the live video and part view windows. The video image can now be probed and measured. In this example, only portions of the super image that are important to the desired feature measurements were collected and stitched (areas shown in blue were not collected).

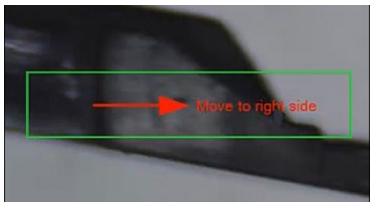
Preset Stage Stitching

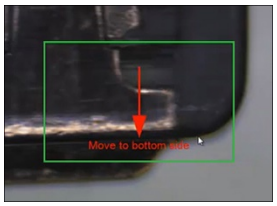
Three color coded stage areas can be stitched by the user using Preset Stage tools. Used in conjunction with pattern recognition, each of these three stage areas can be used by the system for temporary part fixturing. To perform preset stage stitching:

1 Move the stage to the first frame location.
2 Long-press the colored rectangle on the screen to store the first stage location of the stitched zone and begin the storage sequence (green in this example).

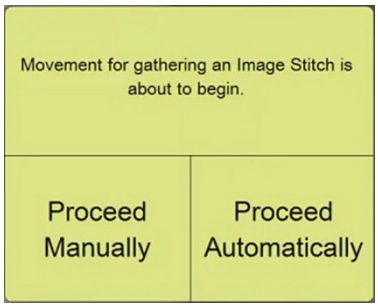
3 Follow the red arrow which is now enclosed by the colored preset rectangle. Press in the preset box or press Enter to store each preset location boundary on the stage. When the presets have enclosed the image, the Stitch dialog will be displayed.

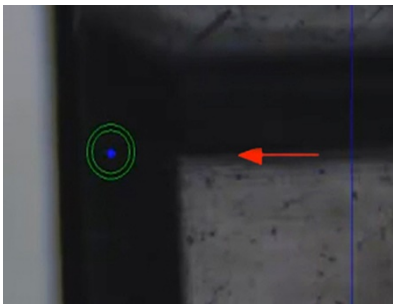






4 Press Proceed Automatically for CNC image stitching or Proceed Manually for manual stitching.
5 When manual stitching, follow the user guidance arrow to store the locations. Move the stage to insert a red arrow into the green target circle. CNC systems will perform this task automatically. Now a stitched image can be constructed from camera images of the colored preset stage locations.





6 Later, to stitch a super image located within the preset stage location defined by the steps above, press the Stitch button to enable stitching and then press the appropriate colored preset button (green in this example).
System toolbar > Stitch button
Stitch toolbar > Preset Color button
7 The Stitch dialog will be displayed. Select Proceed Manually or Proceed Automatically to collect and stitch images.

Opening a DXF Overlay over a Stitched Image

DXF overlays can be superimposed over stitched images within the field of view. To display a DXF overlay over a stitched image:

1 Register the super image part and perform feature measurements that will define its location and orientation. The super image part registration should match the registration of the DXF part file that will be used as an overlay.

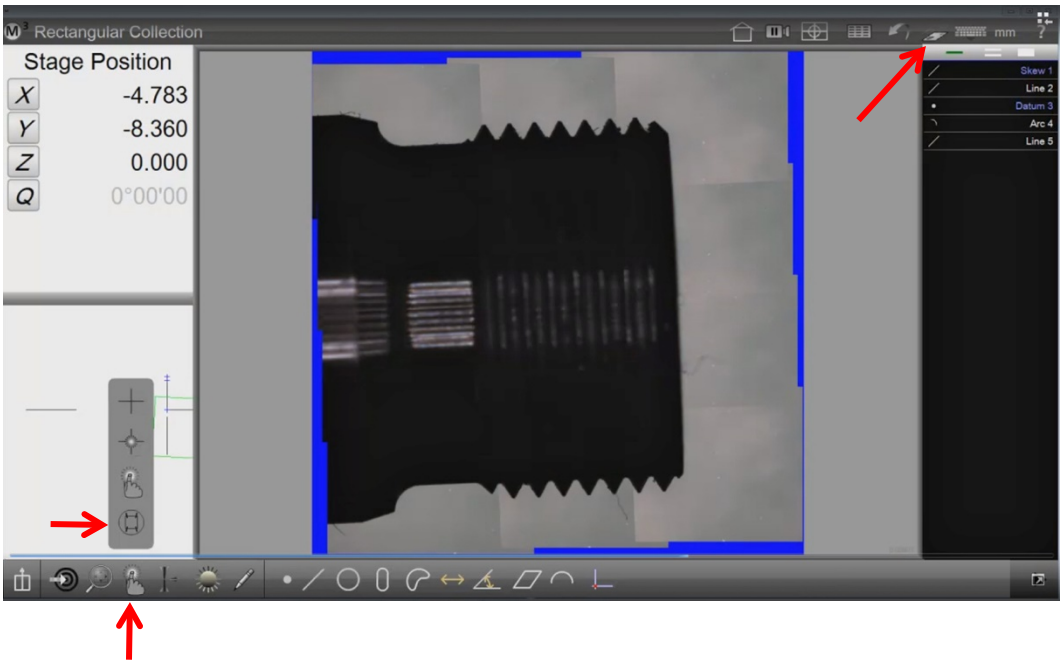
2 Stitch the super image of the part using one of the stitching methods described earlier.

3 Press the Feature Overlay button to enable the overlay function if necessary.

System toolbar > Feature Overlay button

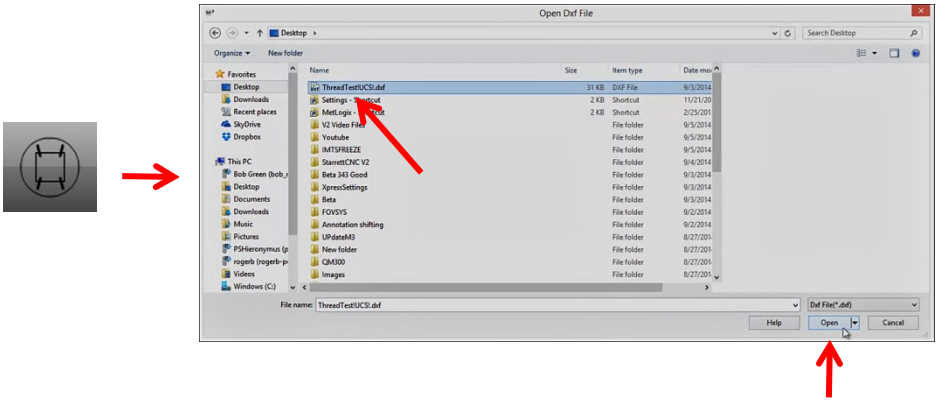
4 Select the Digital Comparator probe.

Measure toolbar > Probe menu > Digital Comparator probe button

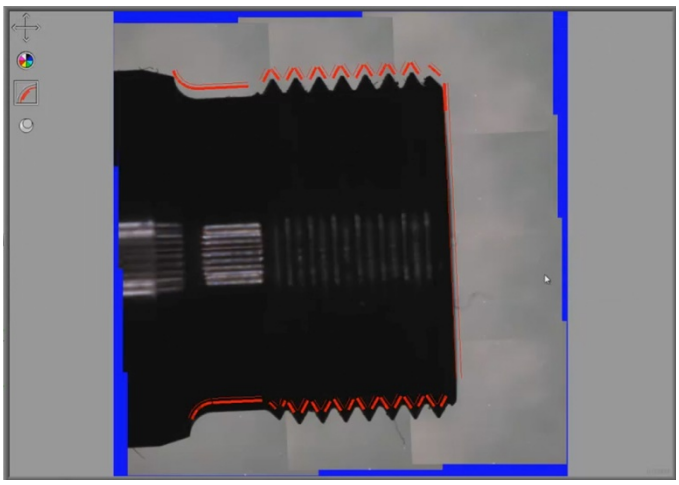


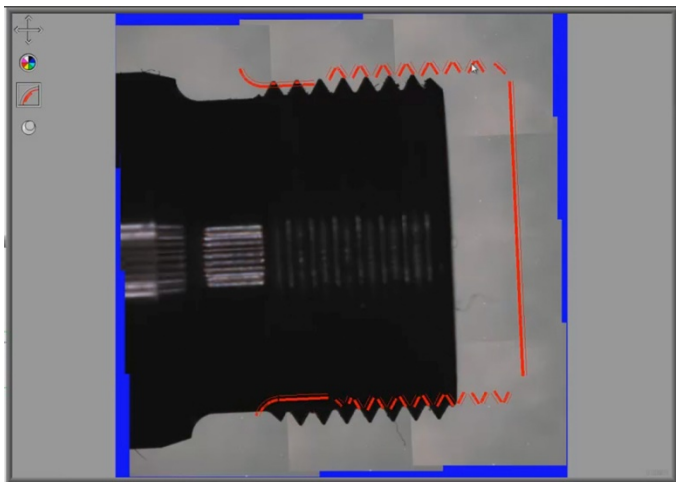
5 Long-press the Digital Comparator probe button to display the Open DXF File dialog.

6 Navigate to and select the desired DXF file.



7 Press Open to load the DXF file. The DXF overlay will be shown in red over the stitched image.
8 Press and drag the DXF overlay to the desired location.

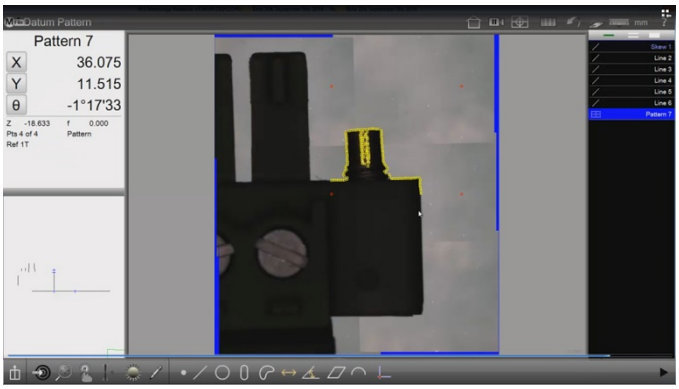


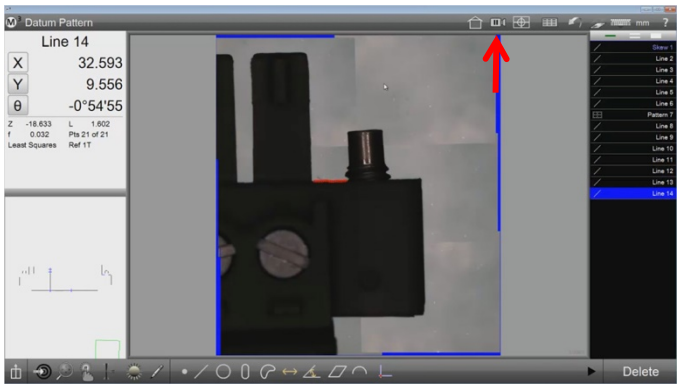


Stitching a Super image Within a Part Program

Stitching a portion of a super image within a part program is a convenient way to apply pattern recognition so that nominal values of a high variation feature area can be applied in program playback. To include stitching in a program:

1 Perform super image measurements that define the part location and orientation at the beginning of the program.
2 Stitch an area bounding the high variability features of interest using one of the methods described earlier.
3 Measure a pattern to register the area of high variability features within the stitched image.
Measure toolbar > Datum button > Pattern measure button
4 Measure features that define the highly variable part geometry within the stitched pattern area.
5 Press the Freeze Live Video button to release the stitched image and continue overall part measurements.





NOTES

While the stitched image is displayed, the registration created by teaching the pattern will be a reference for all measurements performed within the stitched image.

When the stitched image is released, the original part registration will be restored for subsequent feature measurements.

Section 6: Feature List, Feature Detail and Part View Functions

The Feature List, feature detail and Part View are the primary means of viewing and manipulating features that have been measured, constructed or created in the M3 software. The Feature List facilitates feature construction, provides access to the feature detail views and provides a means of selecting and deleting features.

The feature detail screens display all dimension, position, form and tolerance information for features selected in the Feature List or in the Part View. A feature graphic is also displayed that shows the data cloud of probed points. The feature detail also provides the means to change data fit and feature type.

The Part View is a graphical display of all measured, constructed or created features in the current reference frame. The Part View can also contain additional graphic elements such as feature annotations, a reference frame indicator and grid lines. The Part View supports feature selection for constructions using the same simple method as the Feature List. Feature selection from the Part View also provides a means of dimensioning features. A simple set of tools are provided for zooming and panning in the Part View. For touch screen enabled systems, pinch zooming and two finger panning are also supported.

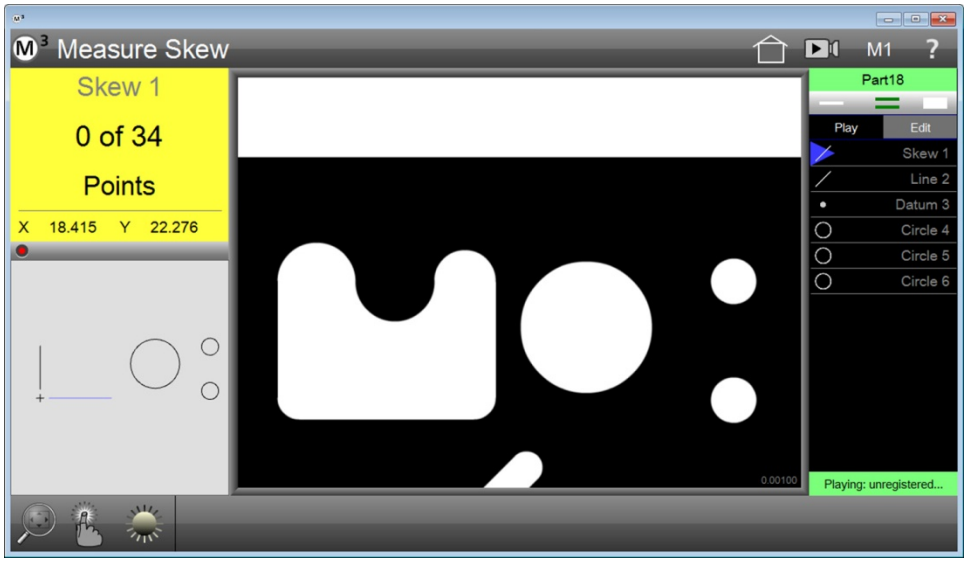
The Feature List

The Feature List always appears at the right side of the screen and changes slightly to support the current user activities. The primary functions of the Feature List include:

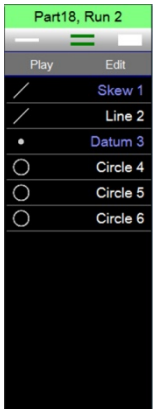
- Displaying part name and run numbers
- Displaying features
- Selecting features
- Deleting features
- Displaying program steps in the programming mode

Displaying Part Name and Run Numbers

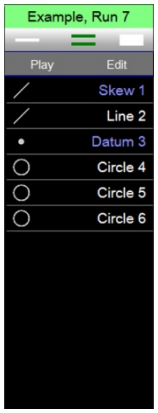
The top of the Feature List displays a generic part name when a program is run for the first time. A run number is also displayed when the program is run again. Run numbers are incremented in subsequent program executions. After the program is saved, the generic program name is replaced by the saved name.



Generic part name after first run



Generic name and run number



Saved name and run number

Displaying Features

All features that have been measured, constructed or created are displayed in the feature list with their tolerance status indicated by color. Features that have passed tolerance tests are shown in white and features that have failed tolerance test are shown in red. Datum features such as levels, skews and datum points are shown in blue. These color representations of tolerance status and datum features are also shown in the Part View.

Scrolling the Feature List

When the feature list contains too many features to be displayed simultaneously on the screen, press and drag the feature list up or down to display features that are beyond the field of view.

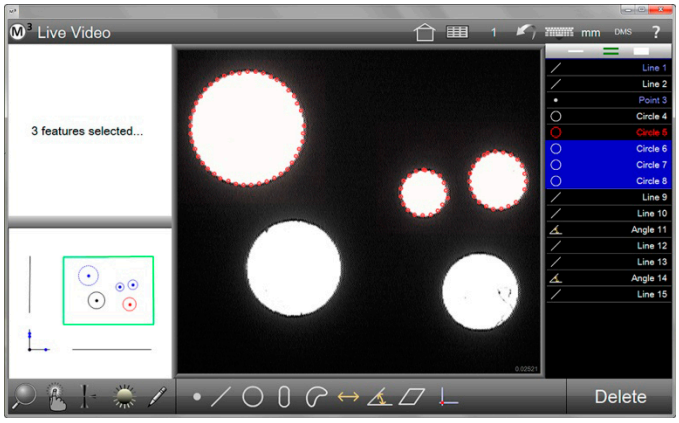
Selecting features

Features are selected and deselected individually or in groups by pressing or clicking features. The Feature Selection tool located at the top of the feature list determines the number of features that can be selected either sequentially or simultaneously as follows:

- Press the narrow single bar for individual feature selection or de-selection
- Press the double bar for multiple feature selection or de-selection in sequence
- Press the wide single bar for group feature selection as a range. In addition, the wide single bar allows all features to be selected simultaneously.

Feature Selection tool

When features are selected, they are highlighted in blue in the feature list, the original probed points are shown in the live video window and the selected features are highlighted in blue in the Part View.

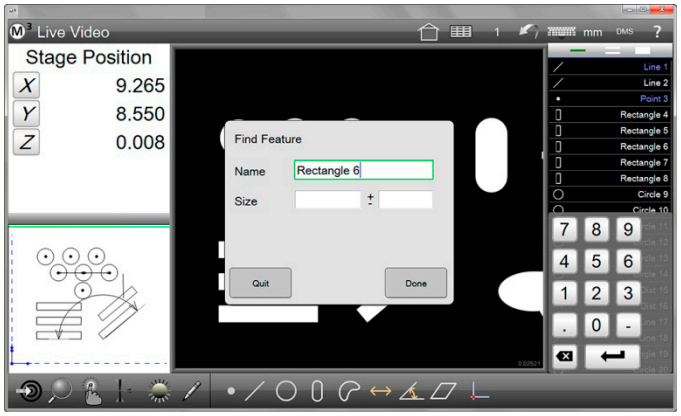


Selected features are highlighted in blue

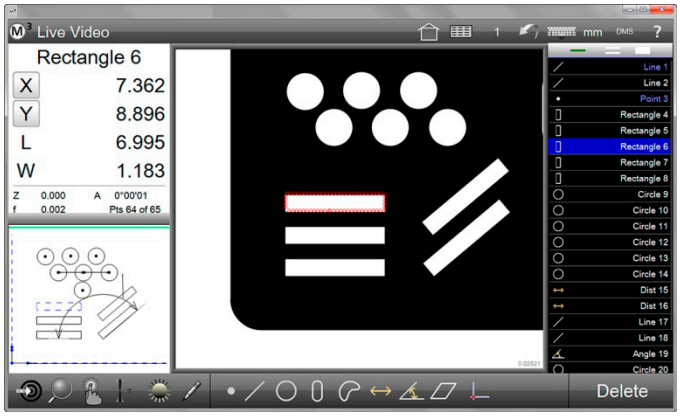
Feature Finder

The Feature Finder is a special application of the Feature Selection tool. The Feature Finder is used to find and select specific features by name, groups of features with similar names or groups of like features that meet certain minimum dimensional criteria. To use the Feature Finder:

1. Long press the Feature Selection tool to display the Find Feature dialog
2. To find a single specific feature, enter the full feature name into the Name field and then press Done. The specified feature will be highlighted in the feature list.

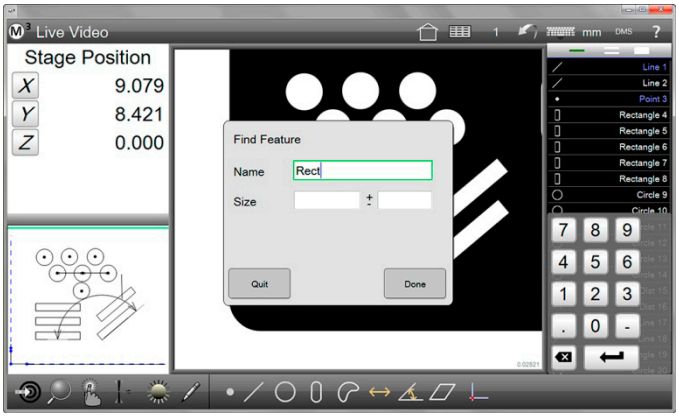


Enter the full name of a specific feature

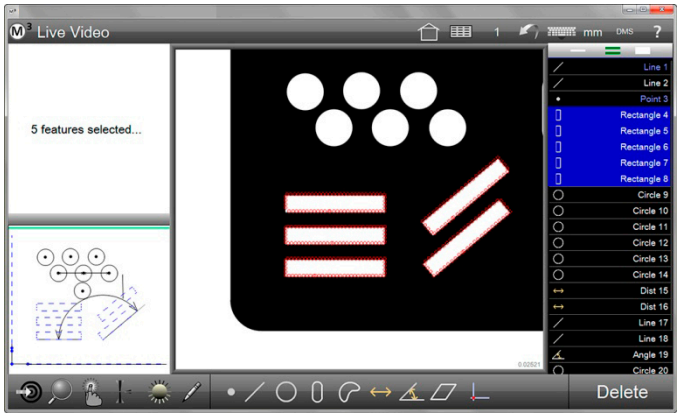


The specified feature will be highlighted in the Feature List

3 To find a group of features with similar names, enter the first part of a feature type name into the Name field and then press done. The specified group of features will be highlighted in the feature list.



Enter the first part of a feature type name

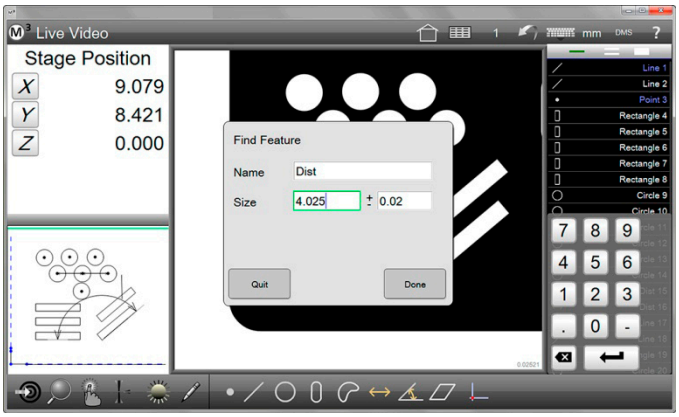


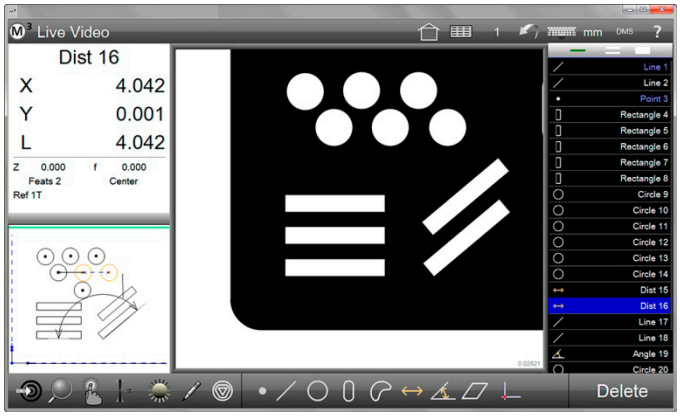
The specified group will be highlighted in the feature list

4 To find a group of like features that meet certain dimensional criteria, enter the feature type name into the Name field and then enter the size and variation criteria into the Size and ± fields. Size and variation criteria apply to circles, arcs, lines and distances as follows:

- Circles: size refers to diameter
- Arcs: size refers to radius
- Lines: size refers to length
- Distances: size refers to length

5 Press done to complete the find operation. In this example, the system was asked to find a distance having a size (length) of 4.025 ± 0.02. Only one of the two distances in the feature list met this criteria and that distance is highlighted.





Deleting Features from the Feature List

When one or more features are selected and highlighted in the Feature List, a Delete button is shown in the Measure Toolbar. To delete one or more highlighted features:

1 Select the features that you wish to delete

2 Press the Delete button. The delete key on the keyboard may also be used.

Measure toolbar > Delete

OR

Computer keyboard > Delete

To delete all the features of the Feature List:

1 Press and hold the Delete button.

Measure toolbar > Delete

NOTES

Features can also be deleted in the program edit mode discussed later in Section 11: Programming.

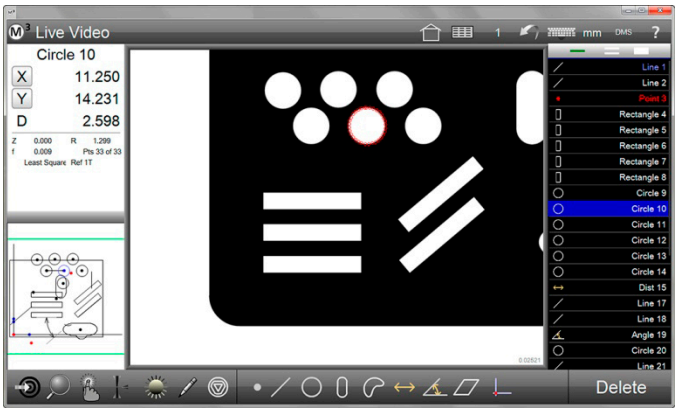
Features that have been deleted can be restored by pressing the Undo button in the System toolbar.

Feature Detail screen

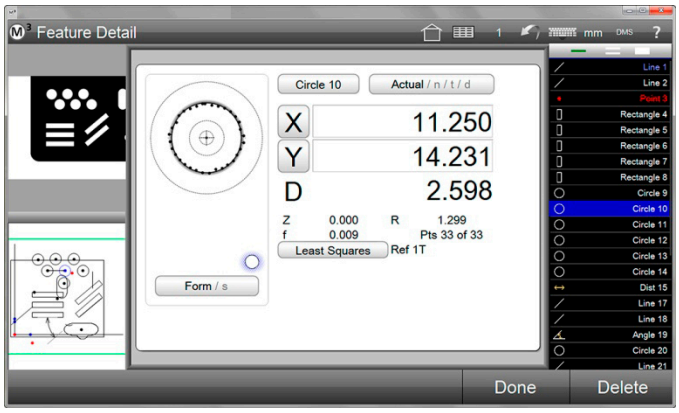
The Feature Detail screen displays the measurement results for the feature selected in the Feature List or in the Part View. A graphic view of the feature is displayed that shows the data cloud of probed points. The feature detail screen also provides access to tolerancing and datuming functions and the means to change data fit and feature type. Tolerancing is discussed in detail in Section 8: Tolerancing. Datuming was discussed in Section 4: Measuring, Constructing and Creating Features, and will be discussed again in Section 7: Reference Frames.

To access the Feature Detail screen:

1 Select the desired feature in the Feature List and then press the DRO/Feature Detail viewport in the upper left portion of the screen. The Feature Detail screen will be shown in the center viewport.



Select a feature and then press the Feature Detail viewport



The Feature Detail screen will be displayed

As shown in this example of a Feature Detail screen for a circle, feature parameters are shown that includes:

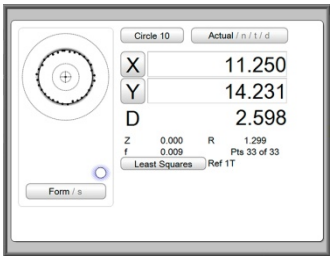
- Feature name
- Position (X, Y and Z)
- Dimension: diameter and radius for this circle
- Form
- Number of points in the data cloud
- Access to tolerance screens
- Data cloud fit
- Graphic of feature data cloud
- Reference frame number

The data cloud fit can be changed when other fits are available by pressing the data cloud fit button (labeled Least Squares for this circle). When the data cloud fit is changed, the graphic of the feature and other feature parameters will change in response.

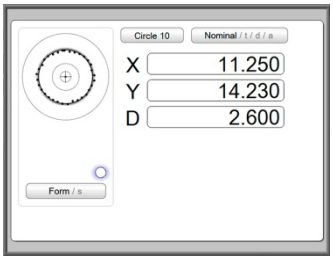
Pressing the Tolerance buttons (Actual, Nominal, Tolerance and Deviation) provides access to the tolerance screens for this feature. Tolerance screens include:

- Nominal
- Tolerance
- Deviation

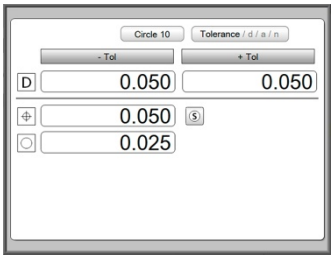
Press the Tolerance button repeatedly to cycle through all of the tolerance screens.



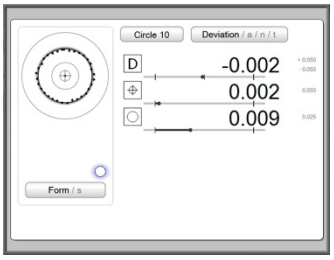
Pressing Actual displays the Nominal screen



Pressing Nominal displays the Tolerance screen



Pressing Tolerance displays the Deviation screen



Pressing Deviation displays the Actual screen

NOTE

The tolerance screens are discussed in detail in Section 8: Tolerancing

Part View screen

The Part View screen is a graphical display of all measured, constructed or created features for the current measurement session. The Part View can also contain feature annotations, a reference frame indicator, the run number and grid lines. Features can be selected for constructions using the same method as the Feature List. Feature selection from the Part View also provides a means of dimensioning features. Tools are provided for zooming and panning in the Part View. For touch screen enabled systems, pinch zooming and two finger panning are also supported.

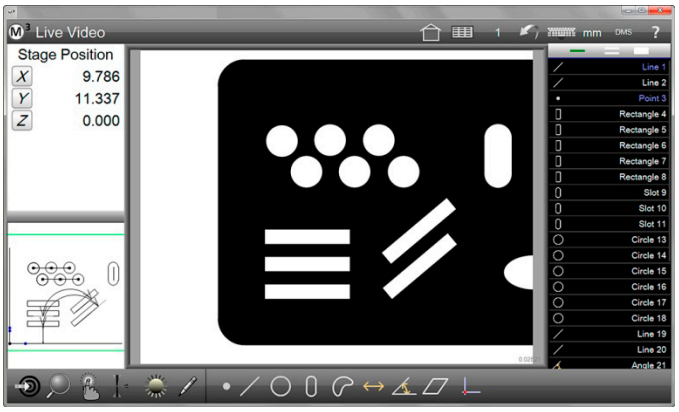
Part View functions include:

- Selecting features
- Panning and zooming features
- Feature annotation
- Box markup
- Text markup
- 3D views

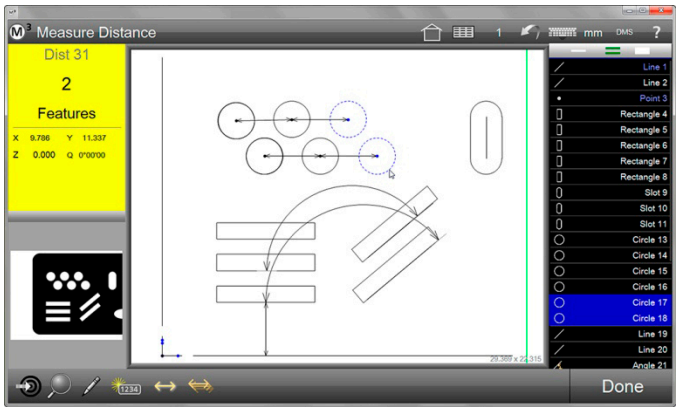
Accessing the Part View screen

To access the Part View screen:

1 Press the small part view in the lower left viewport. The Part View screen will be presented in the center viewport.



Press the small part view in the lower left viewport



The Part View screen will be presented in the center viewport

Selecting Features in the Part View screen

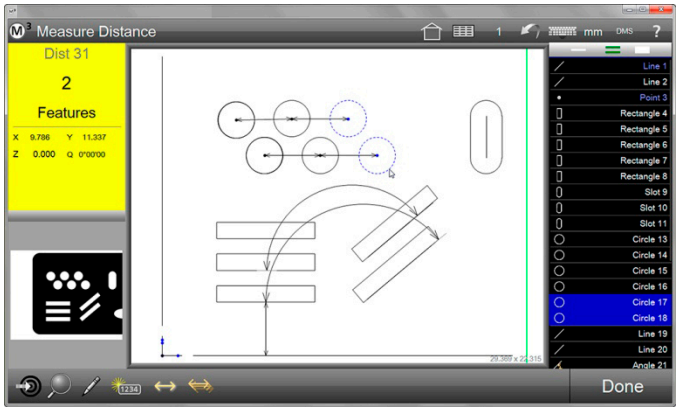
Features are selected in the Part View screen to perform operations including:

- Constructions
- Feature deletions
- Markups of feature parameters

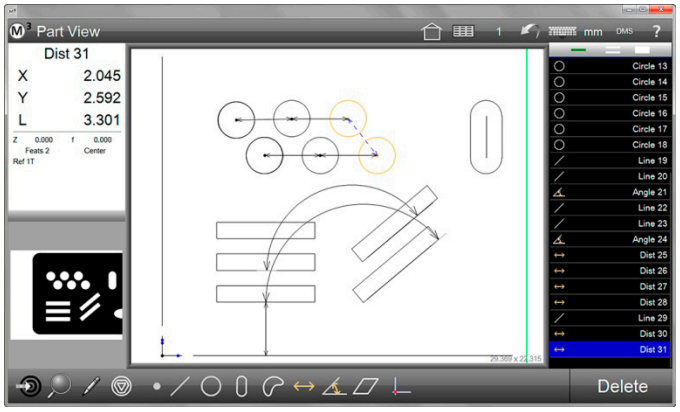
To select features and Part View screen:

1 Press or click the features in the Part View screen. Selected features will appear as blue dashed lines. The features are selected as if they had been selected in the Feature List and will be highlighted in the Feature List.

In this example, two circles were selected in order to create a distance construction between them. When a construction is completed, the constructed feature is shown as a blue dashed line and the parent features are shown as solid orange lines.



Press or click features to select them



A construction between two circles is shown

Zooming and Panning

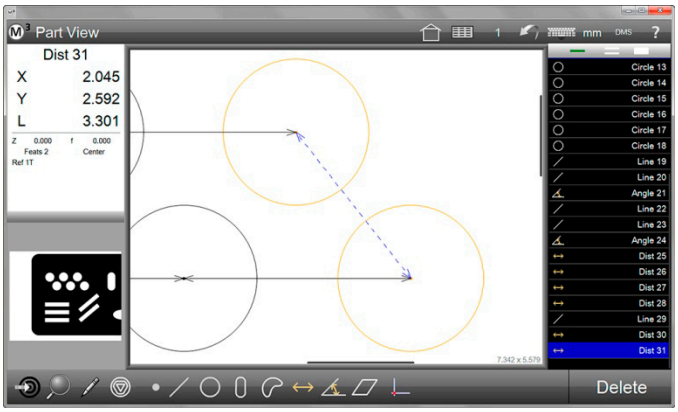
As features are measured, constructed and created, they will populate the Part View screen. Depending on the zoom level, features may extend beyond the viewable area of the Part View. Features that extend beyond the Part View can be brought into viewable range by using one of the several available zoom commands in the Zoom menu found in the Measure toolbar. Pinch-zooming can also be performed in systems with multitouch support. Panning allows the user to move the Part View in any direction using fingertips or the mouse cursor.

To zoom in or out:

1. Select the desired Zoom function from the zoom menu.
   Measure toolbar > Zoom button > Zoom function
2. Or alternatively in multitouch systems, press two fingers to the screen and expand or pinch to zoom out or in.

To pan the Part View:

1. Right-click the mouse cursor in the part view and drag the part view image in any direction.
2. Or alternatively in multitouch systems, press the screen and drag the image in any direction.



In this example, the Part View was zoomed and panned to show the two circles and constructed distance shown earlier in greater detail.

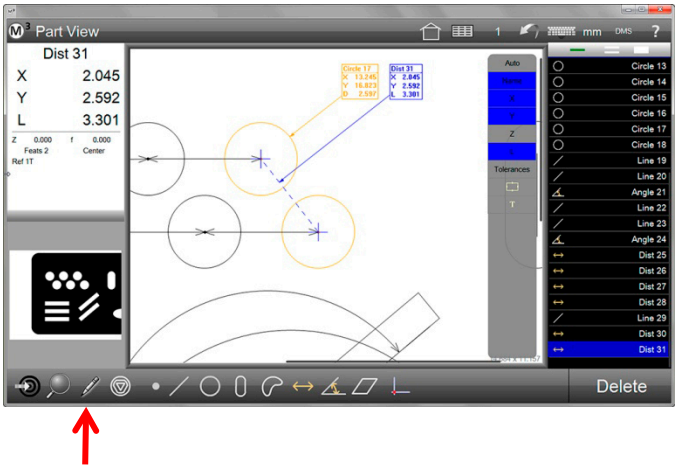
Adding Feature Annotations

Feature annotations can be used to add valuable information to the part view. To annotate a feature in the Part View:

1. Select the feature that you wish to annotate.
2. Press the Markup button to display the Markup menu in the Part View.
   Measure toolbar > Markup button

   A menu of items will appear on the right hand side of the Part View. Select the markup items to apply to the selected feature. The Markup menu changes to support the selected feature type.
   - Auto will draw a single representative coefficient for the selected feature
   - Name will include the feature name in the annotation box
   - Tolerances will include nominal and deviation values for the feature
3. Press the selected feature where you would like to place the annotation box and drag the box away from the feature. An annotation box will appear with an arrow line attached to the feature. Move the annotation box to the desired location and release your finger or mouse cursor.

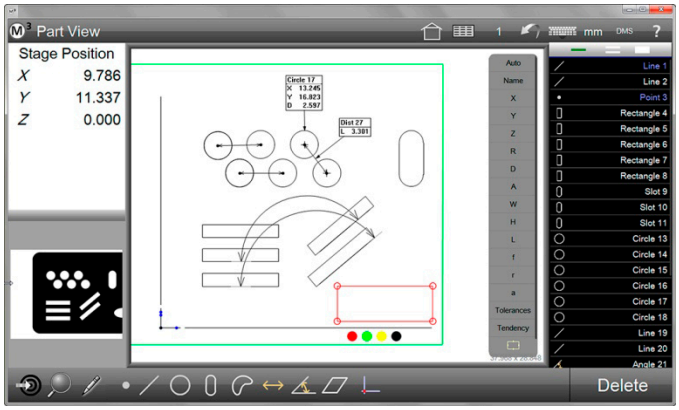
In this example, annotations were made to one circle and one distance.



Adding Box Markups

Boxes can be added to the Part View in one of four colors. To add a box to the Part View:

1. Press the Markup button (pencil) to display the Markup menu in the Part View.
   Measure toolbar > Markup button
   A menu of items will appear on the right hand side of the Part View.
2. Select the Box Markup button from the bottom of the menu.
3. Press or click on the screen and drag to create a box of the desired shape. Release your finger or the mouse cursor to complete the box graphic.
4. Once the box annotation is complete, it can be resized using the circular handles at the corners, or can be moved by pressing or clicking in the box and dragging it to the desired location.
5. To change the color of the box, select the box and then choose the desired color from the color selector dots displayed below the box.

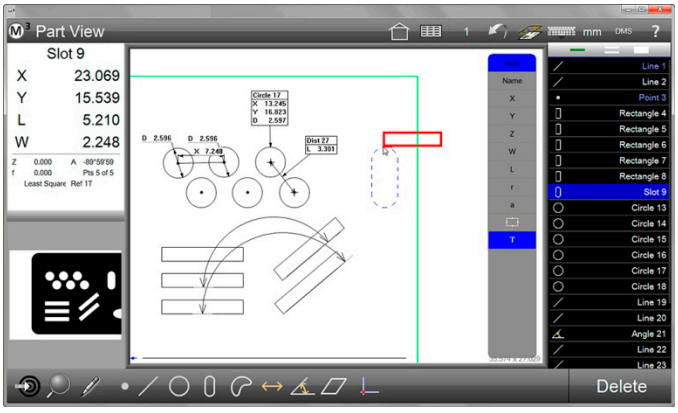


In this example, a red box was added to the bottom right of the part view.

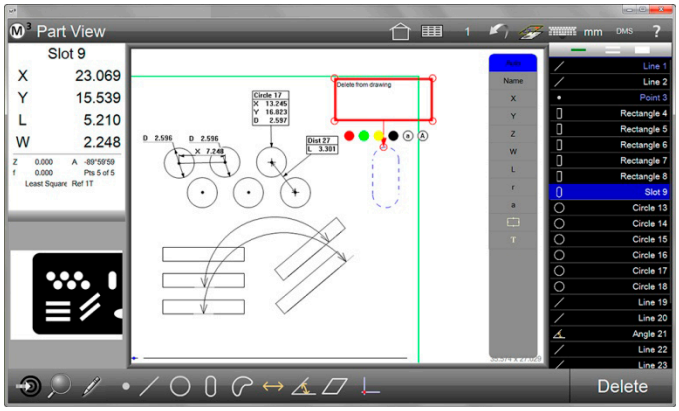
Adding Text Markups

Text can be added to the Part View to convey important information regarding measurements and other observations. To add a text markup:

1. Select the feature that you wish to attach the text markup to.
2. Press the Markup button (pencil) to display the Markup menu in the Part View.
   Measure toolbar > Markup button
   A menu of items will appear on the right hand side of the Part View. Select the coefficients or tolerances to apply to the selected feature.
3. Select the Text Markup (T) button at the bottom of the menu and press or click a point on the feature where you would like to attach the text markup.
4. Drag the text markup box away from the feature. The text markup box will appear with an arrow line attached to the feature. Move the box to the desired location and release your finger or mouse cursor.
5. To change the color of the box, select the box and then choose the desired color from the color selector dots displayed below the box.
6. Enter text into the box using the keyboard.

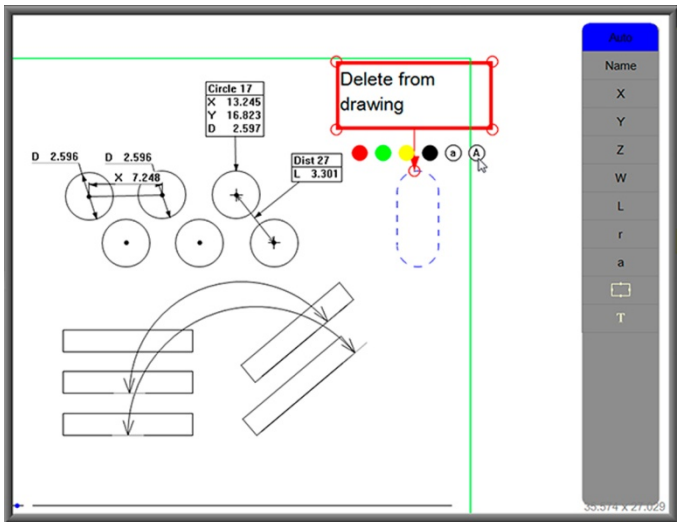


Text markup box attached to the feature



Markup box resized and text entered

7 Resize the text if necessary by pressing or clicking on the lowercase or uppercase character under the markup box.



Markup text resized

Enhanced Data Cloud Display

-Data clouds from measured features can now be displayed directly on the feature graphic in the main part view, and as an overlay on the live video image.

-Data cloud inspection tools such as point display scaling, and "best/worst" point finder tool can help quickly analyze VED measurement results.

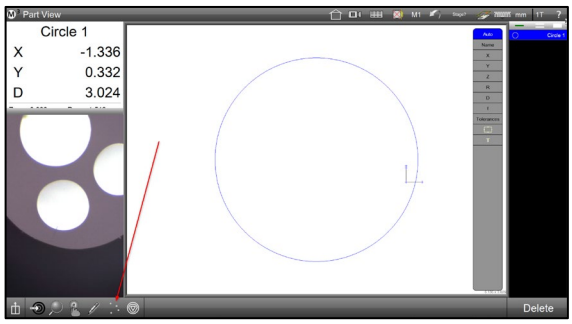
-Enhance feature printouts and reports utilizing this new graphical detail.

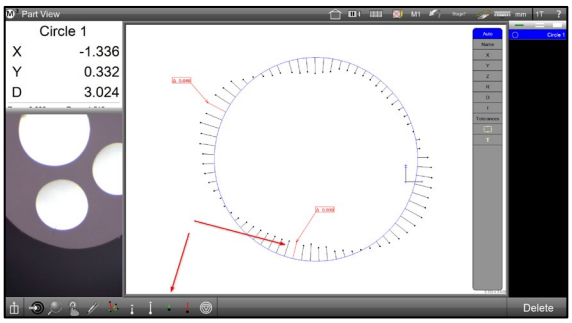
Setup:

NOTE: There are no additional software setup steps required to use the new Enhanced Data Cloud display.

Operation:

The new Enhanced Data Cloud tools are accessed via the new "Show Data Points" button, available in the Annotation menu. First, press the Annotation button(pencil icon) to expand the annotation toolbar, then press the "Show Data Points" button to add the data points for the selected feature to the part view. Adding data points to a feature also exposes the additional data cloud tool bar buttons, seen in the second screenshot below.





The following identifies and describes the functionality of the data cloud tool bar buttons.

(1) Annotation Menu Button – Opens and closes the Annotation Menu

(2) Show/Hide Data Points Button – Toggles the selected feature(s) datapoints between shown and hidden.

(3) Decrease Whisker Mag Button – Decreases the magnification(scaling) of all datapoints in the selected feature. Long-press the button to display a numeric entry dialog. Enter the numeric magnification you would like to use. Press Done to set the specified magnification.

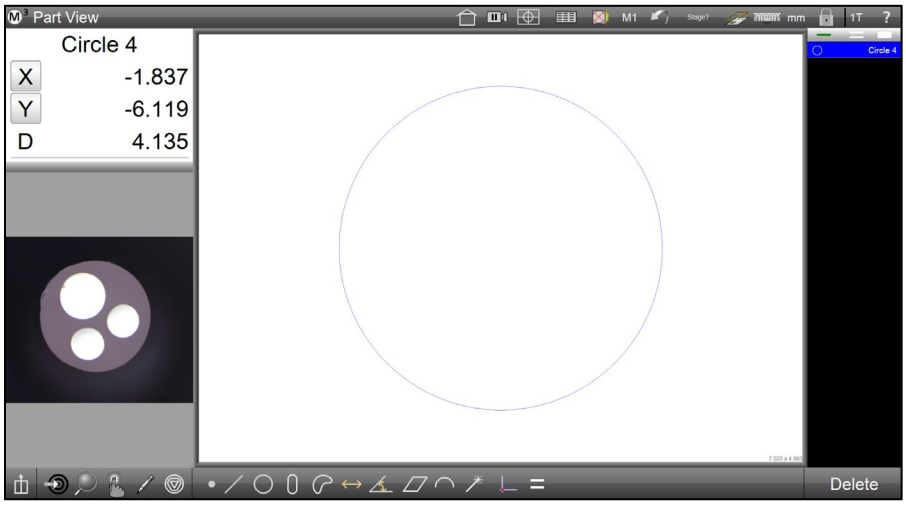
(4) Increase Whisker Mag Button – Increases the magnification(scaling) of all datapoints in the selected feature. Long-press the button to display a numeric entry dialog. Enter the numeric magnification you would like to use. Press Done to set the specified magnification.

(5) Better/Best Whisker Button – Steps through the selected features data cloud, point by point, in the direction of smaller/less form error. Long-press the button to automatically select the data point with the least form error.

(6) Worse/Worst Whisker Button – Steps through the selected features data cloud, point by point, in the direction of greater/more form error. Long-press the button to automatically select the data point with the greatest form error.

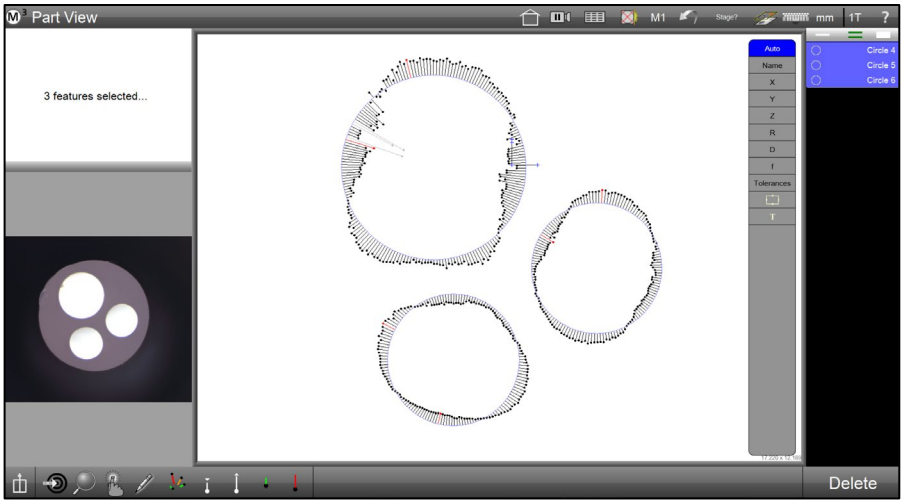
Sample Procedure Steps for adding "data cloud markup" to the part view:

1. Measure a feature, preferably with a moderate to high point density, to use for demonstration in the following procedure.
2. Press in the small part view to bring it to the large center viewport.

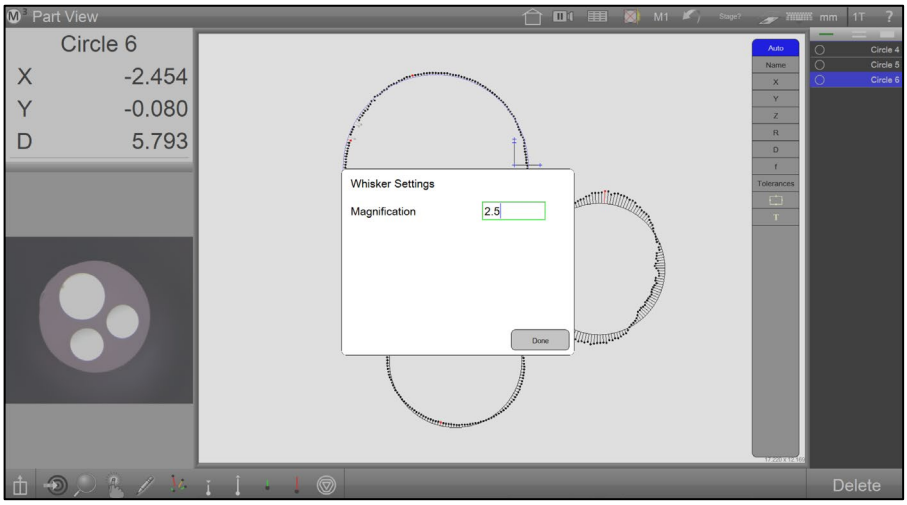


3. Press the Annotation Menu button to open the annotation toolbar.
4. Select the features from the feature list that you would like to enable data point display for.

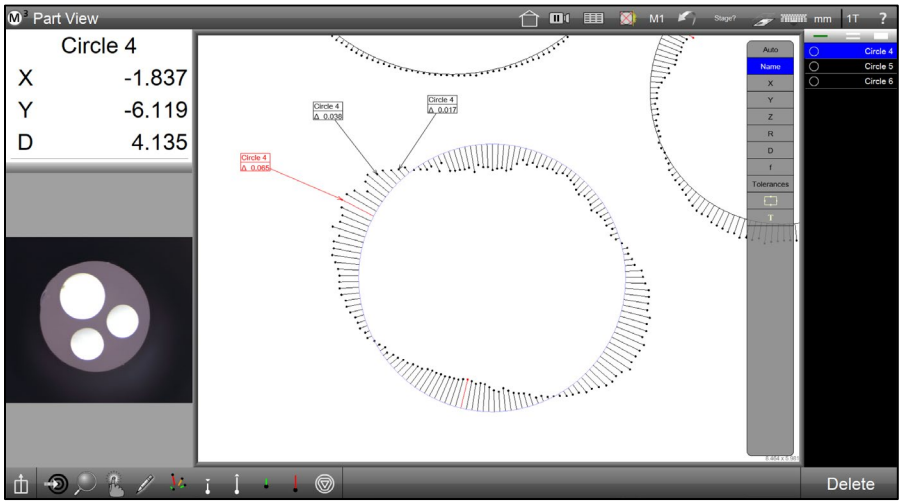
5. Press the Show/Hide Data Points button to display data points and whiskers for the selected features.



6. Press the Increase and Decrease Whisker Mag buttons to set the desired scaling for the display of data point whiskers. Features with very low total form error may benefit from increased magnification in order to more easily observe variation in form error graphically. The inverse is true for features of very large total form error.
7. Long-press on either the Increase or Decrease Whisker Mag button to display the numeric entry dialog for manually setting whisker magnification. Magnification can be set to individual features, or to a selected group of features.

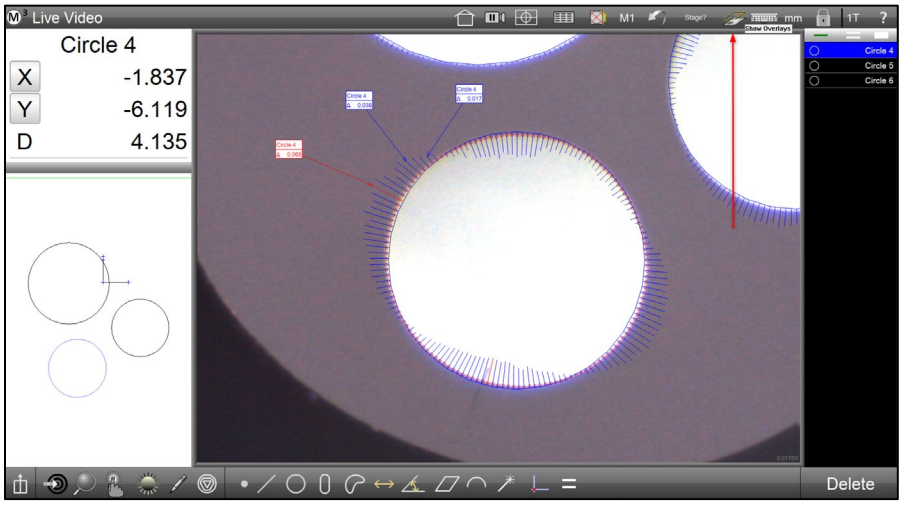


8. Select a single feature in the feature list, then a single data point in its data cloud. An annotation label displaying the datapoints "delta from fit" will be added for that data point. Press delete to remove the annotation label or select additional data points to add more labels. Press annotation items from the list at the right to add and remove data from the label.



NOTE: The data point with the greatest delta from fit, per side, will be displayed in red. Added annotation labels will also be displayed in red.

9. Long press the form error buttons to automatically select the whisker with the least or most form error.
10. Press in the live video viewport to bring the live video into the large center viewport.
11. Press the Part View Overlay button in the top right toolbar to display the data point markup on top of the live video image.



12. Data cloud markup can be printed along with the part view or live video by pressing the print button from the M3 menu.

3D Viewing

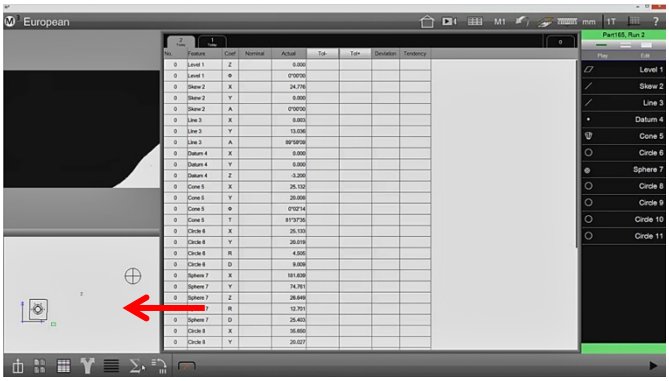
Systems that include the touch probe option also include 3D viewing functions. 3D part drawings are displayed in the Part View window and include all the viewing tools and functions of 2D drawings, such as Zooming, Panning and Annotation. Additional 3D functions include:

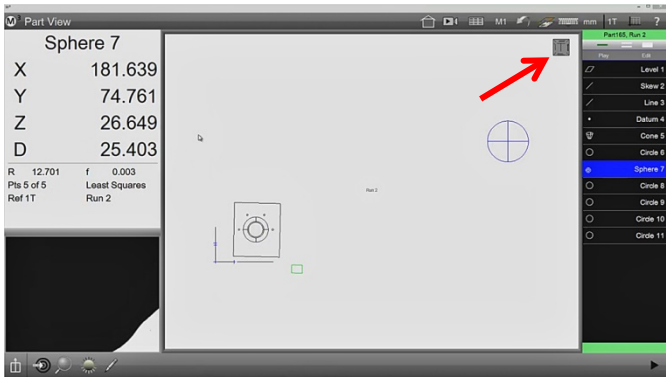
- Displaying 3D part views
- Changing 3D horizontal and vertical rotation
- Displaying preset 3D views

Displaying 3D Part Views

To display the 3D part view:

1 Press the small Part View window. The Part View will be displayed in the center window as a 2D image in the top view.





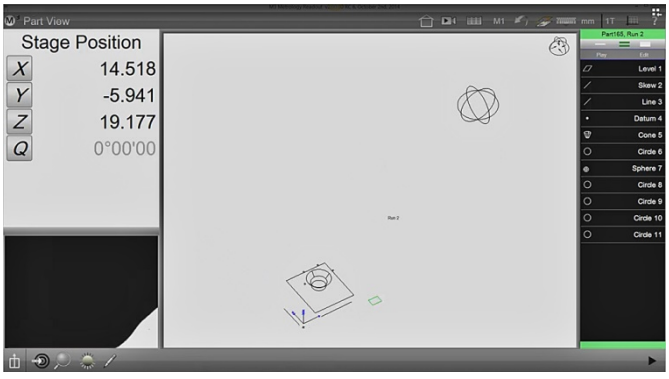
2 Long-press and hold the control key to enable 3D viewing.

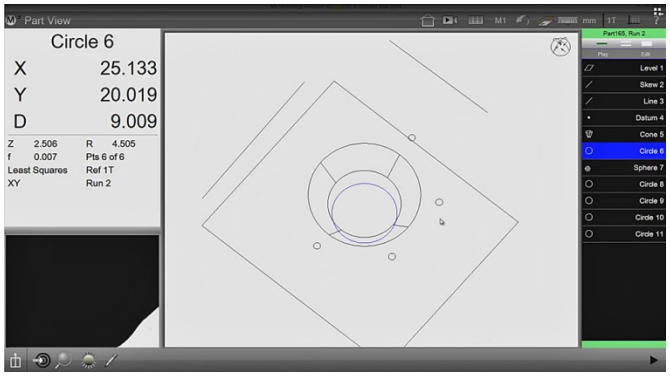
OR

2 In touch probe enabled systems, long-press and release the View Cube at the top-right of the center window to enable 3D viewing.

In either case, the View Cube will change to show 3D drawing orientation and the part view will change to 3D.

3 Use the Pan and Zoom functions to optimize the display of features.





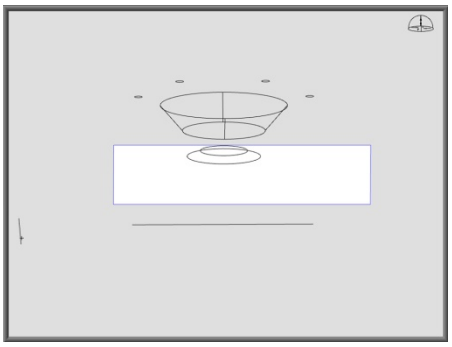
Rotation and elevation view controls

When 3D viewing is enabled by control key or View Cube, rotation and elevation of the view are controlled by pressing and dragging on the screen.

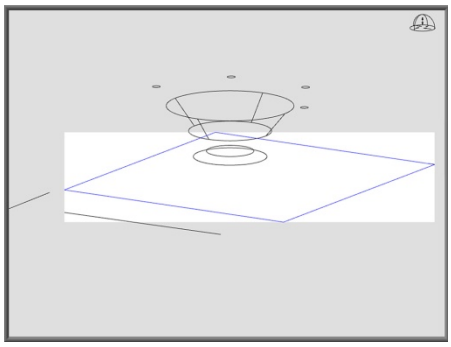
Changing the Horizontal Rotation

To change the horizontal rotation:

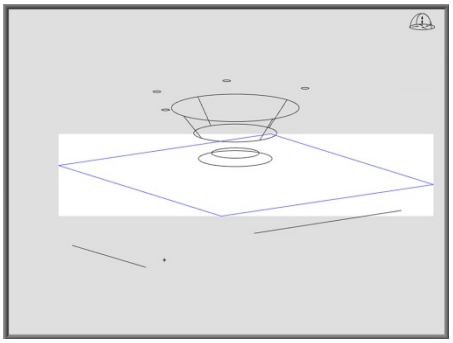
Press and drag horizontally on the screen. Drag to the left to rotate the front to the left.



Not rotated



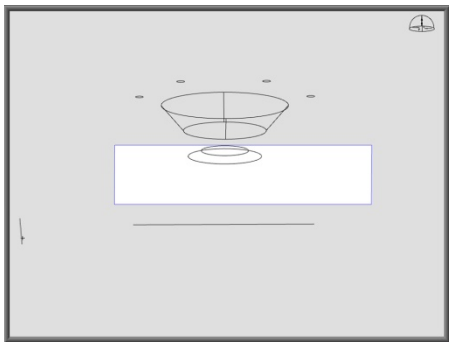
Rotated left



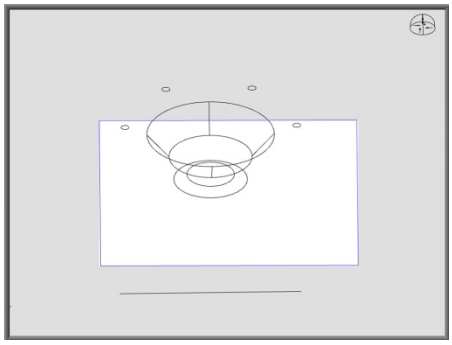
Rotated right

Changing the Vertical Rotation

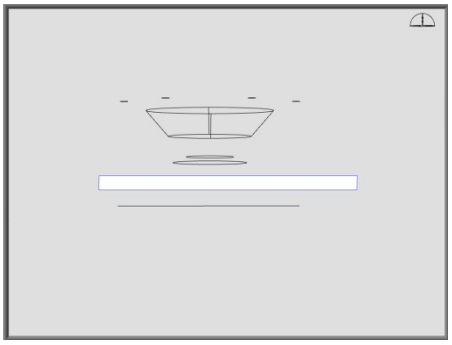
To change the vertical rotation:
Press and drag vertically on the screen. Drag up and down.



Not rotated

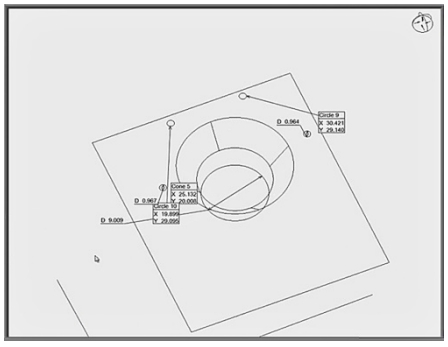


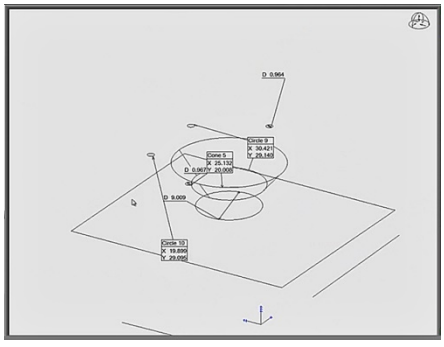
Rotated down



Rotated up

Markups retain their position and readability throughout rotation changes.

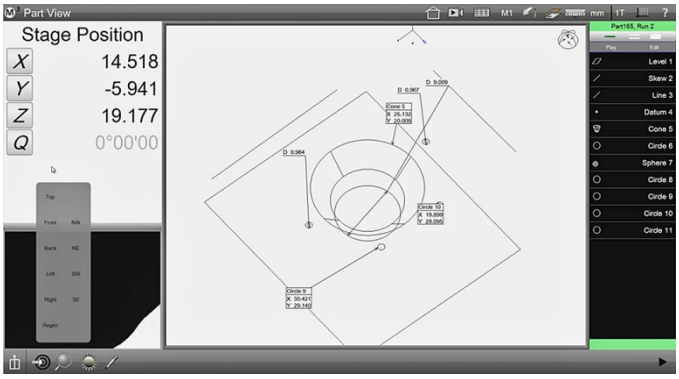


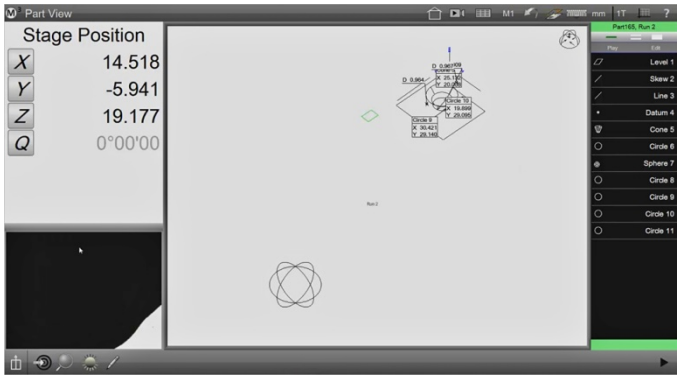


Preset 3D Views

Standard preset viewing orientations can be selected from the 3D Preset menu.
To select a viewing orientation:

1. Long-press the Zoom menu button. The Preset menu will be displayed.
2. Press the desired preset view. This example shows the SW (southwest) view.

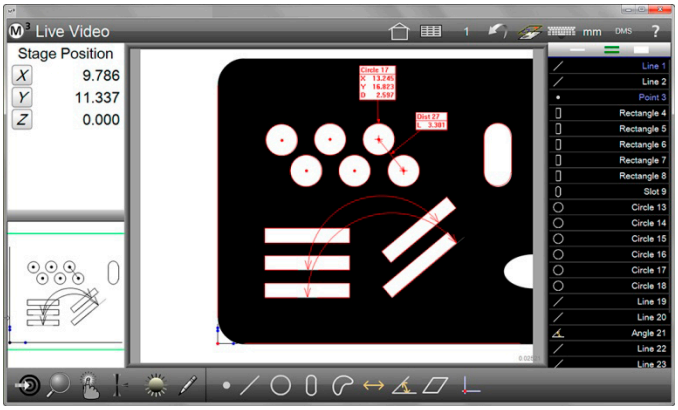




The Part View Overlay

Once the Part View has been marked up with measurement parameters in text, the marked up view can be used as an overlay to add valuable information to the live video window view. To display the Part View overlay:

1. Press the Home button to display the live video feed in the center viewport.
   System toolbar > Home button
2. Press the Overlay button to position the Part View markup over the live video image of the part.
   System toolbar > Overlay button



Section 7: Reference Frames

Establishing a reference frame and datuming is accomplished using any of a variety of alignment operations. These operations may be used individually or in combination, and will be based upon the capability of the measuring machine as well as the particular alignment requirements of the application.

Datuming operations can be performed in the form of explicit datum feature measurements, by modifying the individual coefficient values of previously measured features or by constructing datums from existing features.

In three axis systems, the M3 can perform a primary part alignment by evaluating the surface-level of the part, a secondary x-axis part skew alignment and finally establish a datum point which is the origin of the 2D or 3D coordinate system.

When measuring a datum-dependent part, it is advisable to establish a level, skew and datum alignment first, and then measure the features associated with that datum. If multiple datums or reference frames are required, establish the initial datum and measure the associated features. Then proceed to the next datum and measure the features associated with it. Always try to follow the sequence of establishing a datum and then measuring the features associated with that datum.

The skew, or secondary alignment, can be offset by the user when the skew line position is located where it cannot be measured directly by the system. This offset skew position is identified by values from other measured or constructed features. The skew alignment can also be rotated to specified angular values by the user to facilitate measurements based on angular secondary alignments.

NOTES

Establishing a complete primary, secondary and tertiary reference frame may not always be required, even in three axis systems. Some three axis systems and the associated applications will not require that a level be performed. In these cases a skew alignment and datum zero point are typically adequate for successful part of alignment and measurements.

Most well-maintained systems include stage surfaces that are level with only extremely minor variations. In video systems that have level variations across the surface of the stage that are smaller than the video camera depth of field, performing a video leveling operation is not beneficial.

Establishing a Reference Frame

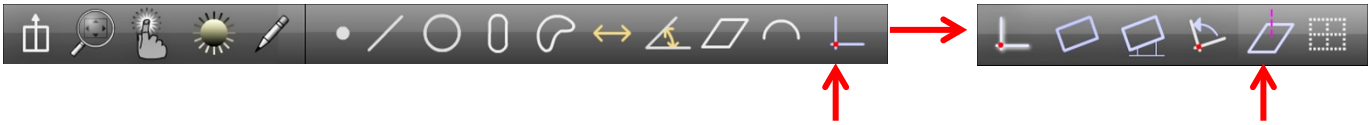
The M3 measurement reference frame can consist of:

- Part leveling (a primary datum plane)
- Skew alignment (secondary alignment)
- Part Datum (measurement coordinate system origin)

Part Leveling

When the surface plane of a part is not orthogonal to the measuring axes, small cosine errors can be generated during the measurement process. These errors can be eliminated by leveling the part.
To measure a primary datum plane:

1 Select the desired probe from the probe menu.
Measure toolbar > Probe menu > Desired probe
2 Press the Datum Plane button of the Datum menu.
Measure toolbar > Datum menu > Datum plane button



3 Probe a minimum of three points evenly distributed across the surface of the part. Probing more than three points increases the accuracy of the measurement.
4 Press the Done button to complete the measurement and enter the datum plane into the Feature List.
Measure toolbar > Done button

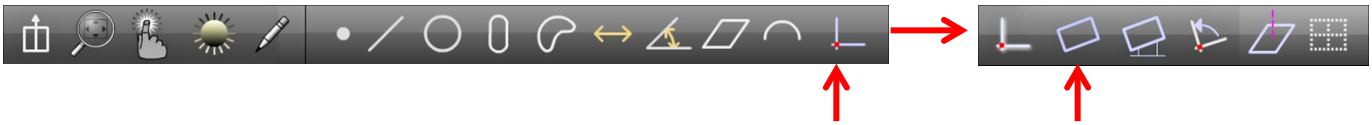
NOTE
Previously measured or constructed plane features can also be leveled by manually zeroing the theta angle coefficient and the Z position coefficient. Pressing the label buttons to the left of these coefficients in either the feature detail screen or the DRO viewports will set the corresponding values to zero.

Skew alignment

When a part is misaligned (twisted) in the XY plane, small cosine errors can be generated during the measurement process. These errors can be eliminated by creating a skew alignment for the part. The skew alignment performs a precise measurement of the part of misalignment in the XY plane. Once the misalignment is known to the system, subsequent feature measurements are automatically compensated to eliminate cosine errors. Measurement data in the feature detail screen and DRO viewports will reflect measurements of a perfectly aligned part.

To create a skew alignment:

1 Select the desired probe from the probe menu.
Measure toolbar > Probe menu > Desired probe
2 Press the Skew button of the Datum menu.
Measure toolbar > Datum menu > Skew button



3 Probe a minimum of two points well distributed along the entire length of the desired part reference edge.
4 Press the Done button to complete the measurement and enter the skew line into the Feature List.
Measure toolbar > Done button

NOTE
Previously measured or constructed features can also be used to perform a skew alignment by manually zeroing the theta angle coefficient and the X or Y axis coefficient. Pressing the label buttons to the left of these coefficients in either the Feature Detail screen or the DRO viewport will set the corresponding values to zero.

Datum Creation

Creating the part datum point (origin) is the final step in establishing a valid reference frame for subsequent measurements. The part datum can be created by:

- Probing a single point
- Constructing a point from one or more parent features
- Manually zeroing the X and Y coefficients of a feature

Probing a Part Datum Point

The part datum can be created directly by probing a point. To probe a part datum point:

1 Select the desired probe from the probe menu.
Measure toolbar > Probe menu > Desired probe
2 Press the Datum button.
Measure toolbar > Datum button
3 Probe the desired point and press the Enter button or the probe prompt window.
Measure toolbar > Enter button
4 Press the Done button to complete the measurement and enter the part datum point into the Feature List.
Measure toolbar > Done button

Constructing a Part Datum Point from one or more Parent Features

The part datum can be constructed from one or more parent features when simply probing the desired point is not convenient or possible. Supported point constructions are specified in Section 4: Measuring, Constructing and Creating Features. To construct a datum point from a single feature:

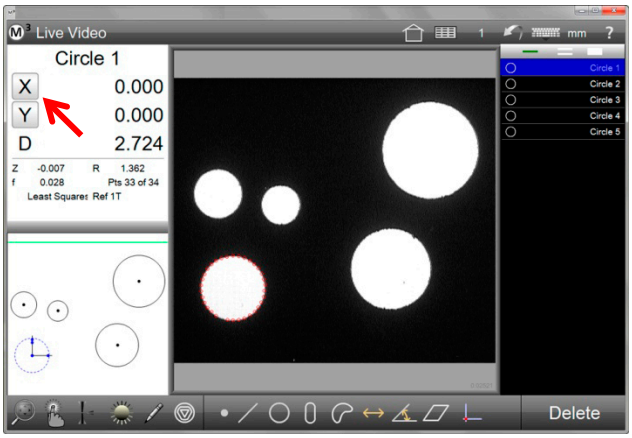
1 Press the Datum button.
Measure toolbar > Datum button
2 Select (highlight) the desired parent feature or features in the Feature List or Part View.
3 Press the Done button to complete the construction and enter the part datum point into the Feature List.
Measure toolbar > Done button

Creating a Datum by Manually Zeroing the X and Y Coefficients of a Feature

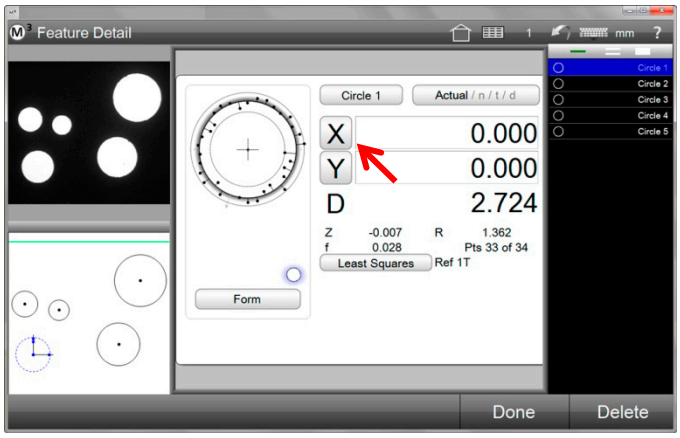
Any of the features contained in the Feature List can be used as a part datum by manually zeroing the X and Y feature coefficients. The X and Y coefficients can be zeroed in the DRO viewport or in the feature detail screen. To zero the X and Y coefficients of a feature:

1 Select (highlight) the desired feature in the Feature List.
2 Press the X coefficient button and then the Y coefficient button in the DRO viewport or in the feature detail screen. The X and Y values will become zero establishing the geometric center point of the feature as the part datum point.

In this example, the geometric center point of circle 1 was set to zero and will serve as the part datum point for subsequent measurements.



X and Y set to zero in the DRO viewport



X and Y set to zero in the Feature Detail screen

Rotating and Offsetting a Reference Frame

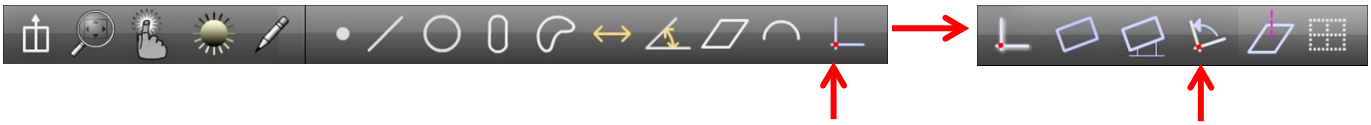
The M3 reference frame can be rotated and offset to accommodate custom measurement applications.

Rotating the Measurement Reference Frame

The measurement reference frame can be rotated about the part datum in clockwise or counterclockwise directions and in angular increments as small as one second or 1/1000 of a degree depending on a unit of measure being used. To rotate the measurement reference frame:

1 Press the Rotate Coordinates button of the Datum menu

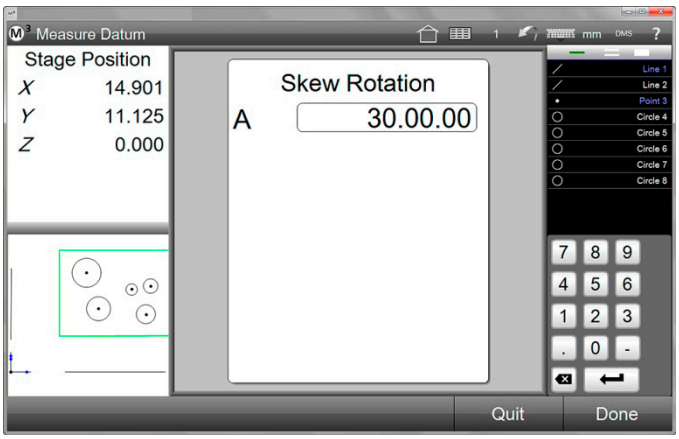
Measure toolbar > Datum menu > Rotate coordinates button



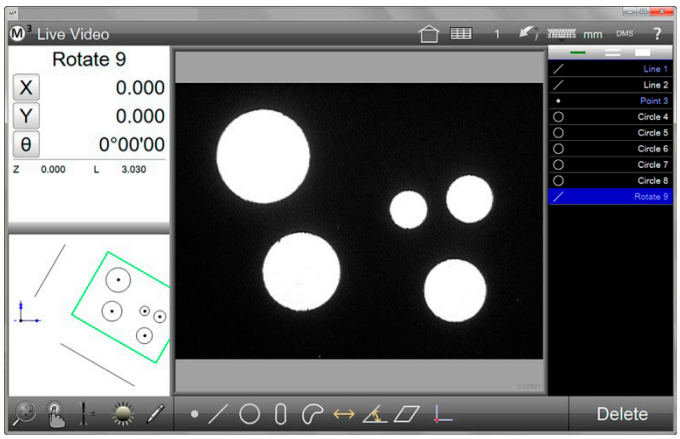
2 Enter the desired angular rotation value into the Angle field. Clockwise angles are positive, and counterclockwise angles are negative.

3 Press the done button to complete the reference frame rotation. A rotation feature will be added to the Feature List.

Measure toolbar > Done button



Enter the desired rotation angle and press Done



Reference frame rotated

To reset the reference frame to the original rotation:

1 Delete the rotation feature from the Feature List.

2 Repeat the process described above but enter a skew rotation equal in value but opposite in polarity to the initial Skew Rotation value.

3

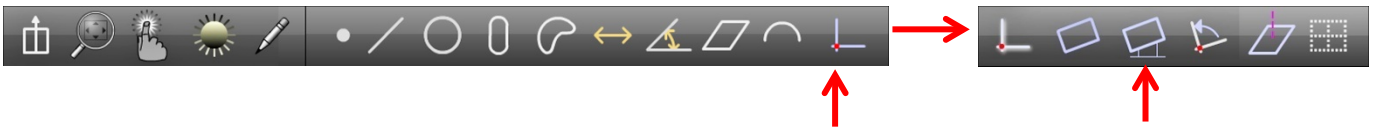
Offsetting the Skew Alignment

The skew alignment can be offset from the X or Y coordinates by increments as small as one micron or 1/10,000 of an inch depending on the unit of measure being used. To offset the part skew alignment:

1 Measure two features that will determine the skew offset. Make a note of the X and Y values of each feature. These features will be referred to as Feature #1 and Feature #2 in this procedure. During the procedure, the user will specify offsets from these two features to the skew axis. The system will then construct a line between the two features which will be used as the new part skew alignment.

2 Press the Offset Skew button of the Datum menu

Measure toolbar > Datum menu > Offset Skew button



3 Select an alignment axis from the Axis drop-down list. In the next step, offsets will be specified extending from the two features previously measured to this axis. In this example the X axis was selected.

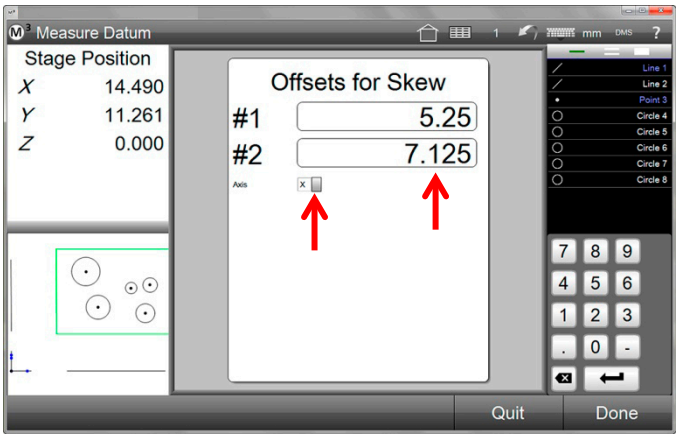
4 Enter the desired skew offsets from Feature #1 and Feature #2 to the selected axis in the same order in which the features were measured. These offsets are the distances from the geometric centers of the features to the selected axis.

5 Press the Done button to complete the skew offset value entry. The system will proceed to the measure mode.

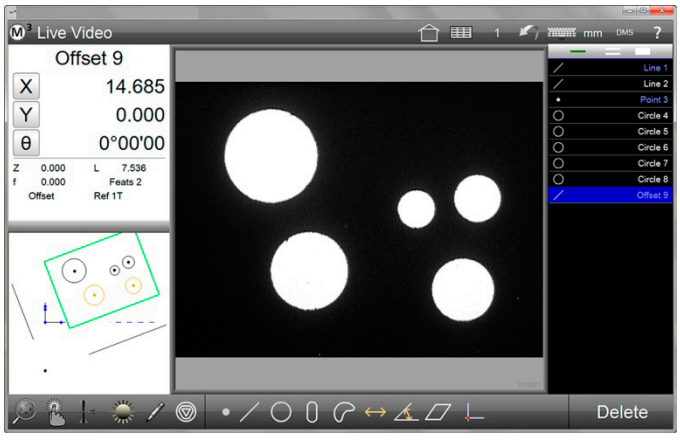
Measure toolbar > Done button

6 Select Feature #1 and Feature #2 in the Feature List in the same order in which they were measured and then press the Done button. The offsets entered in the previous step will be applied to the features and a new offset part skew will be created by the system. An offset feature will also be added to the Feature List.

Measure toolbar > Done button



Select a skew axis and enter offsets



The skew alignment will be offset

To reset the part skew to its original orientation:

1 Delete the offset feature from the Feature List.

2 Repeat the process described above but enter the original X or Y value offset of Feature #1 and Feature #2 noted from step 1.

Managing Multiple Reference Frames

The M3 system supports multiple reference frames as follows:

- By default, while several reference frames can be created during a measurement session, the reference frames are not automatically saved and all measurements will be referred to the last reference frame created (the current reference frame).
- When multiple reference frames are created and then saved, either manually or automatically, measurements will be referred to their own reference frames.
- Reference frames are identified in the small Feature Detail viewport, in the large Feature Detail screen that appears in the center viewport and in the System toolbar or Extra menu when the Reference Frame button is enabled in the Desktop Settings screen.
- Saved reference frames and their related measurements are displayed with color coded bars in the Feature List and in the System toolbar.
- Measurements can be temporarily unlocked from saved reference frames by the user. Unlocked measurements are no longer referred to their own reference frames and will be referred to the current reference frame. Measurements can be relocked to their saved reference frames later by the user.
- All reference frame operations will become part of recorded programs and can be played back later. However, no new reference frame creation is permitted in the program edit mode.

Unsaved Reference Frames

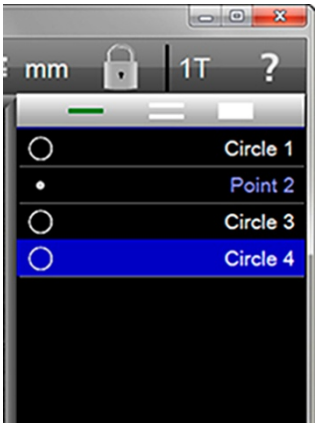
As mentioned earlier, by default, reference frames are not saved automatically and while several reference frames might have been created, all measurements will be referred to the last reference frame created (the current reference frame). Reference frames created earlier cannot be saved later, only the last reference frame can be saved.

Saving Reference Frames

Reference frames can be saved manually by the user or automatically by the system. In either case, measurement results are associated with their own reference frames. However, saving a reference frame manually requires a user operation and relies on the user to remember to do it. Saving reference frames automatically eliminates these issues.

To save a reference frame manually:

1. Confirm that the Reference Frame button is shown in the Extra menu or in the System toolbar. If it is not, configure the button in the Desktop Settings screen.
   System toolbar > System menu > Desktop Settings screen > Reference Frame button > Desktop or Extra
2. After the creation of one reference frame, and before the creation of another, long-press the Reference Frame button. The reference frame indication will change from temporary (1T) to permanent (1), indicating that the reference frame was saved and a color coded bar will appear in front of the datum and associated feature measurements.



Reference frame button is 1T

Reference frame button is changed to 1

To save a reference frame automatically:

1. Set Automatically Save UCS (Universal Coordinate System) to yes in the Measure Settings screen.
   System toolbar > System menu > Measure Settings screen > Automatically Save UCS > Yes

When the reference frame is saved automatically, the observed behavior of the system is the same as shown above except that no user intervention is required to save any reference frame.

Locking and Unlocking Features from their Reference Frames

Feature measurements can be locked or unlocked from saved reference frames by the user. Unlocked measurements are no longer referred to their own reference frames and will be referred to the current reference frame. To lock or unlock feature measurements:

1 Confirm that the Feature Lock button is shown in the Extra menu or in the System toolbar. If is not, configure the button in the Measure Settings screen.

System toolbar > System menu > Measure Settings screen > Feature Lock button > Desktop or Extra

2 The default condition of the Feature Lock button is locked. Press the Feature Lock button to toggle between locked and unlocked.

Section 8: Tolerancing

Tolerances are applied to the feature measurements in the Feature Detail screen. While some tolerances are independent of a reference frame, typically tolerances are applied after a valid reference frame has been established. Tolerances can be applied to measured and constructed features. Each feature type has a specific set of tolerances that can be applied. Tolerances can be applied to features interactively or as part of program editing and are included in part programs for playback later. Tolerance status is indicated by color in the Feature List, Part View, Feature Detail screen and in the Report screen. Tolerance failures are displayed in red.

Tolerances can be applied to single features or to a group of like features. Nominal values for tolerances can be entered by the user or imported from DXF drawing files. Tolerances can be applied on a feature to feature basis, or by using the place tolerancing system. Engineering drawings might indicate that all features reported to two significant digits will receive one tolerance value while other features reported to three significant digits will receive another. The M3 software supports the application of universal tolerance values according to these tolerance groupings.

Supported Tolerances

Tolerances supported by the M3 system are shown in the table below.

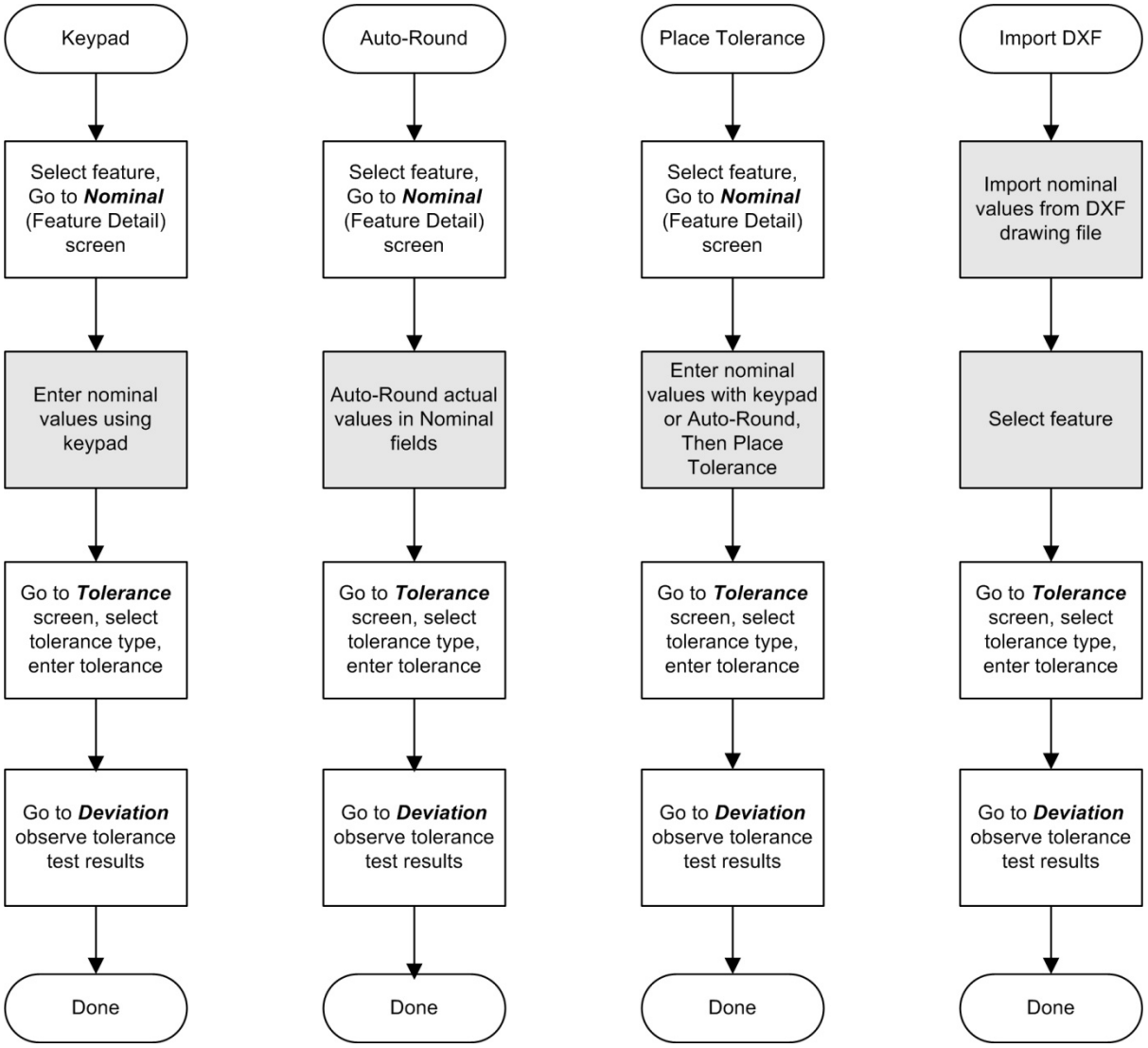
Angle (theta)	∠	Angle, Line, Pattern
Angularity	∠	Angle
Area	Ar	Blob
Circumference	C	Blob
Concentricity	◎	Circle, Cylinder
Cylindricity	⌭	Cylinder
Diameter	D	Arc, Circle
Flatness	▱	Plane
Form	*f*	Rectangle, Slot, Cone
Length	L	Blob, Distance, Rectangle, Slot
Parallelism	//	Line, Plane
Perpendicularity	⊥	Line
Radius	R	Arc
Roundness	○	Circle
Runout	↗	Circle
Sphericity	○	Sphere
Straightness	—	Line
Symmetry	≡	Line
Taper	T	Cone
True position	⌖	Arc, Circle, Point, Rectangle, Slot, Sphere, Cylinder, Cone
Width	W	Blob, Rectangle, Slot
X position	X	Plane
Y position	Y	Plane
Z position	Z	Plane
XY position	X Y	Arc, Blob, Circle, Line Rectangle, Slot, Cylinder, Cone, Pattern
XZ position	X Z	Arc, Circle, Line Rectangle, Slot, Cylinder, Cone
YZ position	Y Z	Arc, Circle, Line Rectangle, Slot, Cylinder, Cone
XYZ position	X Y Z	Distance, Point Sphere

Applying Tolerances to a Single Feature

The process of applying tolerances to features is nearly identical for all feature types, only the tolerance type and the entry of nominal values are different. The four methods of entering nominal values and applying tolerances include:

- Enter nominal values using keypad
- Auto round actual values
- Enter values with keypad or auto round and then use place tolerancing
- Import nominal values from a DXF drawing file

The four methods are diagrammed here. The differences between these methods are shown in shaded blocks. The contents of each block will be explained in detail. The blocks that are common to all processes will be explained as part of an example of tolerancing a circle. The blocks that are unique in each process will be explained separately.



NOTE

Tolerances can also be applied using the Quick Tolerance functions discussed at the end of this section.

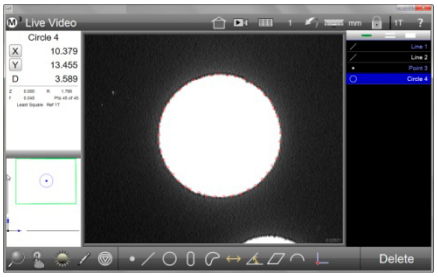
Keypad Entry of Nominal Values

In all cases the first step of the tolerancing process is navigating to the Nominal Feature Detail screen and will only be shown once. This example shows nominal values entered using the keypad and tolerances being applied to a circle.

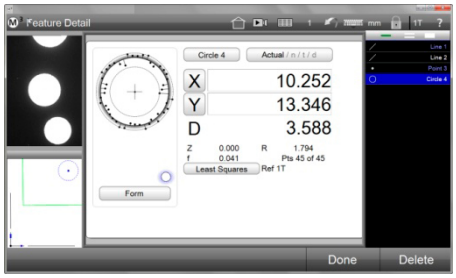
Select Feature and go to the Nominal Feature Detail screen

To select a feature and go to the Nominal screen:

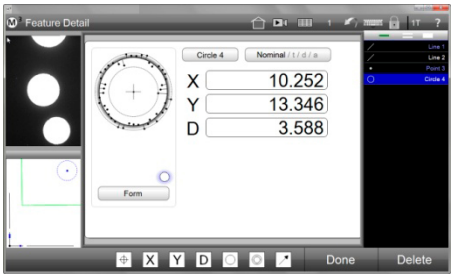
1. Select the desired feature from the Feature List and press in the small Feature Detail viewport to display the Feature Detail screen in the center viewport. Actual feature measurement values will be shown.
2. Press the Actual button to go to the Nominal screen.



Select a feature and press in the small Feature Detail viewport



Press the Actual button

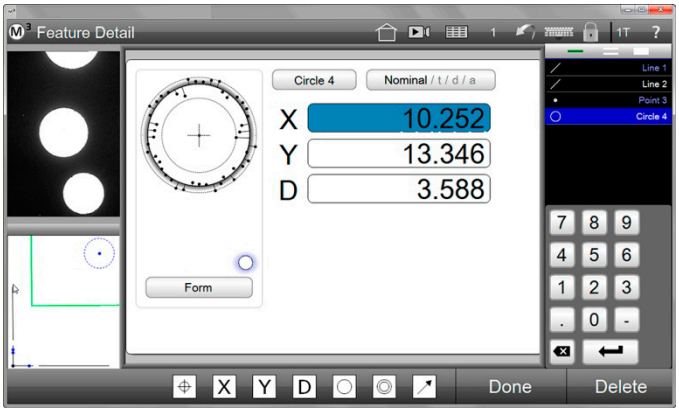


The Nominal screen is displayed

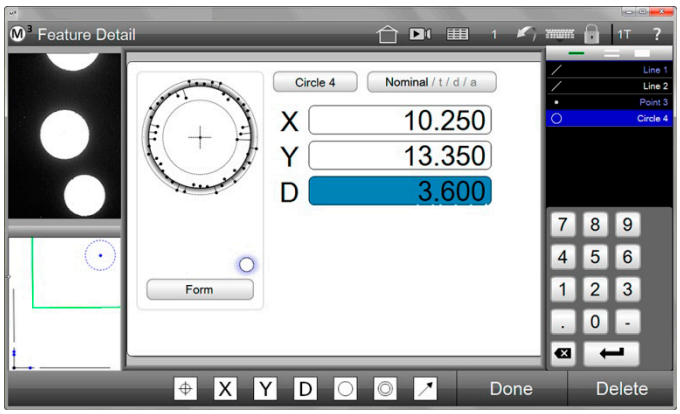
Enter Nominal Values using the Keypad

To enter nominal values:

1. Press in a nominal value field. The field will become blue and the keypad will appear. Enter the nominal values for each field.



Press in a nominal field



Enter nominal values using the keypad

Go to the Tolerance Screen, Select and Enter Tolerance

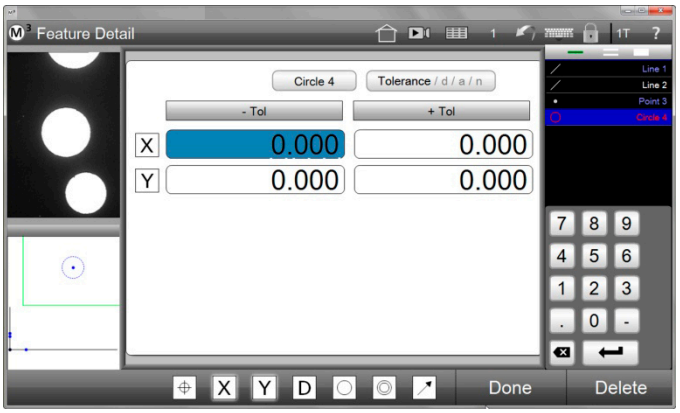
To select and enter tolerances:

1 Press the Nominal button of the Nominal Feature Detail screen to display the Tolerance screen.

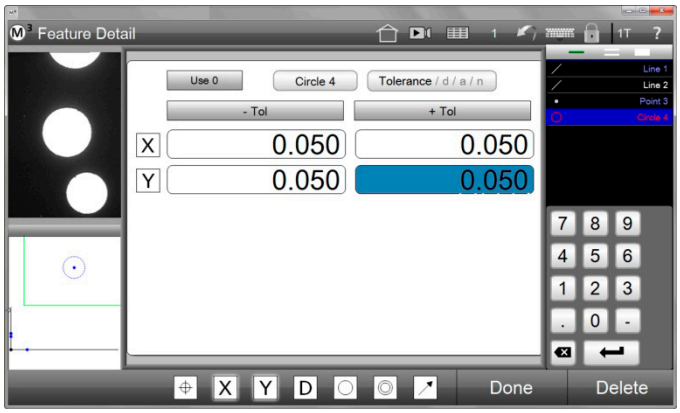
2 Select the desired tolerance type.

Measure toolbar > Desired tolerance button

3 Press in a tolerance field. The field will become blue and the keypad will appear. Enter the desired tolerance values into the tolerance fields.



Select a tolerance type and press a data field



Enter tolerance values using the keypad

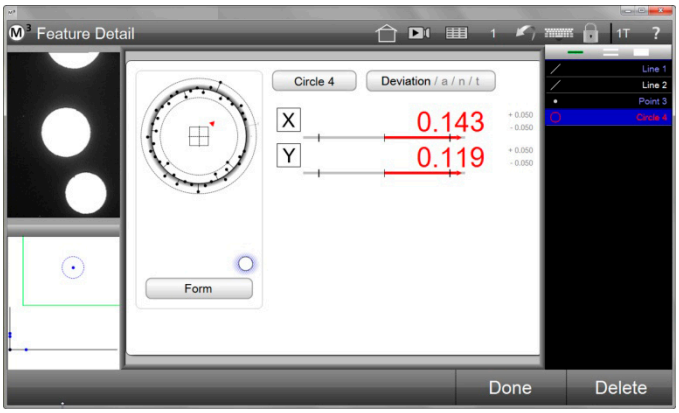
Go to the Deviation Screen to Observe Results

To observe the results of tolerance tests:

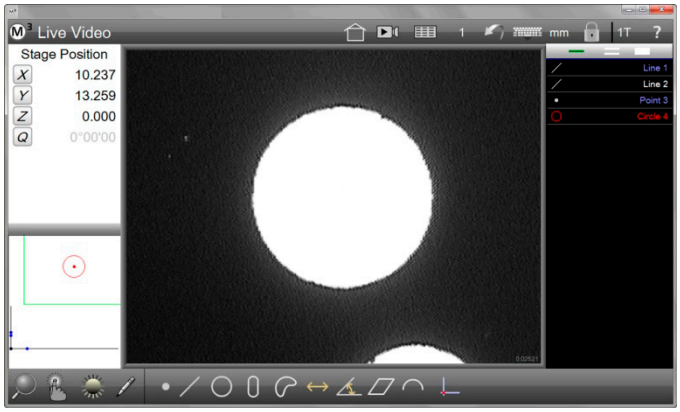
1 Press the Tolerance button of the Tolerance Feature Detail screen to display the Deviation screen. Tolerance results and the applied tolerances will be displayed numerically and graphically.

2 Press the Done button to return to the Home view.

Measure toolbar > Done button



Tolerance results are shown on the Deviation screen



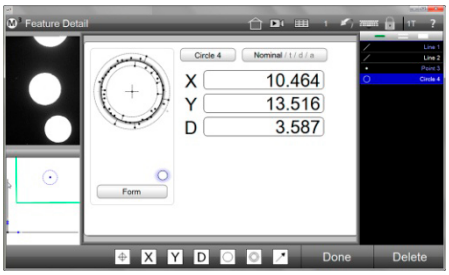
Failed tolerances are shown in red

Failed tolerances will be shown in red in the Feature List, Part View, Part Detail screen and on the Report screen.

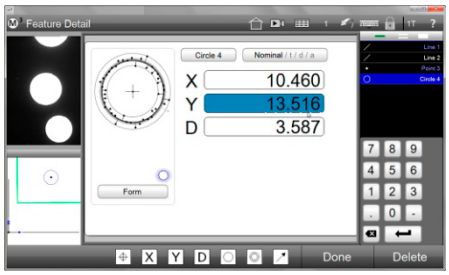
Entering Nominal Values using Auto-Round

Auto-rounding can often save time when repetitively entering nominal feature data. Often measured values are close to blueprint nominal values. Using Auto-rounding, the nominal values for each coefficient of a feature can quickly be rounded up or down to match the nominal values on engineering drawings. To enter nominal values using Auto-rounding:

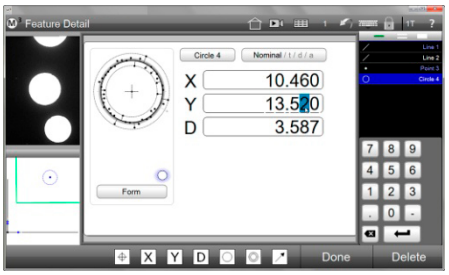
1 Once a feature has been selected in the Feature List, navigate to the Nominal Feature Detail screen as described earlier.
2 Press in a nominal field. The field will become blue and the keypad will appear.
3 Press the digit that you would like to round. In this example, the second digit to the right of the decimal point was pressed which rounded the last two digits from 16 to 20. The digit selected will cause rounding either up or down based on the value of the digit to its immediate right.



Navigate to the Nominal screen



Press in a nominal value field



Press and hold the digit to round

4 To make adjustments up or down to any rounded digit, press and hold it down and then slide your finger or mouse pointer downward off the digit. Do not release your finger or the mouse cursor. This will select the digit place to be incremented or decremented.
5 Slide your finger or mouse pointer to the left to increase the value or to the right to decrease the value, and then release your finger or mouse cursor.

Entering Nominal and Tolerance Values using Place Tolerancing

The place tolerancing method uses tolerance information found in the detail block of an engineering drawing. The detail block specifies tolerance values to be used for drawing features based on the number of decimal places reported for the feature. This example shows how a detailed block might appear on a drawing.

.X	+/-	0.1"
.XX	+/-	0.01"
.XXX	+/-	0.001"

The example indicates that if a feature coefficient includes two decimal places, a tolerance of ±0.01 should be applied.

The M3 system provides a place table for the user to include decimal place information from drawing detail blocks. This table is used in the place tolerancing method to apply detail block tolerances automatically to features measured by the M3 system.

To perform place tolerancing:

1 Confirm that the Enable Tolerance Places field in the Measure Settings screen is set to Yes.
System toolbar > System menu > Measure settings screen > Place Tolerancing > Yes

NOTE

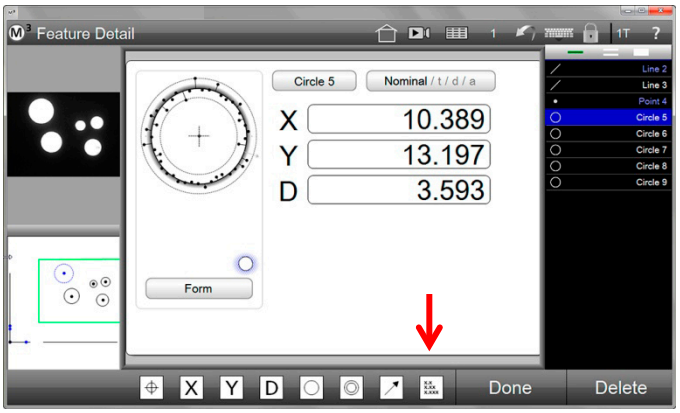
If you wish to apply the same decimal place information to all new M3 measurement sessions, confirm that the Tolerance Places Data on New Part is set to Keep in the Measure settings screen.

System toolbar > System menu > Measure settings screen > Tolerance Places Data on New > Keep

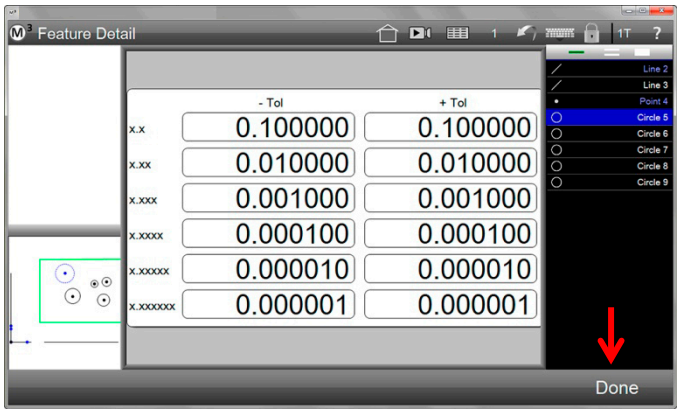
2 Navigate to the Nominal Feature Detail screen as described earlier.

3 Press the Place Table button to display the Place Table.

Measure toolbar > Place Table button



Press the Place Table button



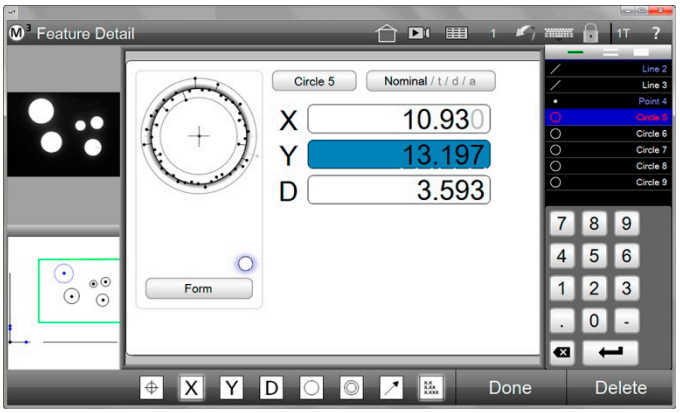
Place table values

4 Modify the values of the place table to represent the values found in the detail block of your engineering drawing if required, then press the Done button to return to the Feature Detail screen.

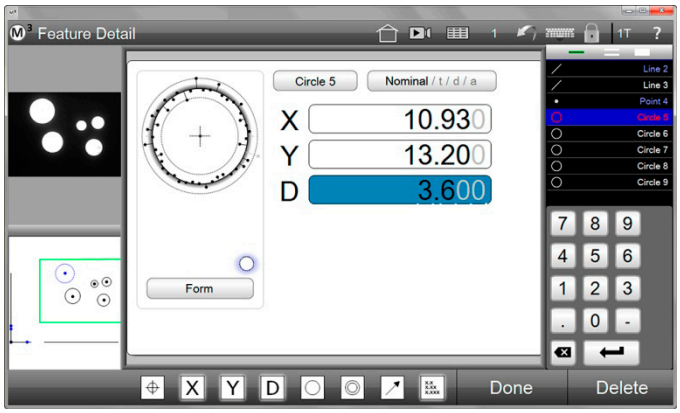
Measure toolbar > Done button

5 Navigate back to the Nominal Feature Detail screen if necessary, and enter the nominal values using one of the methods described earlier. Click the non-significant digits in each field to apply the Place Table values. Non-significant digits will become gray to indicate that the values have been applied. Press the Done button to conclude this process.

Measure toolbar > Done button



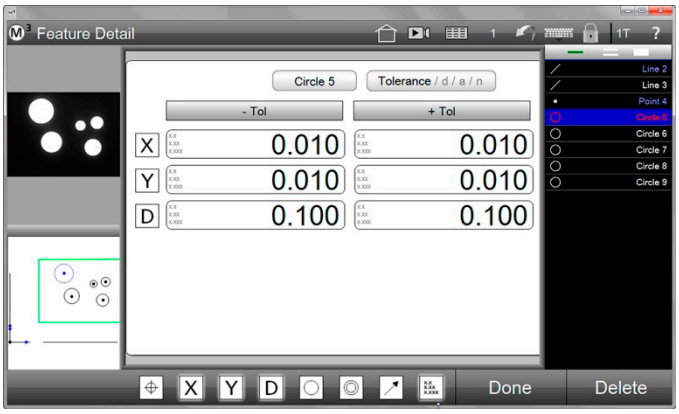
Enter nominal values



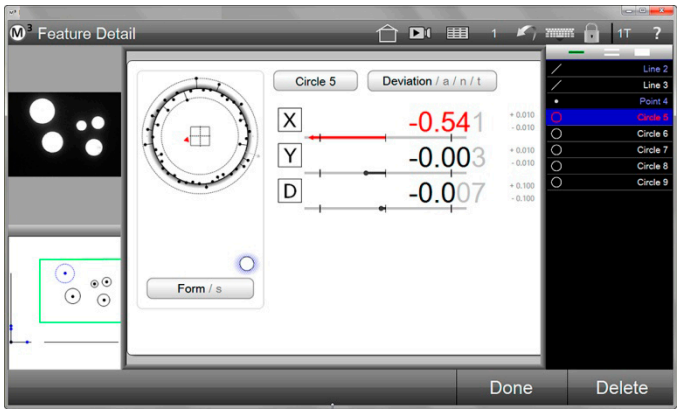
Press non-significant digits

6 Press the Nominal button to go to the Tolerance Feature Detail screen. Tolerances from the Place Tolerance table will have been applied.

7 Press the Tolerance button to go to the Deviation Feature Detail screen and observe the results of the tolerance tests.



Place Table tolerances applied

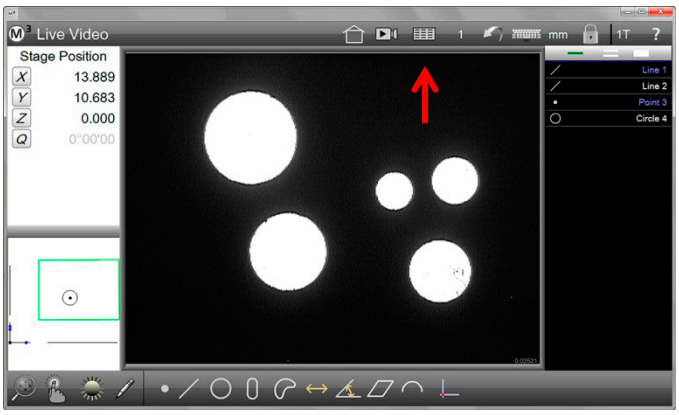


Tolerance results

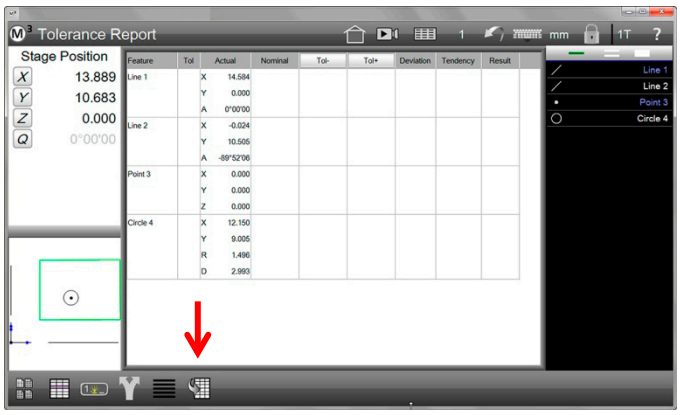
Importing Nominal Values from a DXF Drawing File

Nominal values for part features can be imported directly into the nominal data fields from a DXF drawing file, eliminating the need to do manual nominal data entry. Nominal values can be imported for single feature or group feature tolerancing. To import nominal values from a DXF file:

1 Press the Report button to display the Report screen and Report toolbar.
System toolbar > Report button



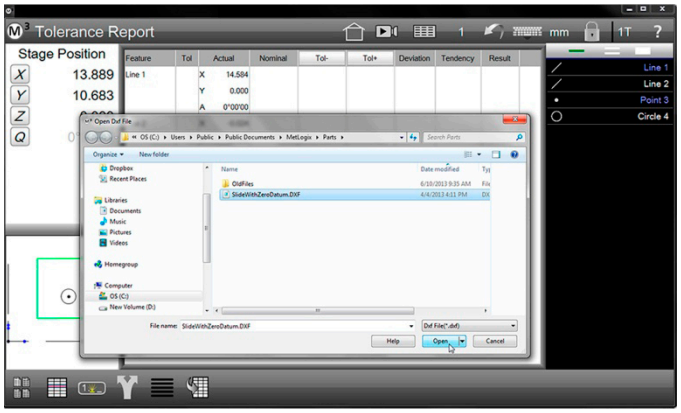
Press the Report button



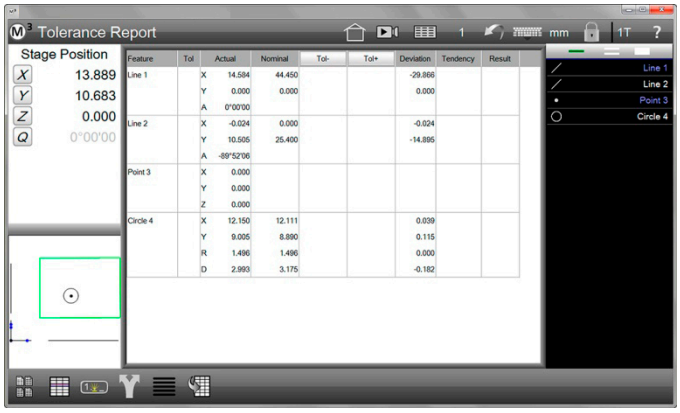
Press the Import DXF button

2 Press the Import DXF button to display the DXF Import dialog.
Report toolbar > Import DXF button

3 Select and open the desired DXF part drawing file. Nominal values will be shown for part features in the Report screen.

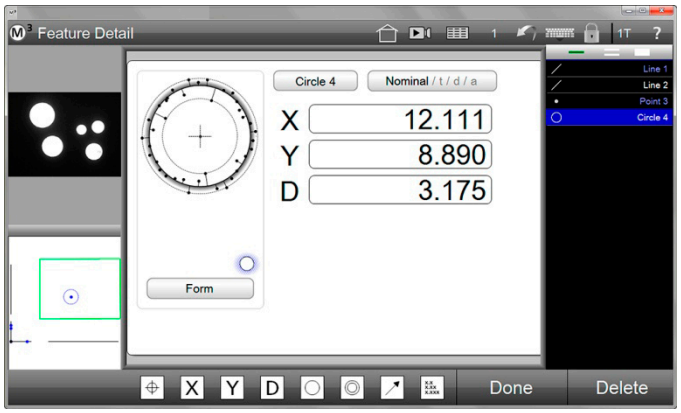


Open the DXF drawing file



Nominal values will be imported

4 Select a feature from the Feature List and navigate to the Nominal Feature Detail screen to confirm the presence of nominal values.



Nominals are present

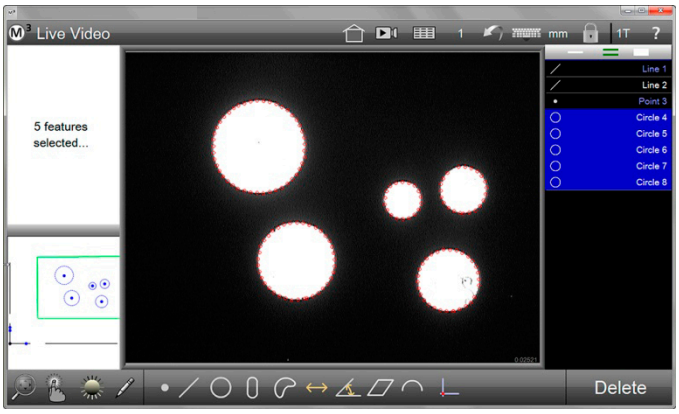
5 Proceed to the Tolerance Feature Detail screen to enter tolerance values.

Applying Tolerances to a Group of Features

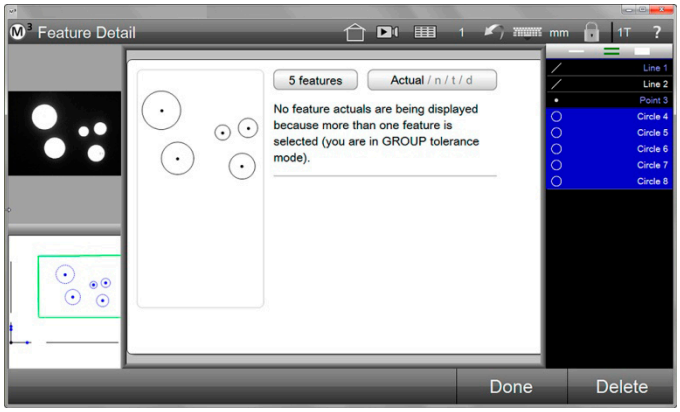
Only a group of like features can be toleranced as a group. This example demonstrates applying a tolerance to a group of circles. To tolerance a group of like features:

1 Select the desired group of like features using any of the methods described earlier in this guide.

2 Press the small Feature Detail viewport to display the large Feature Detail screen in the center viewport.



Select a group of like features

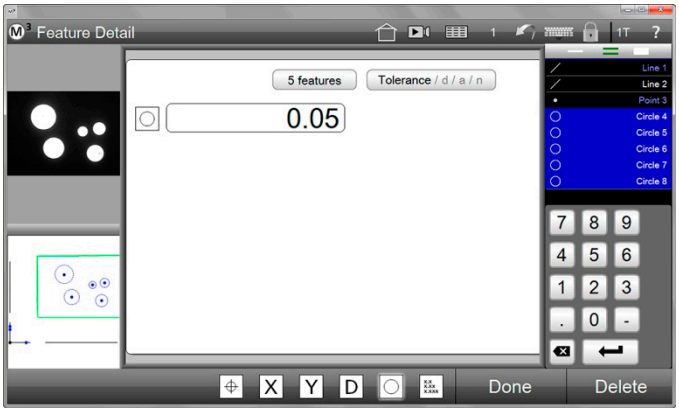


The Feature Detail will indicate group tolerancing

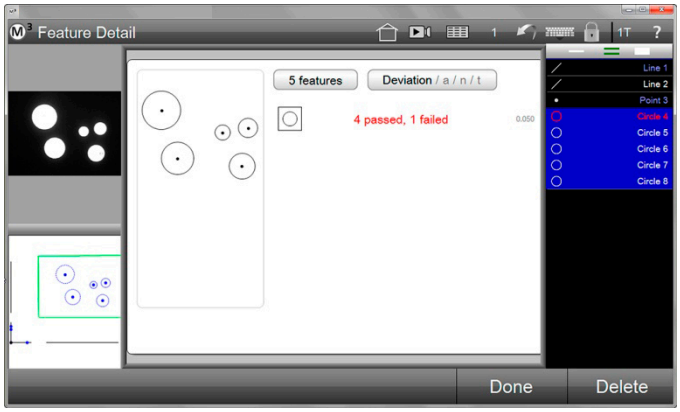
3 Navigate to the Tolerance screen and select the desired tolerance type. Circularity is selected in this example.

4 Enter the tolerance value and press the Done button. Group tolerance results will be shown in the Deviation screen.

Measure toolbar > Done button



Select a tolerance type and enter the value



Group results are shown in the Deviation screen

Quick Tolerance Functions

Quick tolerance functions are available in the interactive and program editing modes to reduce the time required to apply nominal and tolerance values to feature coefficients.

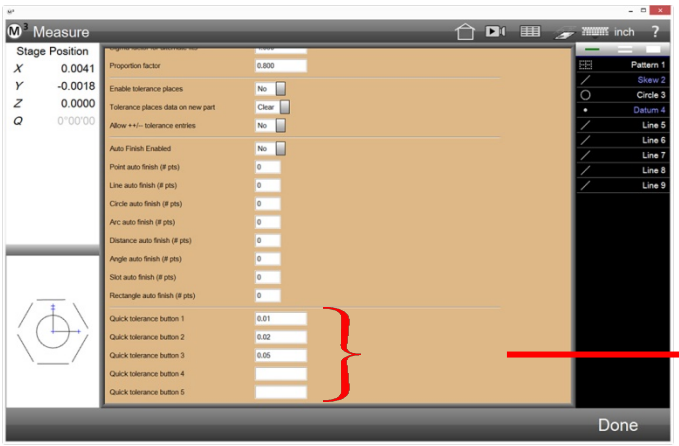
Preset quick tolerance values are provided in the Report toolbar when the Edit Cells function is enabled in the measuring or program editing modes. Up to 5 preset values can be configured by the user to best support tolerance activities for the current measurement session. Simple numeric and angular (degrees, minutes, seconds) values can be included.

1 Edit Cells
2 Preset tolerance values

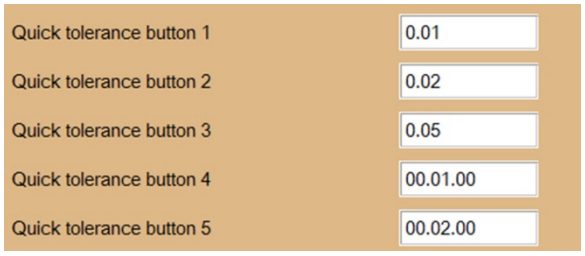
Configuring Preset Tolerance Values

To configure quick tolerance preset values:

1 Display the Measure Setting screen and scroll to the Quick Tolerance preset fields. Enter the desired values.
M3 Menu > Settings > Measure button



Measure Settings screen



Quick Tolerance preset value fields

NOTE
Angles are entered in the format: DD.MM.SS

Accessing the Quick Tolerance Functions

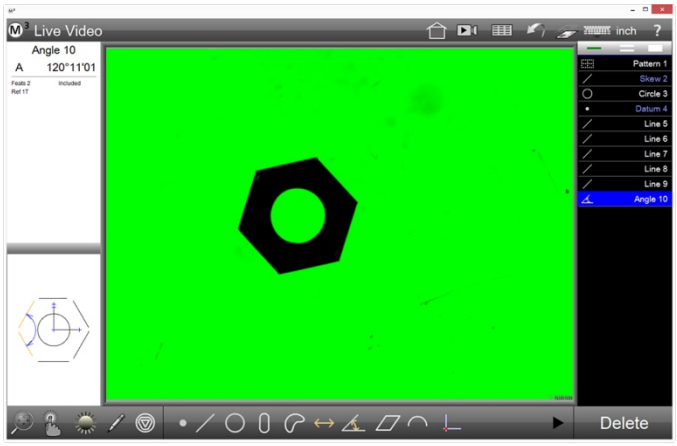
Quick tolerance functions are available in the Measure and Program Edit modes.

Interactive Mode

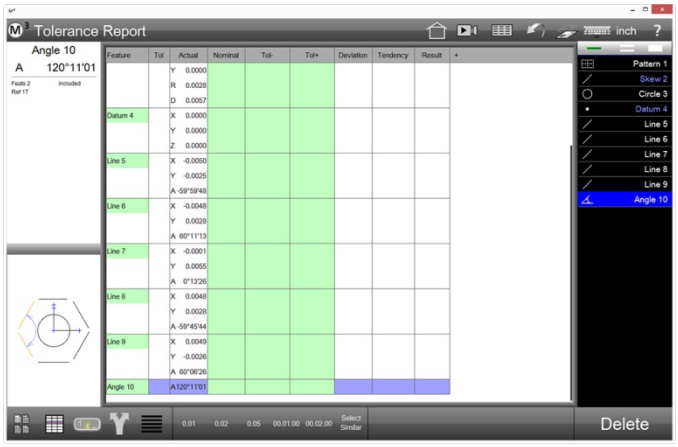
To apply quick tolerances before completing a measurement session and part program:

1 Press the Report button to display the Report screen and then press the Edit Cells button to display the editable cells.

System toolbar > Report button
Report toolbar > Edit Cells button



From measure mode, press the Report button



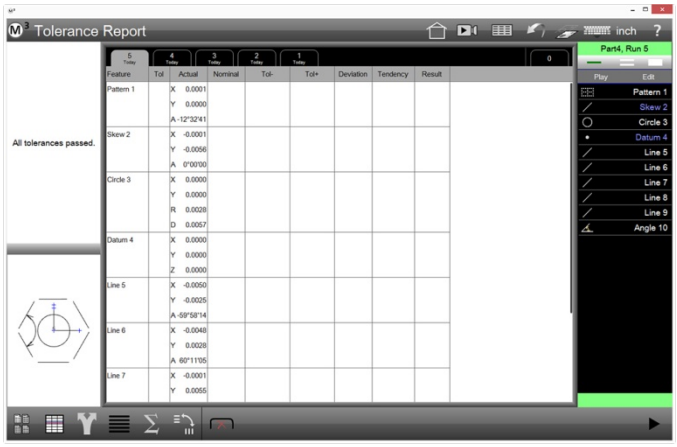
Press the Edit Cells button

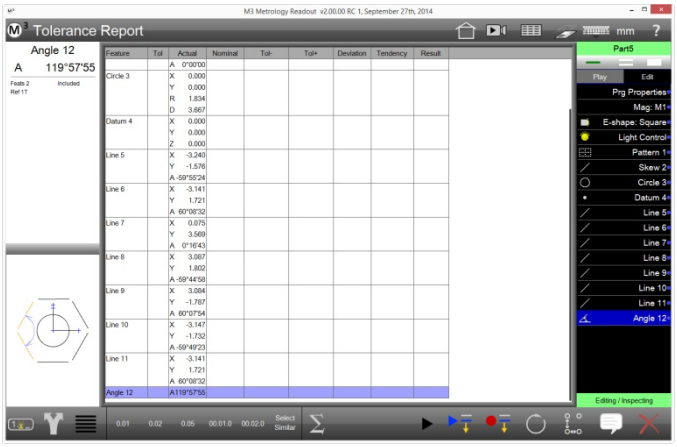
Program Edit Mode

To apply quick tolerances from the edit mode after completing a part program:

1 Press the Edit tab, select a feature in the Feature List and then press the Edit Cells button.

Feature List > Edit tab > Select feature
Report toolbar > Edit Cells button





Applying Quick Tolerances

Nominal values and tolerances can be applied to individual feature coefficients, or to multiple feature coefficients with identical nominal or tolerance value requirements.

Applying Nominal Values and Tolerances to Individual Coefficients

To apply nominal and tolerance values to individual feature coefficients:

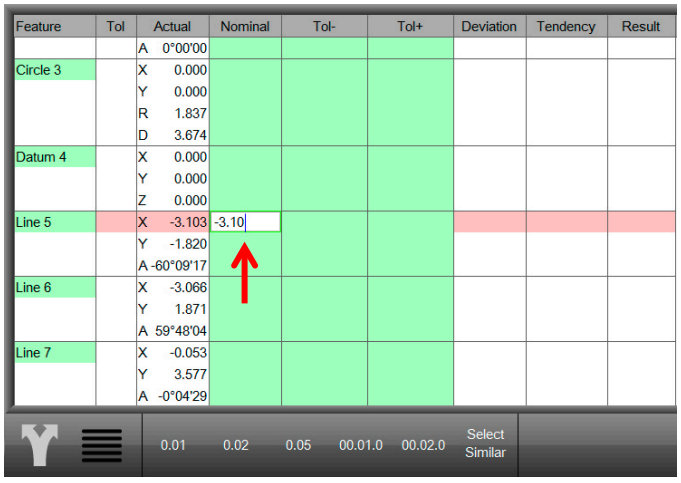
1 Press the single feature selection tool.

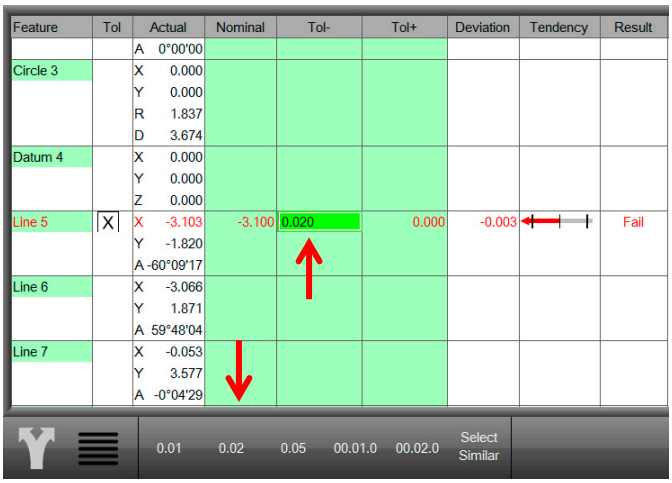
Feature List > Single Select bar

2 Press a nominal cell and enter the required coefficient nominal value.

3 Press Enter to advance to the first tolerance cell for this feature coefficient and press a preset tolerance value to enter it into the cell.

Report toolbar> Preset tolerance

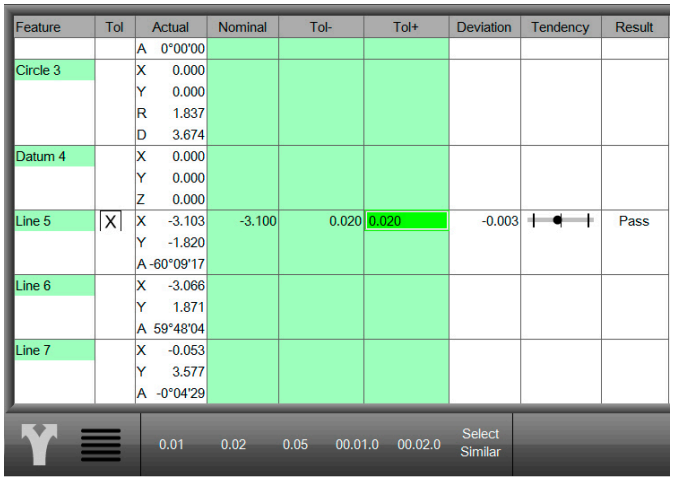


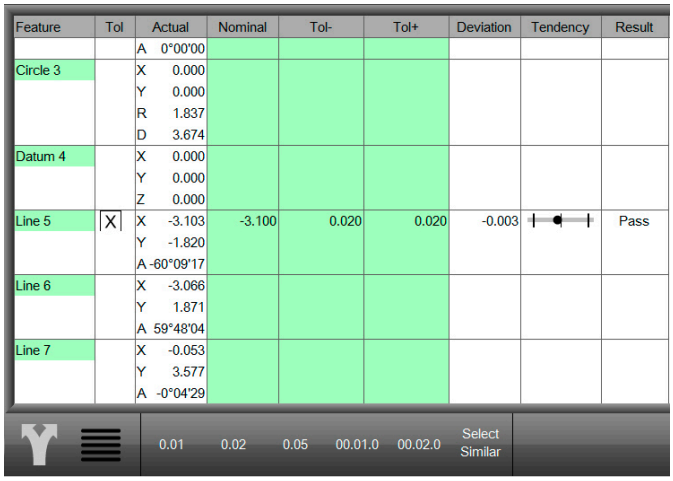


4 Press Enter again to advance to the next tolerance cell and press a preset tolerance value.

Report toolbar > Preset value

5 Press Enter to finish this feature coefficient.



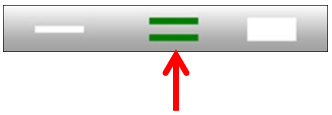


Applying Nominal Values and Tolerances to Multiple Feature Coefficients

Identical nominal values and tolerances can be applied to multiple feature coefficients.

Applying One Nominal Value to Multiple Coefficients

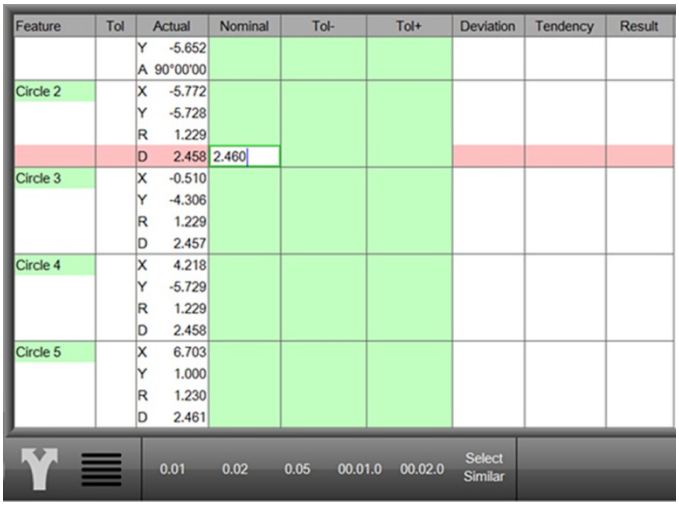
To apply the same nominal value to multiple feature coefficients:



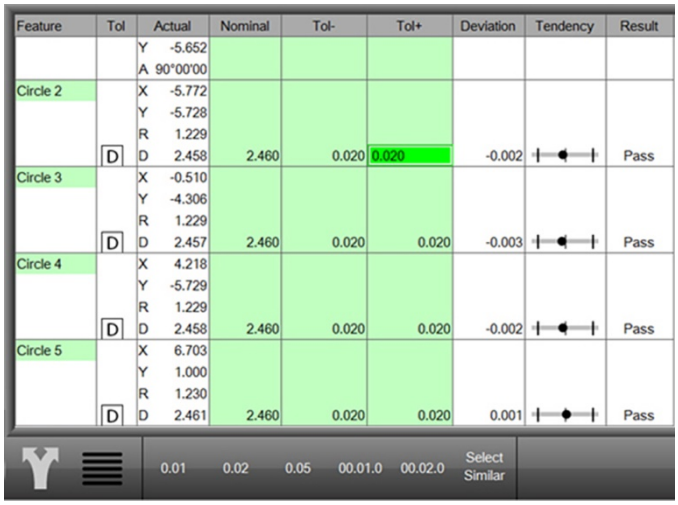
1 Press the multi-feature selection tool
Feature List > Multi-Select bar

2 Press the first of a collection of feature coefficient nominal cells that will contain the same nominal value and then enter the value. In this example 4 Identical Circle diameters have a nominal value of 2.460 mm.

3 Press the remaining feature coefficient nominal cells in the collection and then press Enter. The nominal value will be entered into all the cells and the first tolerance cell of the collection of coefficients will be highlighted. Press a preset tolerance value to enter a tolerance into the first tolerance cell of the collection.
Report toolbar > Preset value



4 Press all remaining cells that require this same tolerance value. In this example, all the diameter tolerance values were 0.02 mm. Press enter to complete the assignment of tolerances to the feature coefficients.



Section 9: Reporting Data and the Results Screen

As part programs are run, the runs database is accumulated and stored in the associated part file. The runs database is shown in the tabbed Results screen. The Result screen is displayed automatically at the conclusion of each part program execution with a new tab showing measurement results for the last program run. The Results screen can be accessed from other screens by pressing the Reports button the System toolbar.

The display of data in the Results screen shows program run tabs across the top with the most recent run tab on the left. Data is organized with features on the Y-axis and feature coefficients on the X-axis. Any run can be selected by pressing the desired run tab. Press and swipe the screen horizontally to display runs that are off-screen.

Report Template Formats

Feature measurement results are displayed in one of four standard report formats and can be reviewed within the Report screen. By default, all features contained in the Feature List will appear in each of the four report formats. Report styles range from simple, including only feature numbers and actual measured values, to complex, including graphic representations of the results and tolerance data. It is also possible to save templates to user defined template names.

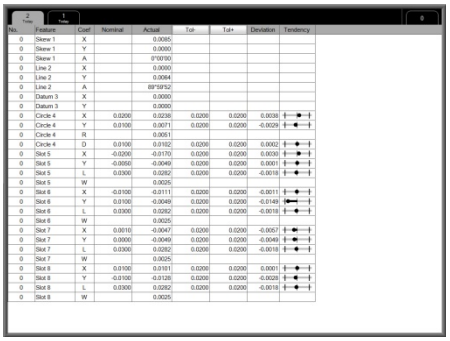
Reports are made up of rows and columns of data. Individual data cells within a report view can be edited to make changes to feature names, feature numbers and tolerance values. The report editing mode can be enabled or disabled quickly using the Edit Cell button. Individual rows of feature data can be hidden from any report format to create fully customized reports for printing or exporting. The data format, report header information and header/footer graphics can be customized by the user. Part view graphics, time and date stamp and operator information can be included in any report format.

Reports can be viewed on screen, printed or exported at the conclusion of a single part measurement or included in a part program to support repetitive and automated measurement and reporting. Reports can be printed as hardcopies to standard Windows-compatible printers or exported as data files in popular file formats.

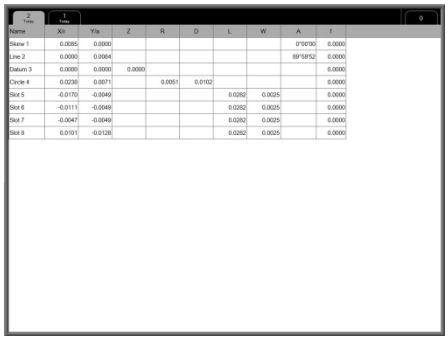
Selecting a Report Format

Any one of four report formats can be selected:

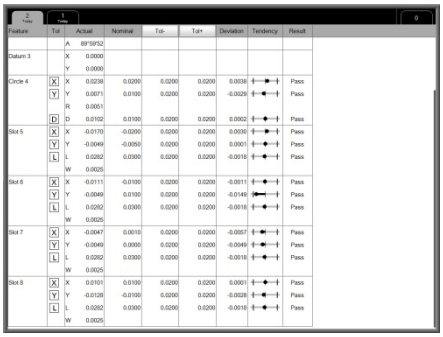
- European
- CSV
- Tolerance Report
- European 2



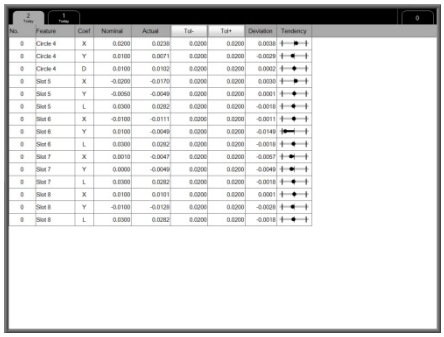
European



CSV



Tolerance Report

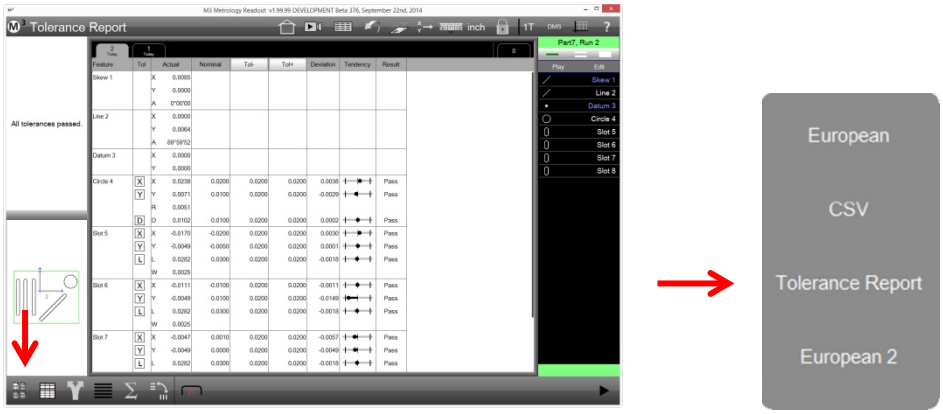


European 2

To select a report format:

1 Press the Report button to display the Report screen in the center viewport.
System toolbar > Report button

2 Select the desired report format from the Report Format menu.
Report toolbar > Report Format button



Press the Report Format button

The selected format will be displayed with the report name in the upper left corner of the Report screen.

Editing Report Templates

Any of the four report template types can be modified when the system is in the interactive mode. Templates cannot be modified in the playback or program editing modes. Template modifications include:

- Adding or deleting template categories
- Column order
- Column width
- Number of horizontal lines
- Number of vertical lines
- Number of feature names for European templates
- Row height

Template modifications can be:

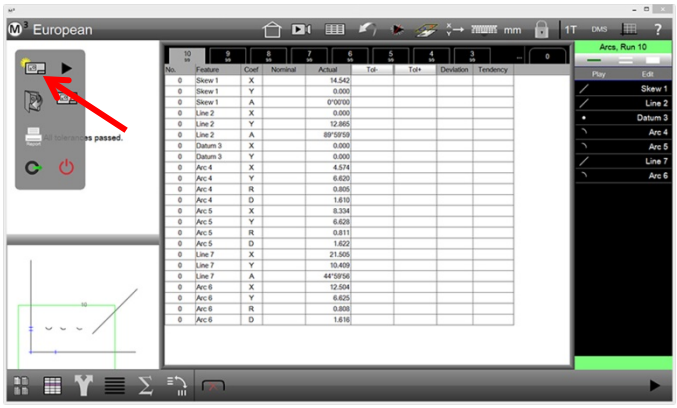
- Reset to the factory default
- Discarding changes before saving
- Saved as the new default for the modified template

Accessing Report Template Editing Functions

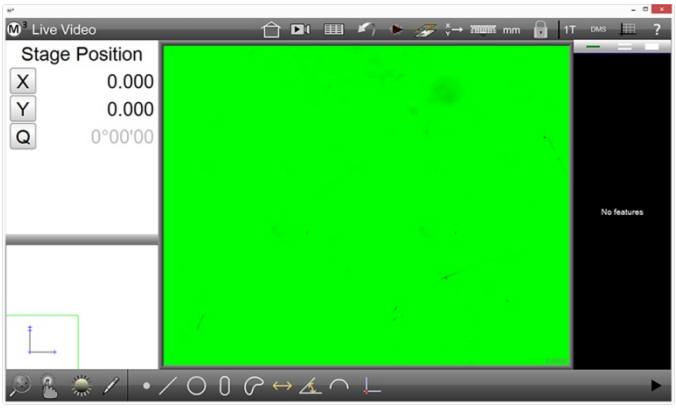
To access the report template editing functions:

1 Press the load New Part icon off the M3 menu to load a new part.

System toolbar > M3 Menu > New Part

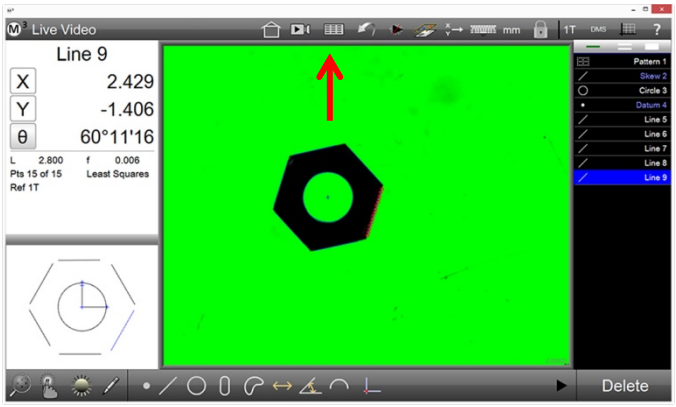


Load a new part

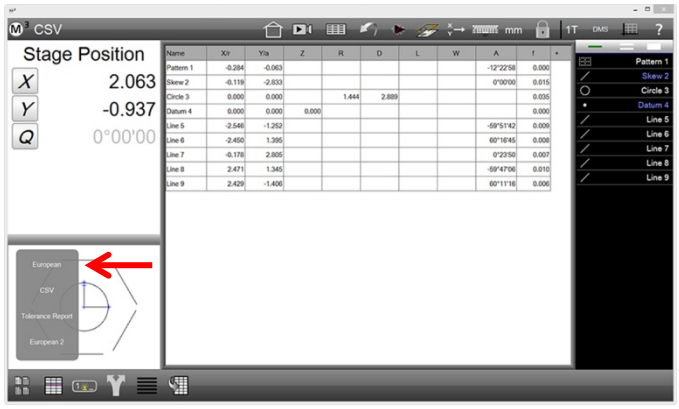


New part in interactive mode

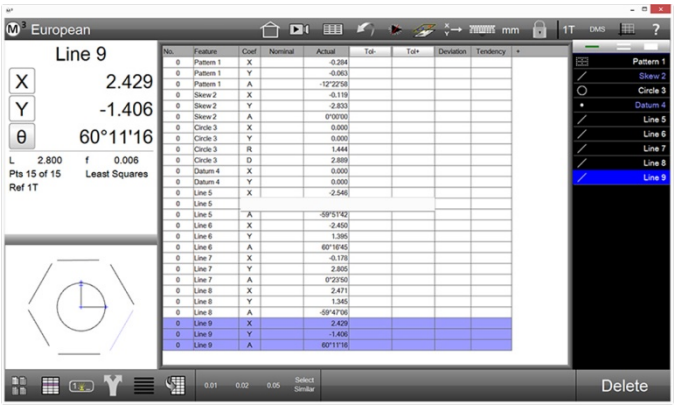
2 Place a piece in the field of view and measure a few features to populate the Feature List with data.



3 Press the Report button in the System toolbar and then the Report Type button in the Report toolbar to select one of the four report template types. The European type is selected in this example.



Select a report type

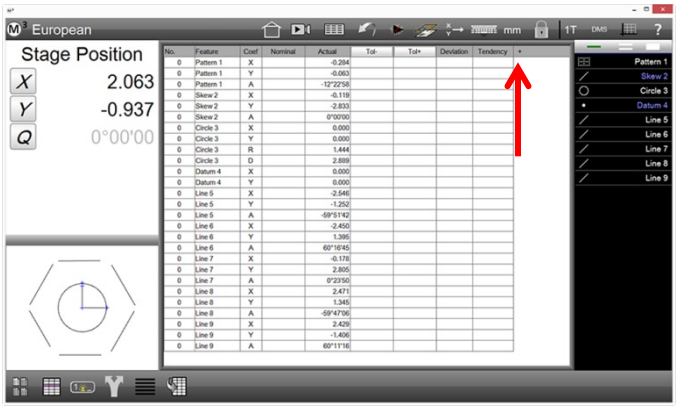


The European Report Type is Selected

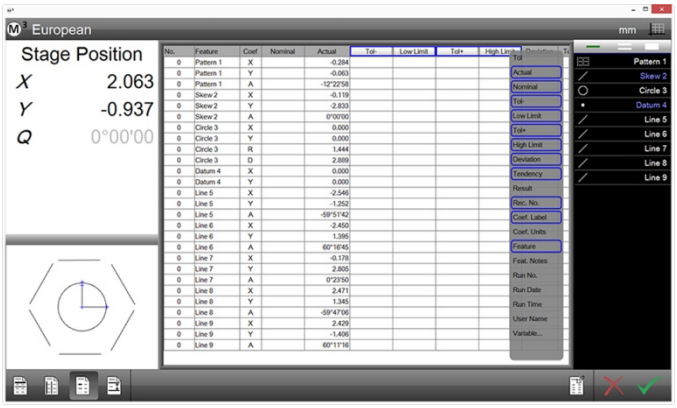
Displaying Report Edit Functions

To display the report edit functions:

1 Press the small + in the top right corner of the report screen to display the list of report categories and the Report Edit toolbar at the bottom of the screen.



Press the + sign to display the report categories list and the Report Edit toolbar



Report category list and Report Edit toolbar displayed

The Report Edit toolbar functions include:

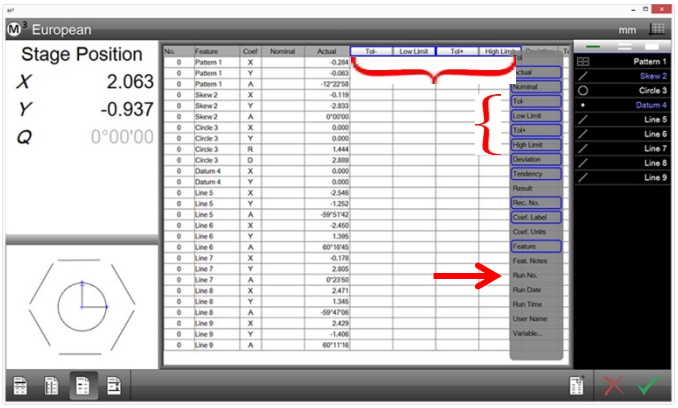
1 Horizontal Lines
2 Vertical Lines
3 Fill Column (European template)
4 Row Height
5 Reset to Factory Default
6 Discard Changes
7 Save Changes

Report Edit toolbar

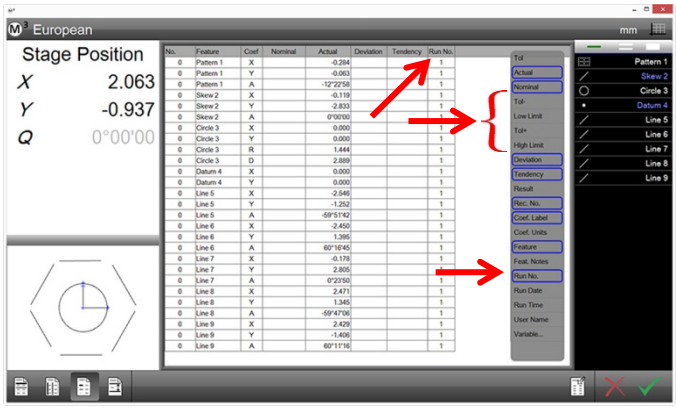
Adding or Deleting Template Categories (Columns)

To add or delete report categories:

1 Press on report categories in the list to add or delete categories from the current report template.



Four report categories deleted and one category added

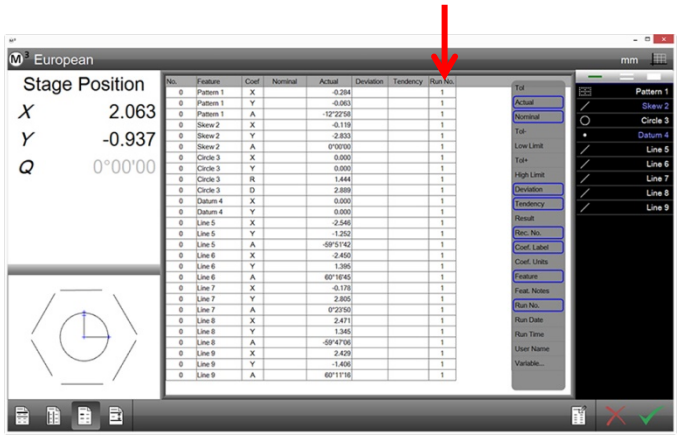


The changes are shown in the modified template

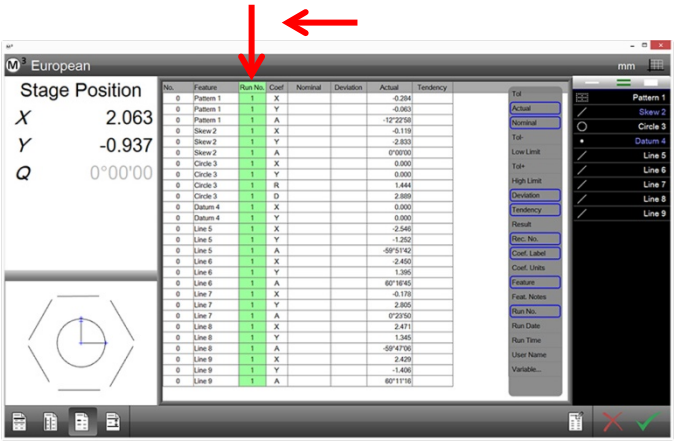
Changing Column Order

To change the column order:

1 Press and drag columns left or right to change the report column order.



Press the column header…

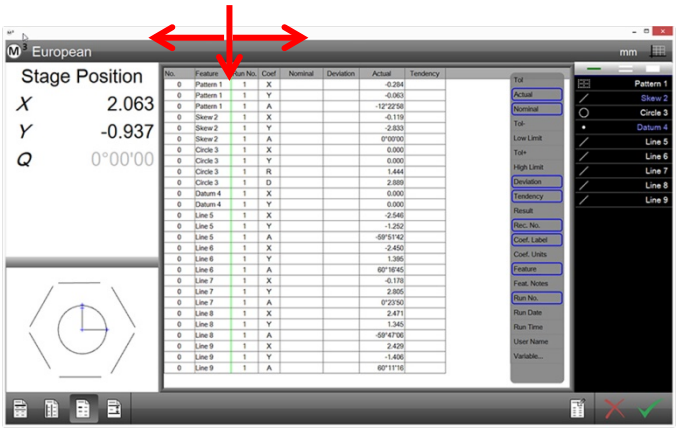


And drag the column to the desired location

Changing Column Width

To change the column width:

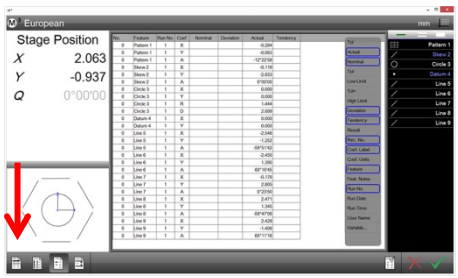
1 Press and drag the column dividing lines to resize column width.



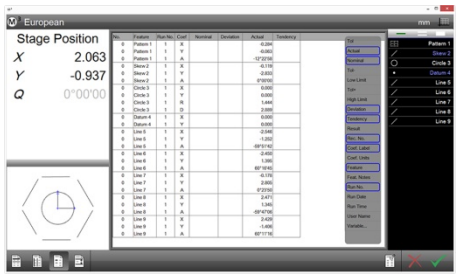
Increasing or Decreasing Horizontal Lines

To increase or decrease horizontal lines:

1 Press the Horizontal Line button in the Report Edit toolbar to toggle through the horizontal report line options:
   - One line per feature coefficient
   - One line per feature
   - No lines



One line per feature coefficient



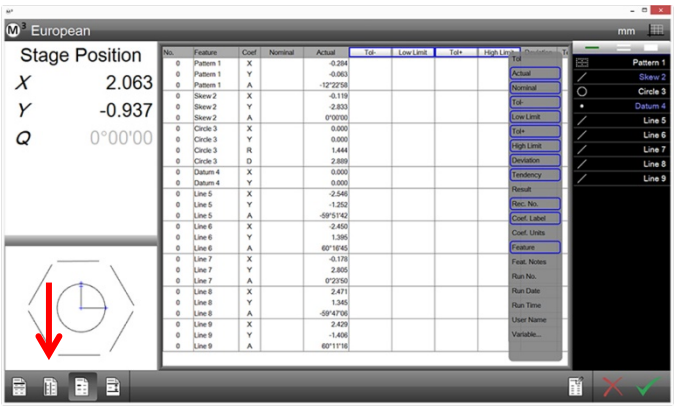
One line per feature

No lines

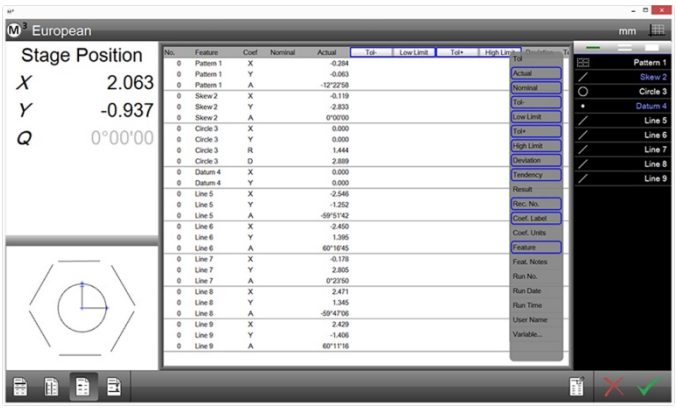
Increasing or Decreasing Vertical Lines

To increase or decrease vertical lines:

1 Press the Vertical Line button to toggle the vertical report line options:
- Vertical lines separating columns
- No vertical lines



Vertical lines

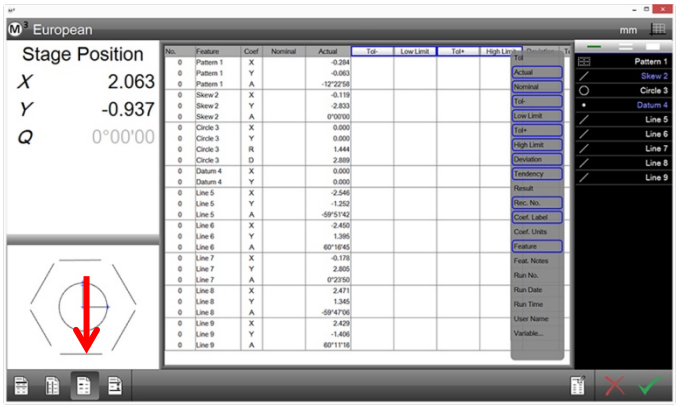


No vertical lines

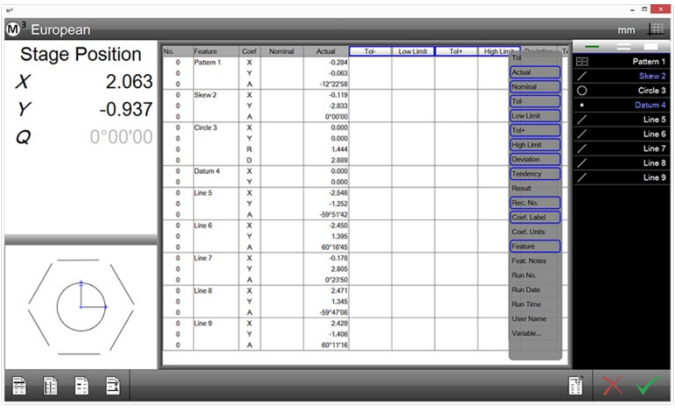
Changing the Number of Feature Names for European Templates

To change the number of feature names:

1 Press the Fill Column button to toggle the feature name options for the European template:
- Feature name on every line
- Feature name on only the first line of each feature



Feature name on every line



Feature name on first line

Changing the Row Height

To change the row height:

1 Press the Row Height button to toggle through the four preset report Row height options.

No.	Feature	Coef	Nominal	Actual
0	Pattern 1	X		-0.284
0	Pattern 1	Y		-0.063
0	Pattern 1	A		-12°22'58
0	Skew 2	X		-0.119
0	Skew 2	Y		-2.833
0	Skew 2	A		0°00'00
0	Circle 3	X		0.000
0	Circle 3	Y		0.000
0	Circle 3	R		1.444
0	Circle 3	D		2.889
0	Datum 4	X		0.000
0	Datum 4	Y		0.000
0	Line 5	X		-2.546
0	Line 5	Y		-1.252
0	Line 5	A		-59°51'42
0	Line 6	X		-2.450
0	Line 6	Y		1.395
0	Line 6	A		60°16'45
0	Line 7	X		-0.178
0	Line 7	Y		2.805
0	Line 7	A		0°23'50

No.	Feature	Coef	Nominal	Actual
0	Pattern 1	X		-0.284
0	Pattern 1	Y		-0.063
0	Pattern 1	A		-12°22'58
0	Skew 2	X		-0.119
0	Skew 2	Y		-2.833
0	Skew 2	A		0°00'00
0	Circle 3	X		0.000
0	Circle 3	Y		0.000
0	Circle 3	R		1.444
0	Circle 3	D		2.889
0	Datum 4	X		0.000
0	Datum 4	Y		0.000
0	Line 5	X		-2.546
0	Line 5	Y		-1.252
0	Line 5	A		-59°51'42
0	Line 6	X		-2.450
0	Line 6	Y		1.395
0	Line 6	A		60°16'45
0	Line 7	X		-0.178
0	Line 7	Y		2.805
0	Line 7	A		0°23'50
0	Line 8	X		2.471
0	Line 8	Y		1.345
0	Line 8	A		-59°47'06

No.	Feature	Coef	Nominal	Actual
0	Pattern 1	X		-0.284
0	Pattern 1	Y		-0.063
0	Pattern 1	A		-12°22'58
0	Skew 2	X		-0.119
0	Skew 2	Y		-2.833
0	Skew 2	A		0°00'00
0	Circle 3	X		0.000
0	Circle 3	Y		0.000
0	Circle 3	R		1.444
0	Circle 3	D		2.889
0	Datum 4	X		0.000
0	Datum 4	Y		0.000
0	Line 5	X		-2.546
0	Line 5	Y		-1.252
0	Line 5	A		-59°51'42
0	Line 6	X		-2.450
0	Line 6	Y		1.395
0	Line 6	A		60°16'45
0	Line 7	X		-0.178
0	Line 7	Y		2.805
0	Line 7	A		0°23'50
0	Line 8	X		2.471
0	Line 8	Y		1.345
0	Line 8	A		-59°47'06
0	Line 9	X		2.429
0	Line 9	Y		-1.406
0	Line 9	A		60°11'16

No.	Feature	Coef	Nominal	Actual
0	Pattern 1	X		-0.284
0	Pattern 1	Y		-0.063
0	Pattern 1	A		-12°22'58
0	Skew 2	X		-0.119
0	Skew 2	Y		-2.833
0	Skew 2	A		0°00'00
0	Circle 3	X		0.000
0	Circle 3	Y		0.000
0	Circle 3	R		1.444
0	Circle 3	D		2.889
0	Datum 4	X		0.000
0	Datum 4	Y		0.000
0	Line 5	X		-2.546
0	Line 5	Y		-1.252
0	Line 5	A		-59°51'42
0	Line 6	X		-2.450
0	Line 6	Y		1.395
0	Line 6	A		60°16'45
0	Line 7	X		-0.178
0	Line 7	Y		2.805
0	Line 7	A		0°23'50
0	Line 8	X		2.471
0	Line 8	Y		1.345
0	Line 8	A		-59°47'06
0	Line 9	X		2.429
0	Line 9	Y		-1.406
0	Line 9	A		60°11'16

Resetting Template Modifications to the Factory Default

1 Press the Reset Template button to revert to the factory default state of the modified template.

Discarding Changes Before Saving

1 Press the Abandon Changes button to clear report template modifications made since the template was last saved.

Saving Template Changes as the New Default

1 Press the Save Template button to save template modifications.

NOTE

Saved template modifications will be reflected in all future uses of the template in the M3 system.

User Defined Report Templates

-Customize and save multiple report template styles for use in various part programs.

-Report templates can now be created and assigned to meet the formatting requirements of specific part inspection routines.

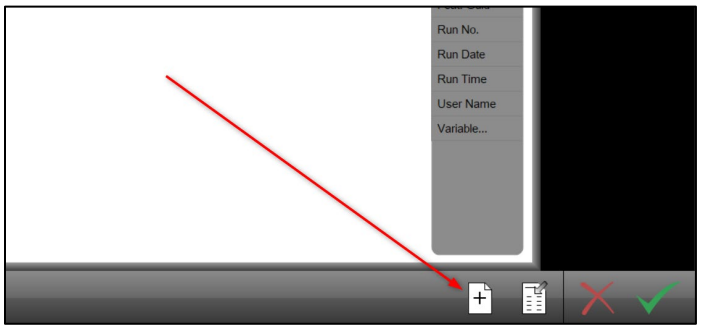
Setup:

No additional setup is required to utilize the new User Templates functionality.

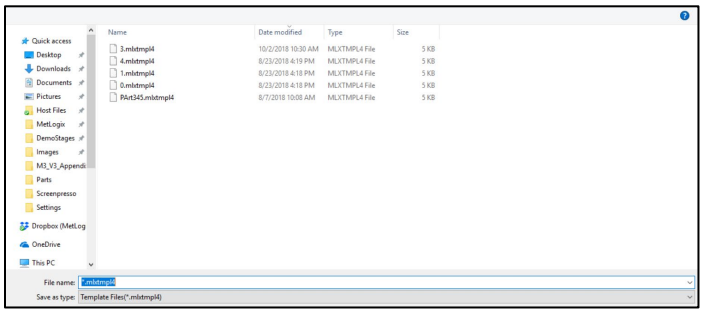
Operation:

IMPORTANT: Template selection, editing, and usage for the purpose of data export and report printing remains unchanged from the previous versions of the M3 software. The following enhancement to the template edit and save system simply allows users to save and recall as many variations of the (4) main default templates as they would like.

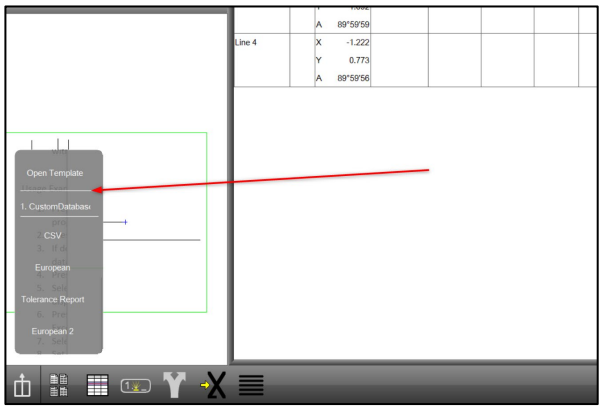
➔ A new button has been added to the bottom toolbar of the template edit mode. This button allows users to save the current template state to a new template filename.



➔ Once the Save New Template button is pressed the user will be presented with a windows dialog to allow the user to name the new template, as seen below.



➔ As new User Defined templates are added to the system they will begin to populate the template selection menu, along with the standard default templates. Select any desired template from this menu to set it as current, or use the Open Template button to choose a template from the Windows file dialog.



Sample Procedure Steps for "Save Custom Report Template" function:

1. In interactive mode(no program loaded) access the M3 data view screen.
2. Select the default template that you would like to make modifications to.
3. Press the small "+" button located to the right of the template headers to enter template edit mode.
4. Configure the template using the various template customization options.
5. Once the desired template is achieved, press the Save New Template button found in the bottom right toolbar.
6. Enter a name for the new template and press save. The newly saved template will be set as the current template and will be added to the template selection menu.

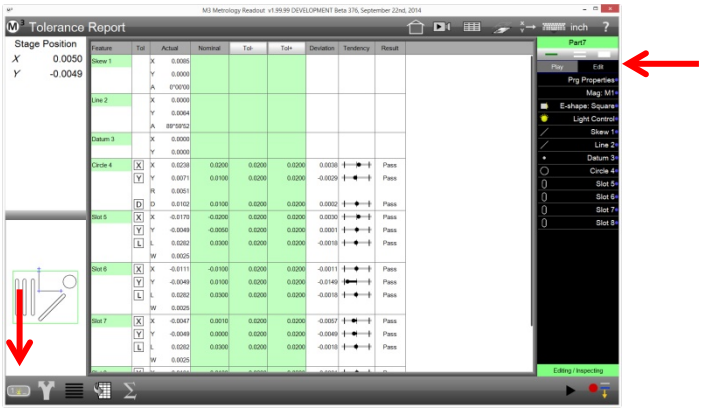
Editing Report Cells

Feature names, feature numbers and tolerance values can be edited in the Report screen. To edit report cells:

1 Press the Report button to display the Report screen in the center viewport.
System toolbar > Report button

2 Press the Edit Tab to display the Edit Cell button in the report toolbar.
Feature List > Edit Tab

3 Press the edit cell button to enable editing. Data fields that can be edited will be shaded light green.
Report toolbar > Edit Cell button



Cells shaded green can be edited

4 Report data that extends beyond the field of view can be moved up or down by pressing and dragging on the screen.

5 Edit feature names, feature numbers or tolerance values. Tolerance nominal and limit values can be edited.

6 Press the Edit Cell button again to leave the edit mode.

Report toolbar > Edit Cell button

7 Press the Edit Tab again to leave the edit mode.

Feature List > Edit Tab

Hiding Report Data

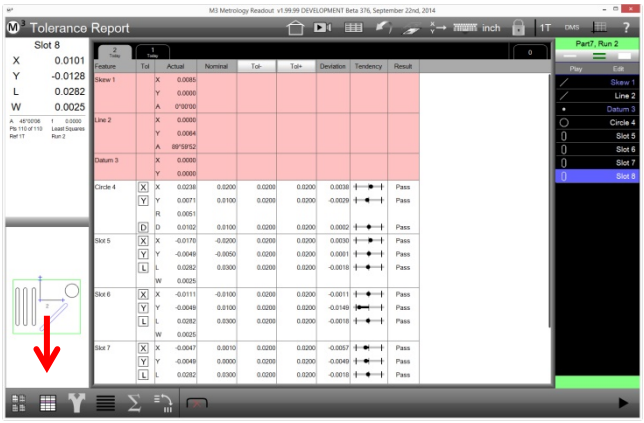
Specific data rows can be hidden from printed or exported reports. To hide one or more rows of data:

1 Press the Report button to display the Report screen in the center viewport.

System toolbar > Report button

2 Select individual rows by pressing on them. Selected rows will be shaded pink. Select multiple rows by pressing the double bar of the feature selection tool and then pressing on the desired rows.

3 Report data that extends beyond the field of view can be moved up or down by pressing and dragging on the screen.



Press the Hide Data button to hide shaded rows
Long-press the button to hide all but the shaded rows

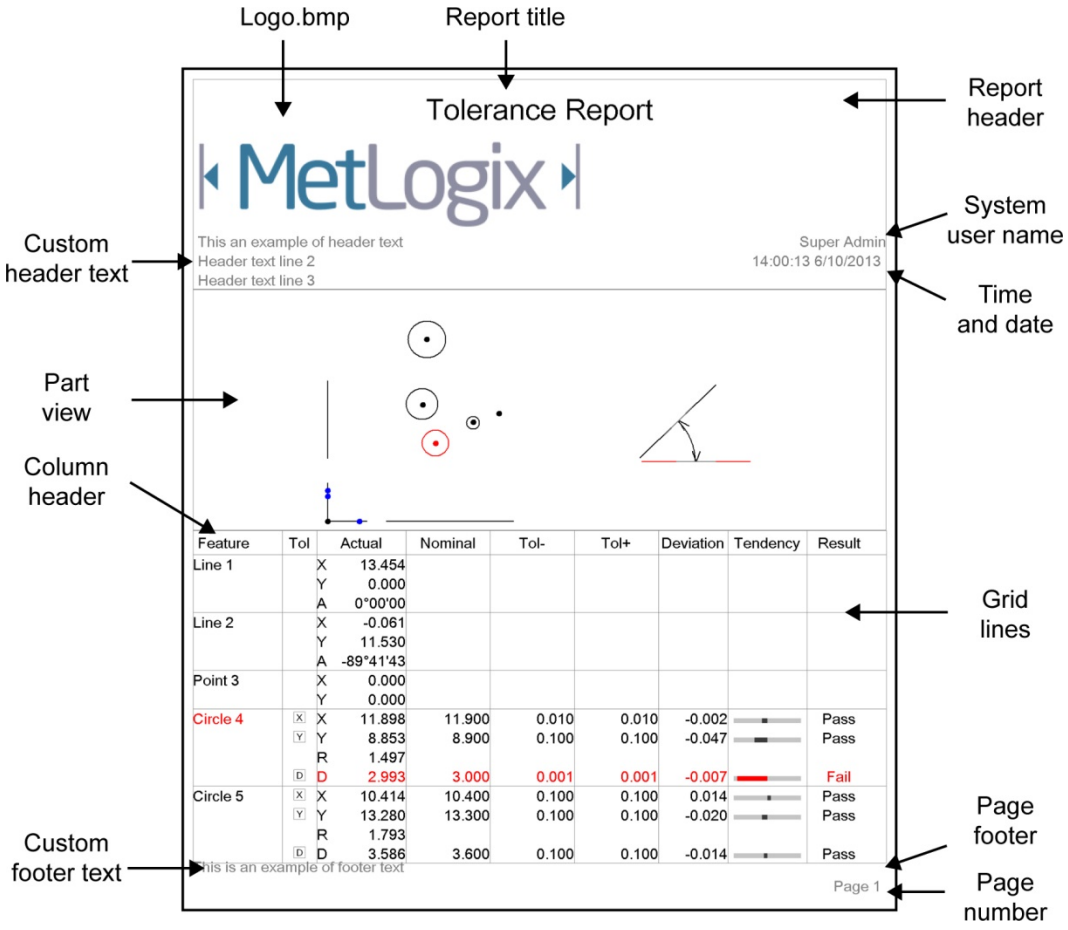
4 Press the Hide Data button to hide the shaded rows from printed reports and exported files. Long-press the Hide Data button to hide all rows except the shaded rows from printed reports and exported files.

Report toolbar > Hide Data button

5 To restore hidden data for a given feature, press the Hide Data button when the feature is selected.

Customizing Printed Reports

Printed report parameters are shared with all report formats and are customized by specifying the printout settings. A graphic representation of report template printout categories is shown below.



Specifying Printout Settings

The printer settings are found in the Printouts Settings screen. To specify printer settings:

1 Display the Printout Settings screen.

System toolbar > System menu > Printouts button

2 Set the desired parameters in the Printout Settings screen. Printout settings are show below.

Print Report Header

Set this field to Yes to include the report header.

Print Column Header

Set this field to Yes to include column headers. The column header labels the report columns by category.

Print Grid to Lines

Set this field to Yes to include grid lines.

Print Bitmap in Header

Set this field to Left, Center or Right to include a custom bitmap image in the specified location of the report header. Set this field to No to omit a custom bitmap. Place the custom bitmap image with the file name *HeaderLogo.bmp* into: *C:\users\public\public documents\Metlogix\logosandbitmaps*. The size of the custom bitmap should be 397 by 98 pixels.

Print Bitmap in Footer
Set this field to Left, Center or Right to include a custom bitmap image in the specified location of the report header. Set this field to No to omit a custom bitmap. Place the custom bitmap image with the file name *FooterLogo.bmp* into: *C:\users\public\public documents\Metlogix\logos and bitmaps*. The size of the custom bitmap should be 397 by 98 pixels.

Print User Name in Header
Set this field to Left, Center or Right to include the currently logged-in user name in the desired location of the header. Set this field to No to omit the username from the header.

Print Date/Time in Header
Set this field to Left, Center or Right to include the system date and time in the desired location of the header. Set this field to No to omit the date and time.

Print Part View with Data
Set this field to Top, Bottom or Watermark to include the image of the current part view in the desired location. Setting the field to Watermark will print an alpha-blended part view image underneath the report data. Set this field to No to omit the part view image.

Print Page Number in Footer
Set this item to Yes to include a page number in the footer.

Printed Part View Height
Set this item to the desired size of the part view to be included with the report. This item can be set to 25%, 50%, 75% or 100% of the native size of the part view image.

Print Custom Text in Header
Enter custom alphanumeric information into these fields to be displayed in the header. Custom text entered into these fields will always be aligned left and bottom in the header.

Print Custom Text in Footer
Enter custom alphanumeric information into this field to be displayed in the footer. Custom text entered into this field will always be aligned left and bottom in the footer.

Prompt for Custom Text
Set this field to Yes to prompt the user to enter the custom text that will be included in the header and footer when header or footer text was specified. When this field is set to Yes, a dialog box will be displayed each time the user requests a printed report. Custom text can be entered directly into this dialog box and will be included on the printed report.

Results View Visual Review Tools

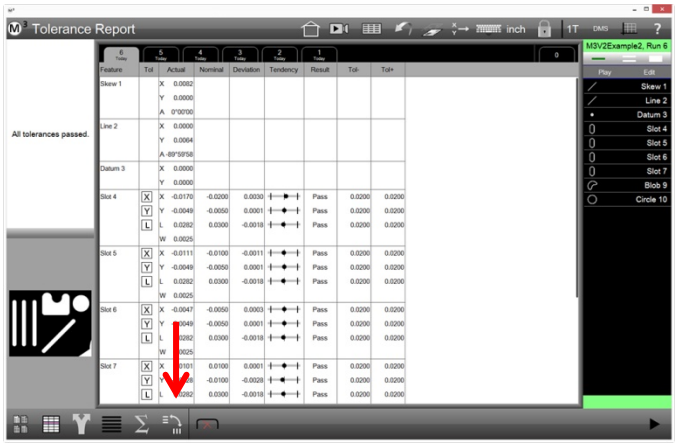
Tools are provided that change and enhance the display of measurement data for visual review. These views cannot be printed as reports but can be exported as .tsv or .csv formatted data. These exports can be performed interactively, but cannot be included in programs. Visual review tools include:

- Pivot view
- Pinned coefficients
- Statistics
- Run zero tab
- Delete button

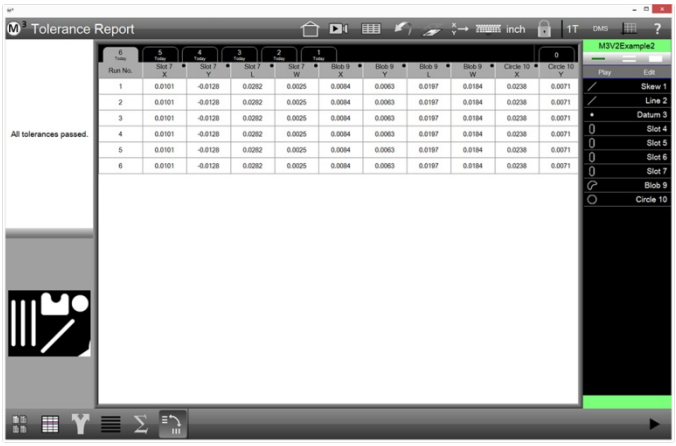
Pivot view

The Pivot view reorganizes the stored run data to display the run numbers on the Y-axis and the feature coefficient data on the X-axis for the selected features. To display the Pivot view:

1 Press the Pivot view button and then select the desired feature(s).
Report toolbar > Pivot view
Feature List > Desired features



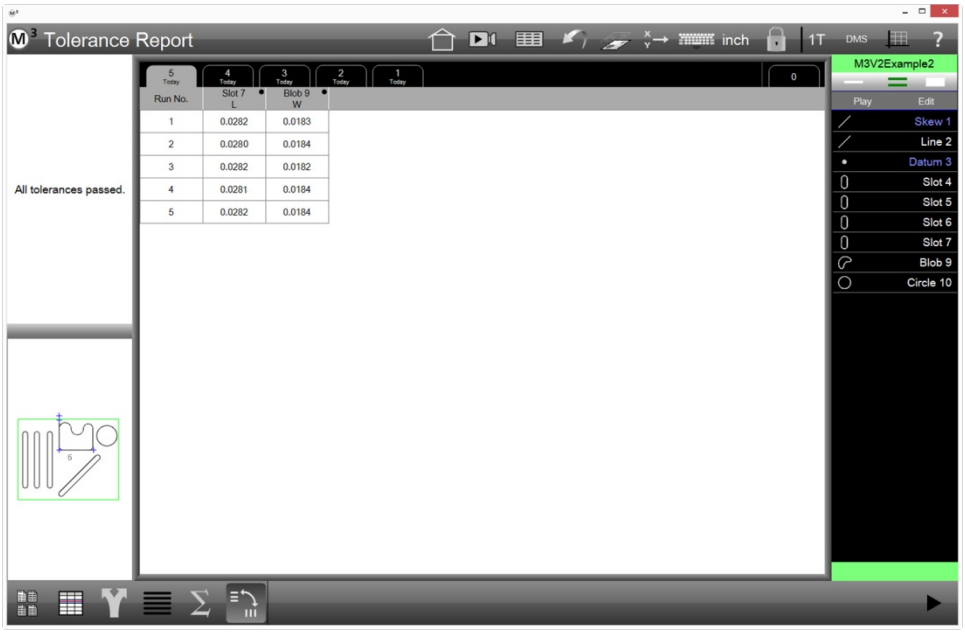
Results view



Pivot view

Pinning Coefficient Data

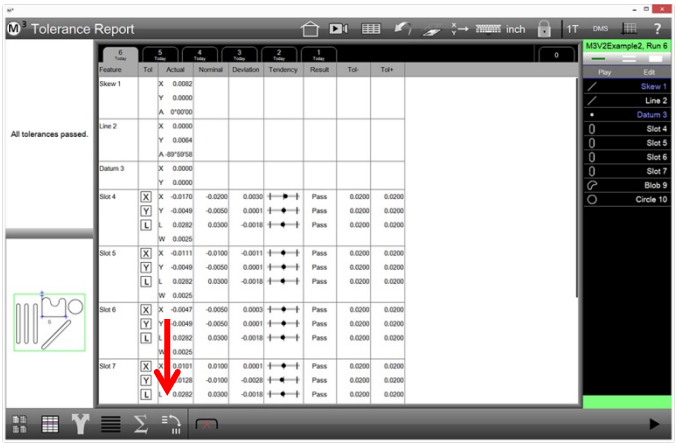
The Pivot view can be changed to display only the desired feature coefficient values across all program runs by pinning the coefficient columns. Pinning coefficient columns are shown in this example. The pinned view shows that the length (L) of slot 7 and the width (W) of blob 9 changed very little over 5 program runs.



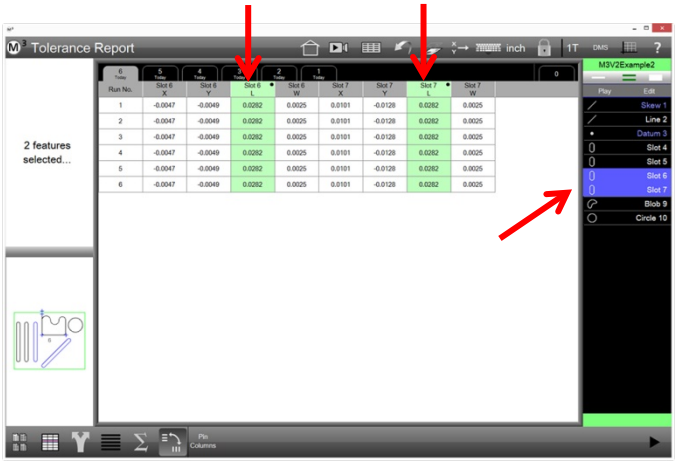
To pin feature coefficients:

1 Display the feature coefficients for a number of program runs as described in the previous pages.
2 Select a feature in the feature list and press a coefficient column header to highlight the column.
3 Press the Pin Columns button in the Report toolbar.
Report toolbar > Pin Columns

A black dot will appear in the right corner of the highlighted header to indicate that the coefficient is pinned.

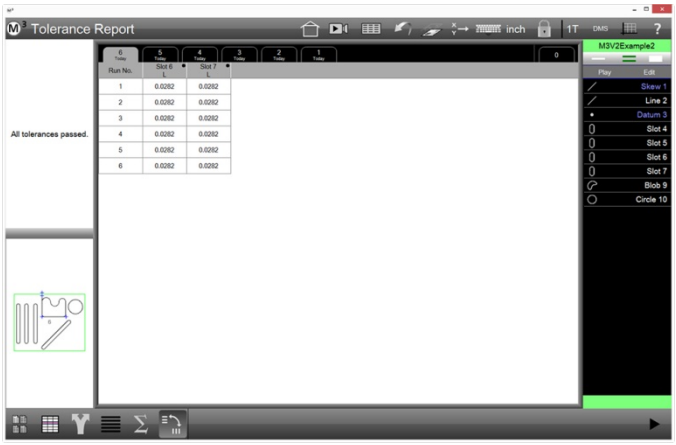


Display Results view then press Pivot button



Select features, press column headers then press Pin Columns button

4 Deselect the feature in the feature list. The pinned column will be retained in the Pivot view.



Repeat steps 2 through 4 to pin additional feature coefficients if desired. Any coefficient from any feature may be pinned.

To unpin coefficients:

1 Press the desired coefficient column header(s) and then press the Pin Columns button.

Statistics

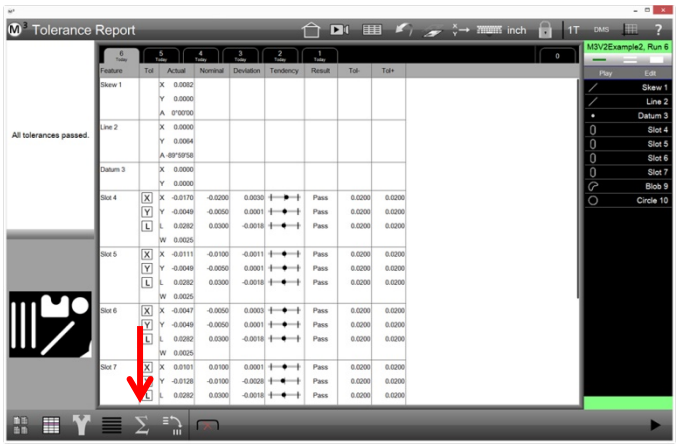
Standard statistics can be shown for the selected feature(s) including:

- Min
- Max
- Range
- Average
- Standard deviation
- Sigma

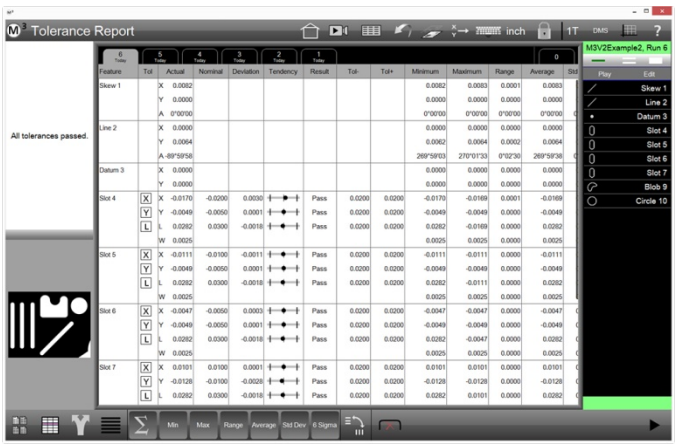
Results View Statistics

To display statistics for features in the results view:

1 Press the Stats button in the Report toolbar. A menu of statistics functions will be displayed in the Report toolbar. All statistics are enabled and displayed for all features in the Results view by default.
Report toolbar > Stats button



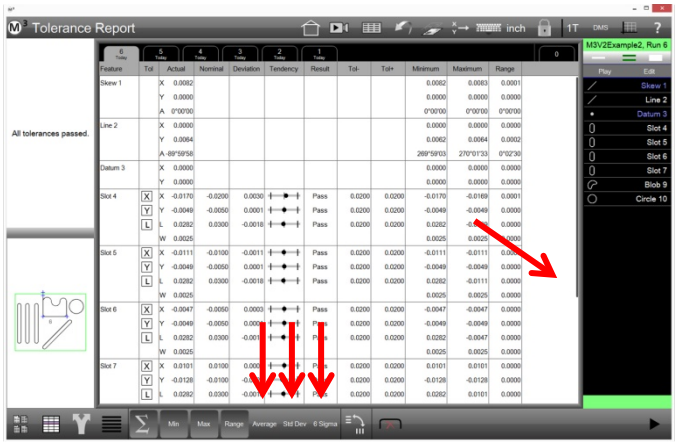
Press the Stats button…



To add statistics to the Results view

2 Press any statistics function button to disable the display of that statistic.
Report > Statistics menu function

3 Press the Statistics button again to remove all statistics from the Results view.
Report toolbar > Stats button



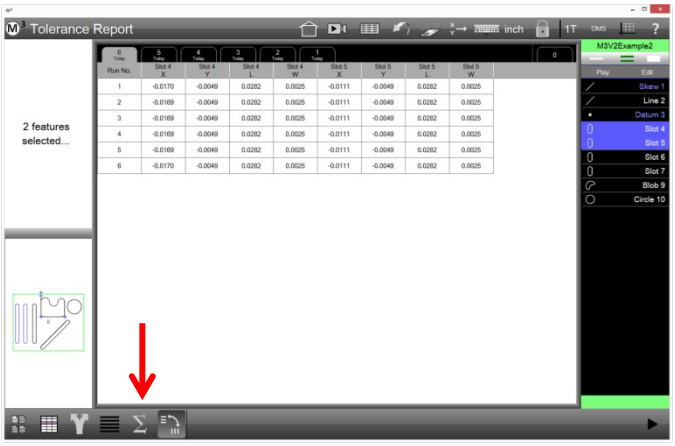
Three statistics are removed from the Results view

Pivot View Statistics

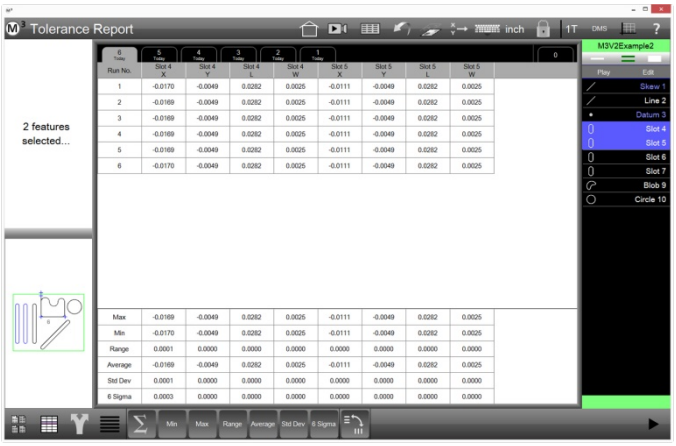
To display statistics in the Pivot view:

1 Press the Pivot view button and then select the desired feature(s).
Report toolbar > Pivot view
Feature List > Desired feature

2 Press the Stats button. A menu of statistics functions will be displayed in the Report toolbar. All statistics are enabled by default and displayed under the feature data by default.
Report toolbar > Stats button



Press the Stats button in the Pivot view…



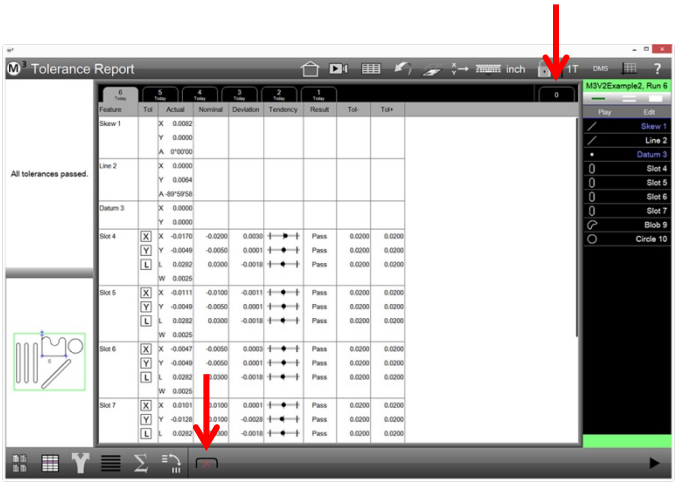
To add statistics to the Pivot view

3 Press the Statistics button again to remove all statistics from the Pivot view.
Results toolbar > Stats

Run Zero Tab

Part programs are often edited over the course of many runs resulting in changes in the results data through its history of runs. The run data is retained throughout the history of runs, but some data might not reflect the current state of the part program. To see the current part program state:

1 Press the Run Zero Tab
Results View window > Run Zero tab



Run Zero and Delete tabs

Delete Tab

Run tabs can be deleted from the Result view if desired. To delete run tab(s):

1 Select a single tab by pressing it, or multiple tabs using the multi--select tool above the Feature List.

2 Press the Delete tab in the Report toolbar to delete the selected run tab(s)
Report toolbar > Delete tab

Statistic Results Feature

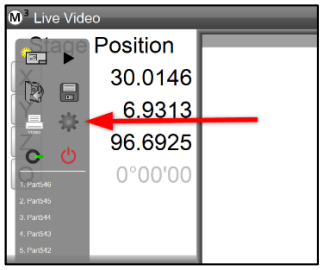
-Added new "Stat" type feature construction for quick analysis of statistical results inside a part program.

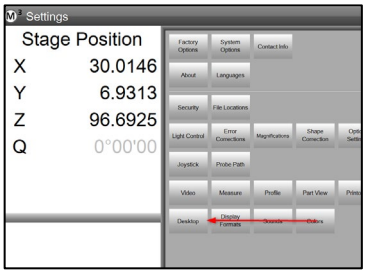
-Report the Min, Max, Range, Average, Standard Deviation, or Six Sigma result for a given population of feature measurements.

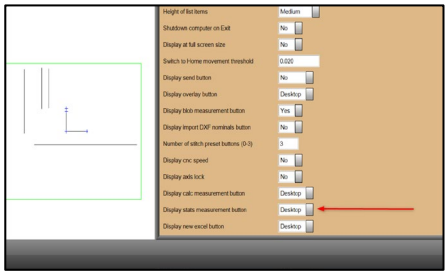
-Support for program playback of the "Stat" feature to enhance GR&R analysis and demonstration.

Setup:

IMPORTANT: The new Statistics Result functionality is OFF by default.
To enable the Stats feature navigate to the Desktop Settings Screen and set the Display Stats Measurement button flag to "Desktop" or to "Extra" depending on whether you would like the new button added to the desktop toolbar or to the Extra menu.

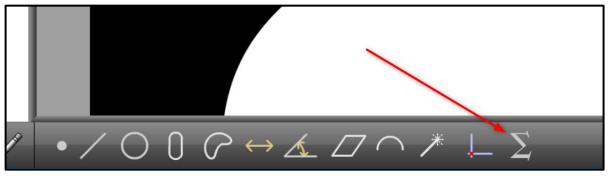






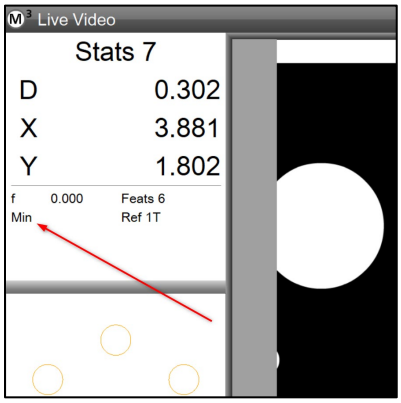
Operation:
The Stats feature is a simple way of comparing measurement results from a number of like features. As an example, if you measure six bore features in a bolt hole pattern, you can measure them and then construct a Stats feature from these parent circles and quickly determine the largest, smallest and average of the six diameters. This applies to all feature types and can be used on all coefficients. The results of the Stats feature can be toleranced and can be also included in a report or part program.

➔ When enabled, a new button will be added to the bottom toolbar(or Extra Menu) of the main M3 desktop. This button will be used to initiate a Stat Feature construction.

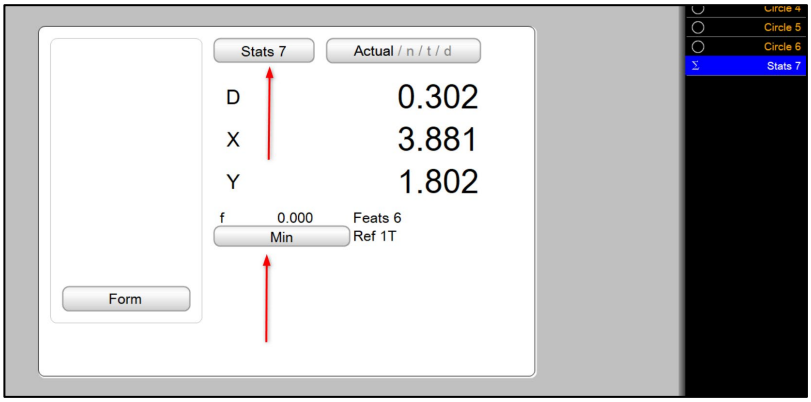


Sample Procedure Steps for the "Stat Feature" functionality:

1. Measure 4 circles of approximately the same size
2. Select the Stats Feature button from the Toolbar or Extra menu.
3. Select the 4 circles from the feature list and press the Done button.
4. The feature detail window(top left) will now display the results of the Stats feature construction, as seen below. The initial result will show "MIN", meaning the minimum size and position value of the entire selected feature population for the Stat construction.



5. Navigating to the full Stat feature detail by pressing in the small feature detail viewport. This screen, as seen below, will provide access to additional settings for the Stat feature result.



-Change the name of the Stat feature by clicking on the box containing the label "Stats X". Once selected, type the name of the desired label for this Stat feature result.

-Under the feature data, you will see a box showing the label "Min". You can cycle through the various Stat calculation types by pressing this button.

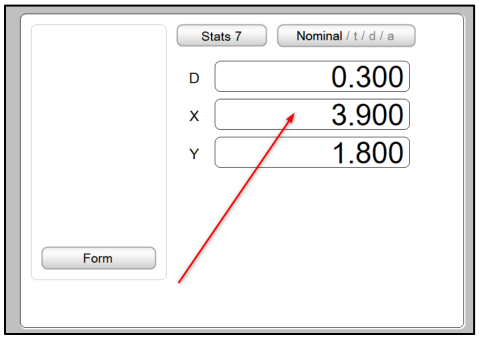
i. MIN = Minimum coefficient characteristic of the Stat population.
ii. MAX = Maximum coefficient characteristic of the Stat population.
iii. RANGE = Range of the coefficient characteristic of the Stat population.
iv. AVE = the average of the coefficient characteristic of the Stat population.
v. STD DEV = Standard Deviation of the coefficient characteristic of the Stat population.
vi. 6 Sigma = Six times the Standard Deviation of the coefficient characteristic of the Stat population.

6. The results of a Stats Feature can also be tolerance and reported.

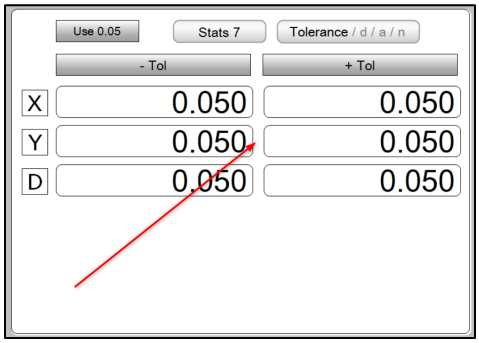
-Toggle the Stat feature result type to the desired Stat type.(Min/Max/Ave, etc)

-Toggle the "ACTUAL/n/t/r" button until it reads NOMINAL.

-Enter the Nominal Value for X, Y, and D in the corresponding fields.

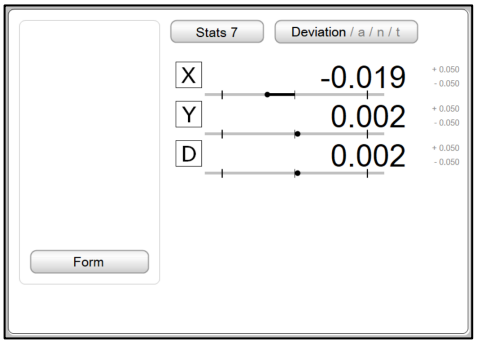


-Tolerance symbol buttons will also be displayed in the bottom toolbar. Click to add the X, Y, and D tolerance limit fields.



-Enter the +/- tolerance limit values into the corresponding fields.

-Toggle the view again using the view type toggle button until the Deviation view is displayed. You will see the results of your tolerance Stat feature.



-The Stat feature, including its tolerance will be displayed in the report view along with other standard feature measurements and constructions.

Section 10: Exporting Data

Feature measurement data can be exported to a file on your computer from the Report screen. Data can be exported in the following formats:

- .CSV
- .TSV
- .TXT
- .DXF
- Cloud CSV
- Cloud DXF

The format of the exported data file will conform to the current report template parameters described in Section 9: Reporting and the Results Screen.The exported data file will contain any modifications made to the report, including hidden or modified data cells.

The basic parameters and destination of exported data file are configured in the File Locations settings screen. Data can be exported to a folder, a fixed drive location or a network location. Data file exports can have custom file names and can be configured to append to an existing file, overwrite an existing file or be added as a new file with an auto-number applied.

Data export steps that are part of a saved part program can be edited to change the report format type

Specifying Export Settings

To specify the basic export settings for feature measurement data files:

1 Confirm that the report template settings are appropriate for your data export file. Report template settings are described in detail in Section 9: Reporting Data and the Results Screen.

2 Display the Export Settings screen.
System toolbar > System menu > Settings > Export button

3 Set the desired export parameters in the Export Settings screen. Settings are show below.

File Name
Configure the desired filename for the file exports.

Include Column Headers
Set this field to Yes to include column headers in the file export. The column header labels the columns by category.

Target File Type
Set this field to Append to append the exported file to an existing target file, set the field to Overwrite to overwrite the target file with the exported file and set the field to Auto Number to create a new file on each export that increments starting with the Auto Number name specified in the Auto Number File Name field.

Auto Number File Name
Set this field to the desired export Auto Number. The file number will automatically increment when the target file type is set to Auto Number.

Prompt for Settings on Each Export
The export options can be configured as global settings from within the Export Settings screen or can be configured on an export by export basis by enabling the Prompt for Settings on each Export from within the Export settings screen. To configure the software to prompt for Export settings at each file export command, configure the Prompt for Settings on each Export to Yes.

Append to Each Export
Enter text into this field to append a message to each export file.

Use Pallet locations in file Name
Set this field to Yes to include pallet locations in the export file data.

Exporting a Data File

To export a data file:

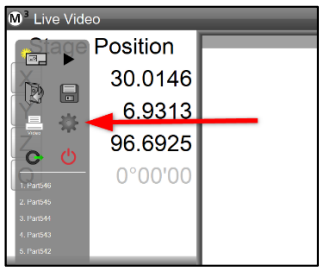
1 Press the Report button to display the Report screen in the center viewport.
System toolbar > Report button
2 Select the desired report format from the Report Format menu.
Report toolbar > Report Format button
3 Press the Export Data button to display the Export Data menu
Report toolbar > Export Data button
4 The exported data file will be written to the location designated in the File Locations Settings screen and a message will appear indicating a successful export of data.

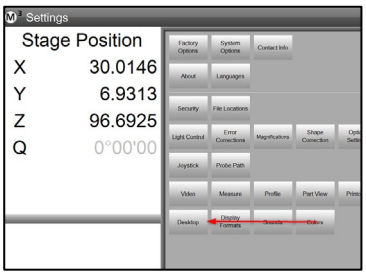
Direct Data Transfer to Excel

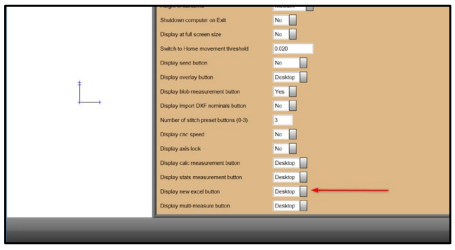
-Export measurement data directly to new or existing Excel spreadsheets.

-Export M3 specific report formats or populate existing quality reports using the custom export options.

Setup:
IMPORTANT: The new Custom Excel Transfer functionality is OFF by default.
To enable Custom Excel Transfer navigate to the Desktop Settings Screen and set the Display New Excel button flag to "Desktop". This will add the .XLXS option to your Data View export menu.







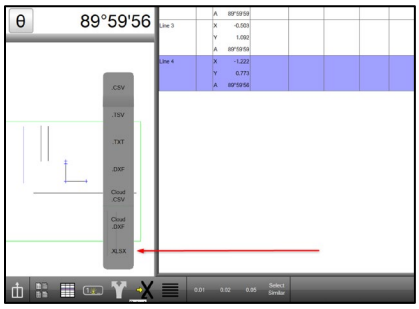
Operation:

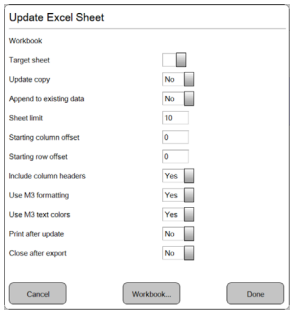
IMPORTANT: Direct data export to excel requires that a "licensed and activated" copy of the Microsoft Excel application be installed and closed at the time the Excel Data Transfer occurs. The M3 will not update open excel sheets. Use the "Close after export" flag(described on the following page) to ensure that sheets being updated programmatically are closed automatically, and ready for the next set of incoming data.

Direct to New Sheet Export:

When viewing measuring results in the M3 Data View the Export to Excel button(found in the bottom toolbar) can be used to export the current data view directly to an Excel spreadsheet. When executed the system will open a new instance of the Excel Application, open a new Workbook, and then populate a blank sheet with the M3 data.

Export to Existing/Custom Excel Sheet:
When viewing measuring results in the M3 Data View the Export Custom Excel button can be used(from the Export Menu) to send data to an existing Workbook and Worksheet. Custom export parameters can also be set using the flags in the Export Dialog, and described below.





Cancel Button – Cancels the current Export and return the operator to the Data view.
Done Button – Accepts the current Export Settings and executes the data export.
Workbook Button – Press to select an existing Excel Workbook file. Data will be sent to this file.

_____
Target Sheet – Once a Workbook has been selected from the Workbook button, select the target worksheet from the worksheet dropdown. This list will be populated with sheets detected within the selected workbook.
Update Copy – When set to Yes the target worksheet will be duplicated in the workbook and the new data will be sent to it.
Append to Existing Data – Set to Yes to append current data view contents to the end of the data within the current worksheet. Set to No to overwrite existing sheet contents.
Sheet Limit – Sets the maximum number of new worksheets that can be created within a workbook.
Starting Column Offset – Sets a cell column offset for the data entry starting point within the worksheet. The offset is specified from cell 1,1 within the target sheet, or from cell 1,1 relative to existing data, in the event the data is being appended.
Starting Row Offset – Sets a cell row offset for the data entry starting point within the worksheet. The offset is specified from cell 1,1 within the target sheet, or from cell 1,1 relative to existing data, in the event the data is being appended.
Include Column Headers – Set to Yes to include the data column header labels from the M3 template in the output excel sheet. Set to No to omit the header labels.
Use M3 Formatting – Set to Yes to enable M3 template emulation for the target worksheet. When set to yes the M3 will attempt to duplicate the template characteristics from the M3, in the target worksheet.
Use M3 text colors – Set to Yes to set text colors in the target sheet the same as M3. This option is typically used do that failed tolerance results, in Red Text color, will be transmitted to the target worksheet.
Print after update – Set to Yes to execute a Printout immediately following the export. The printer settings will be used from the M3 main system setup.

Close after export – Set to Yes to close the target workbook and Excel application, following the export step.

Sample Procedure Steps for "Direct to New Excel Sheet" export:

1. Prepare measurement results either by measurement or by opening an existing program containing run results data.
2. Press the Data View button to access the M3 report data view.
3. If desired, use exiting template editing techniques and data filters to prepare the data for export to excel.
4. Press the Direct to New Sheet export button from the bottom toolbar.
5. An instance of Excel will open and the data will be transferred to a new worksheet within the workbook.

Sample Procedure Steps for "Export Custom Excel Sheet" export:

1. Prepare measurement results either by measurement or by opening an existing program containing run results data.
2. Press the Data View button to access the M3 report data view.
3. If desired, use exiting template editing techniques and data filters to prepare the data for export to excel.
4. Press the Export Menu button from the bottom toolbar.
5. Select the item ".XLXS" from the export menu. The custom Excel Export dialog will be displayed.
6. Press the Workbook button from the bottom of the dialog and browse for an existing Excel Workbook file.
7. Select the desired target worksheet from the Target worksheets dropdown menu.
8. Set the desired formatting parameters in the Custom Export dialog.
9. Press Done to complete the Excel Data transfer operation.

Section 11: Programming

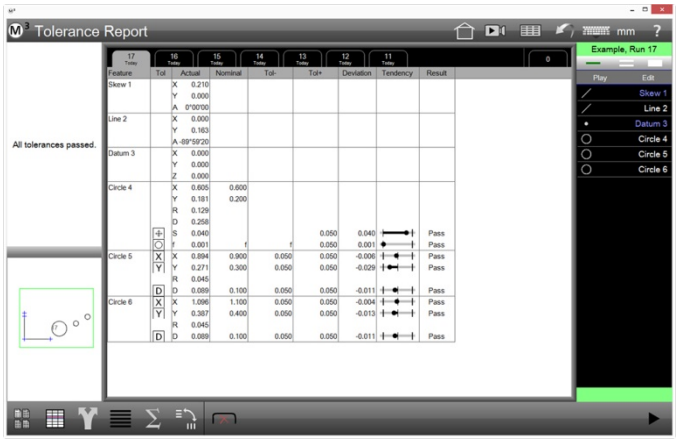
The M3 system records all of the measurement activities that occur during a part inspection session. These activities can be saved at the end of the session as a part program to be played back later. All measurement activities are recorded and include:

- Adjusting part lighting and camera magnification
- Probe path data configurations (CNC systems with touch probe option)
- Measuring, constructing and creating features
- Stitching all or a portion of large images to fit within the field of view
- Applying tolerances
- Preparing and printing reports
- Exporting data

Part program files are saved in the location:
C:\Users\Public\Public Documents\Metlogix\Parts

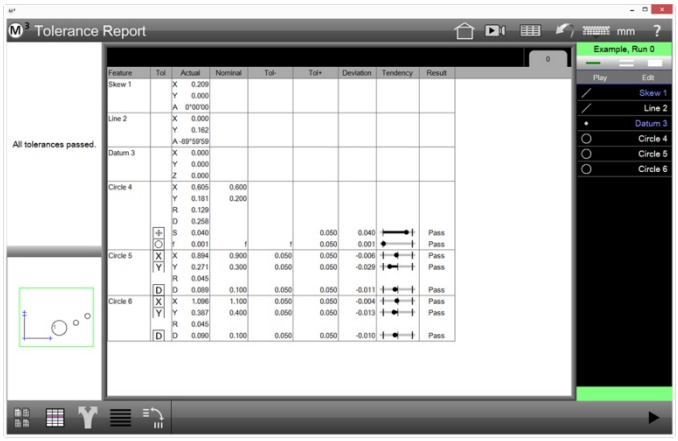
Part program file names are in the form:
filename.mlxpart2

When part program files are loaded, all existing features are cleared from the Feature List making room for new features from the part program file and preparing the M3 software to repeat the sequence of recorded measurement steps. The database of previous measurement results is also loaded and is shown in the center window as a tabbed results database in one of four report formats. When a part program file is loaded that was never played back to collect feature data, the measurement results are shown as an un-tabbed report view of the original data collected as the program file was being created.



Program previously played to measure features

Tabbed database of many run results shown



Program never played to measure features

Report shows only original programming measurements

Part Fixturing Modes

The M3 system supports three part fixturing modes:

- No part fixturing: Manual and CNC systems
- Temporary part fixturing: Manual and CNC systems
- Permanent part fixturing: CNC systems

Part fixturing is designated by editing the program properties for permanent fixturing or by responding to a user prompt when program playback is initiated for no fixturing or for temporary part fixturing. Part fixturing is discussed in detail later in this section.

No Part Fixturing

When no part fixturing is used, the user will be prompted to re-measure the features that define the part registration each time the part is measured. This mode is used primarily for quick inspections of one to a few parts in manual and CNC systems.

Temporary Part Fixturing

When temporary part fixturing is used, the user will be prompted to re-measure the features that define the part registration once, and then subsequent runs will be performed without new registration measurements. This mode is typically used to measure many parts using a manual system, or to measure many parts using the CNC system and a part fixture that will not be permanently attached to the machine.

NOTE

In manual systems, the user is guided to perform the programmed measurements by an arrow that appears in the live video window. In CNC systems, the reference frame measurements are performed automatically.

Permanent Part Fixturing

Permanent part fixturing is used only on CNC systems and must be specified by editing the program properties prior to saving a program. When permanent part fixturing is used, the part fixture is permanently attached to the system stage and the part registration is retained as part of the part program. Part registration measurements are performed automatically by the system each time the part program is loaded and requires no intervention by the user. This mode is typically used to measure a high volume of parts.

NOTE

The use of permanent fixturing relies on a repeatable machine zero based on encoder reference marks. The part program can only be run after the system has performed a home operation that sets the machine zero. Encoder and other changes that might affect the position of the system's machine zero will invalidate the part registration of previously recorded programs.

Creating a Part Program

Program steps are recorded automatically as measurements, tolerancing, reporting and other tasks are performed. When the program is complete, it can be saved or it can be run to gather feature data and then saved. Often programs simply consist of typical measurement session steps without any special adjustments. However, it sometimes becomes necessary to adjust default system behavior to accommodate the requirements of a specific application. This is more common in multisensor systems that include touch probes. Two programming adjustments are:

- Adjusting Probe Path Data
- Inserting Goto steps

Adjusting Probe Path Data

Probe path data settings define the default touch probe behavior for CNC systems and are described in *Configuring Probe Path Data* in Section 3: Probes.

The default probe path data configuration can be changed prior to probing a feature as part of a measuring/programming process. Some reasons for changing the default settings include:

- Changing the non-probing speed to test a new CNC program
- Removing a clearance distance because a Goto step will be used instead
- Changing the search distance for parts with highly variable features

NOTES

The probe path data configuration can also be changed in the edit mode discussed later in this section.

When probe path data are changed, the new probe path data will be used in subsequent touch probe moves until the probe path data are changed again.

To adjust probe path data settings:

1 Long-press the Goto button. The Probe Path Data dialog will be displayed.
Measure toolbar > Long-press Goto button

2 Change the desired probe path data settings.

NOTE

Probe path data settings are discussed in detail in Probe Path Data in Section 3: Probes.

3 Press done to return to the measuring/recording process.

Inserting Goto Steps

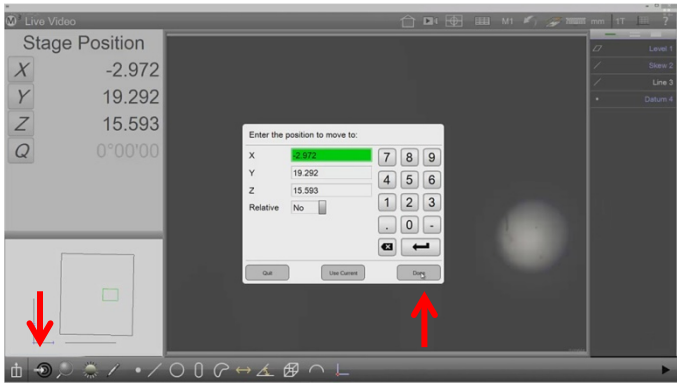
Goto steps are included to avoid probe collisions by moving the probe around or over obstacles between the last point probed and the next point to be probed. Goto moves include X, Y and Z components. Goto moves can be included in addition to clearance moves.

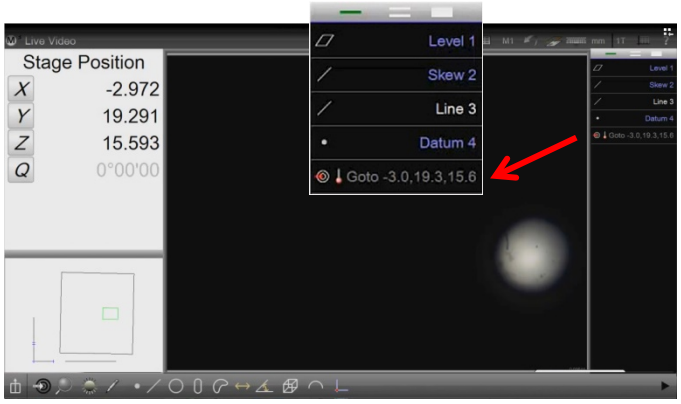
NOTE

Programs that include multiple Goto moves to avoid collisions should be tested at lower non-probing speeds. Adjust non-probing speeds in the Probe Path Data dialog discussed earlier.

To include a Goto step:

1. Move the touch probe to the next desired location.
2. Press the Goto button to display the Goto dialog.
   Measure toolbar > Long-press Goto button
3. Press Done to store the new Goto location and return to the measuring process.



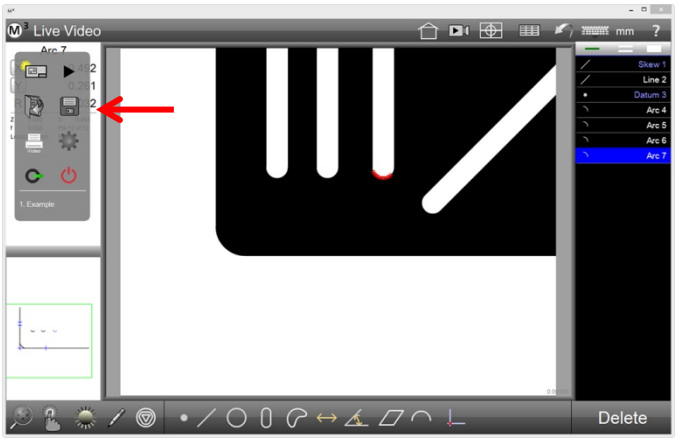


4. Enter more Goto steps in this manner if required to avoid obstacles to the next target point.

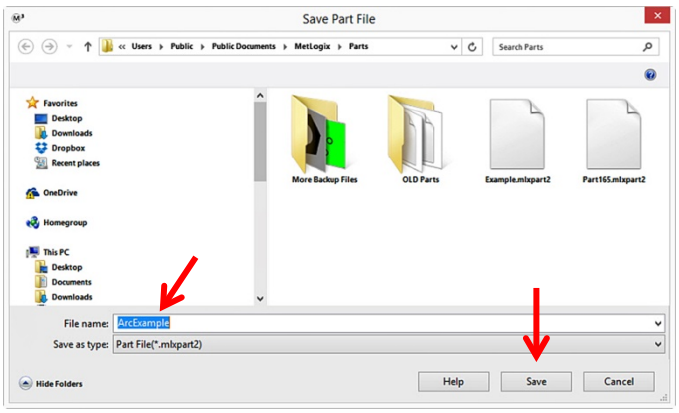
Saving a Part Program

Part programs are created automatically as measurements and other operations are performed. However, part programs must be saved at the conclusion of a measurement session in order to be recalled and used later.
To save a part program ***before running it*** to collect feature data under program control:

1. Press the Save button. The Save Part dialog will be displayed.
   System toolbar > M3 System menu > Save
2. Enter the desired part file name and press Save.



Press the Save button



Enter the new part file Name and press Save

To save a part program ***after running*** to collect feature data under program control or editing it to change its behavior:

1 Press Yes when prompted to save the program.

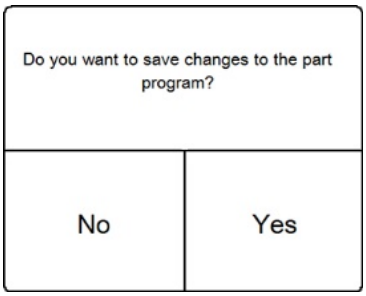
OR

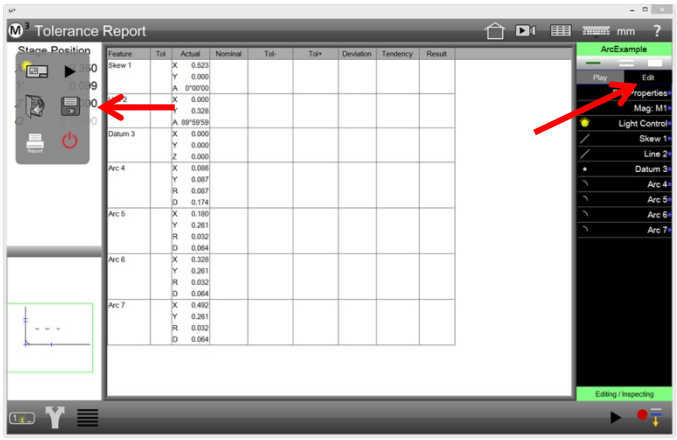
1 Press the Edit tab and then press the Save button. The Save Part dialog will be displayed.

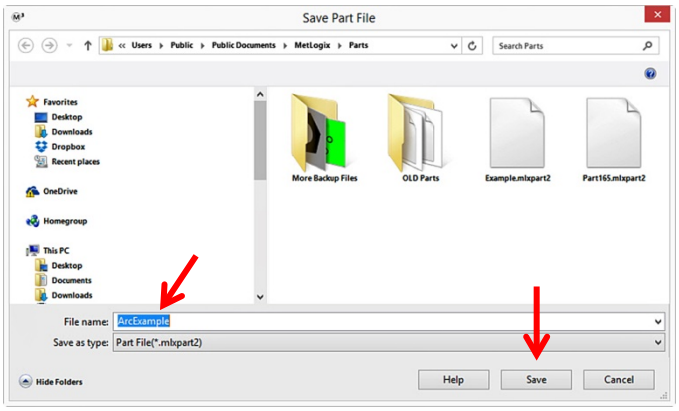
Feature list > Edit tab

System toolbar > M3 System menu > Save

2 Enter the desired part file name and press Save.







Loading a Program for Playback

To load a program for playback:

1 Press the Open File button to display the Open Part File dialog.

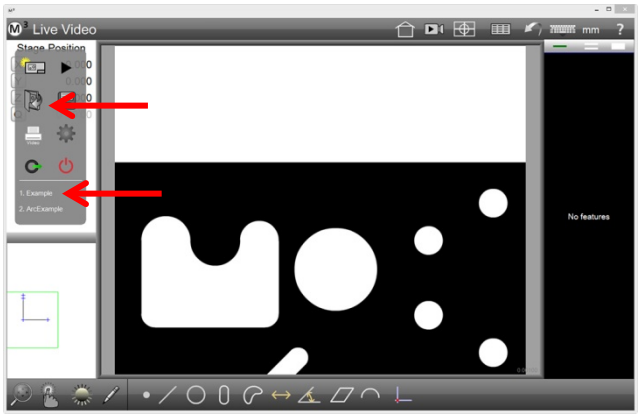
2 Select the desired part file and press Open to load the file.

System toolbar > System menu > Open File button

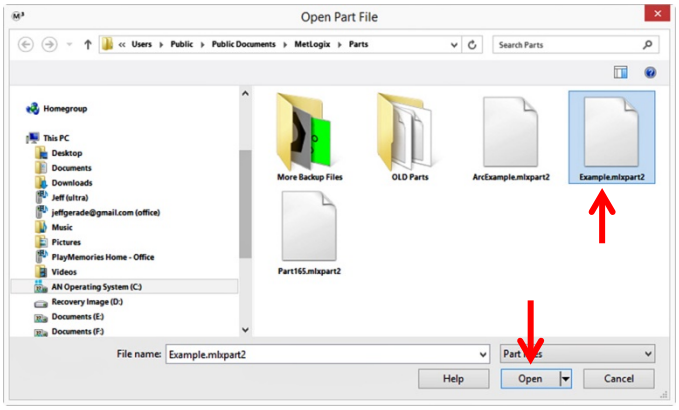
OR

1 Press the program name in the list of recent programs

System toolbar > System menu > Recent program name

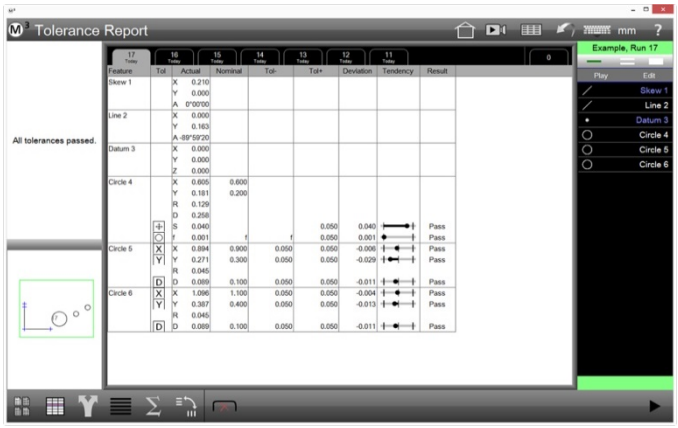


Press the Open File button or select a recent part file



Select the desired file and press Open

In CNC systems, if the part program file contains permanent fixturing, the program will be loaded and executed immediately as described on the next page. In manual or CNC systems, if the part program file contains no fixturing or temporary fixturing, the program will be loaded, the program name will be shown at the top of the Feature List, the program data will be displayed for the original program steps or the previous program runs and the system will wait for the user to press the Play button or the Edit tab.



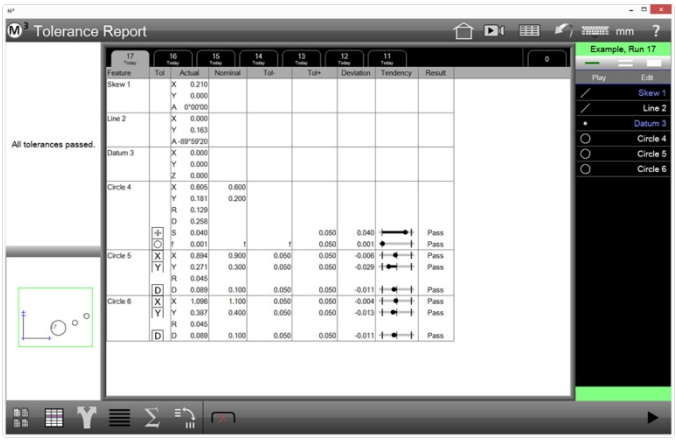
Playing a Program

Programs, in the form of measurement, tolerance and other software operations are played as a sequence of program steps. When a saved part is opened, the Feature List will become populated with features that were stored in the part file the last time the file was used and either:

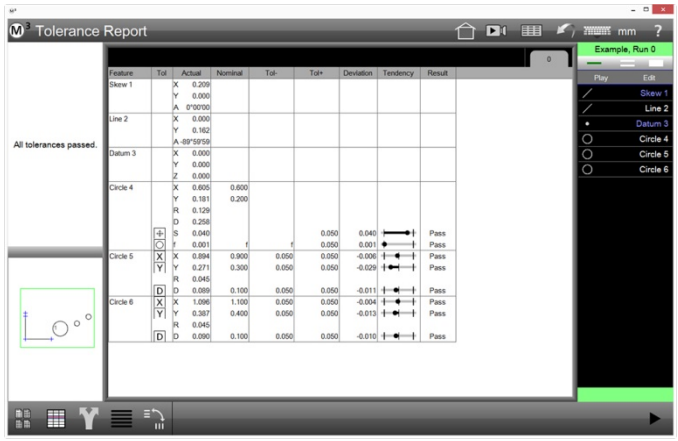
- A tabbed database of measurement results will be shown for a program that has been run to measure features

OR

- The original program feature data will be displayed for a program that was never run to measure features



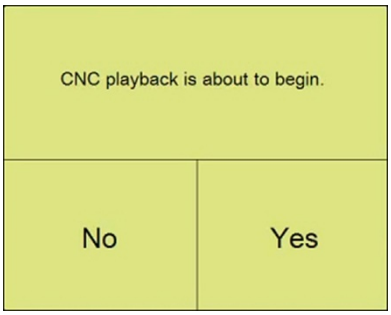
Program previously played to measure features
Tabbed database of many run results shown



Program never played to measure features
Report shows only original programming measurements

Playing Programs with Permanent Fixtures (CNC only)

When programs with permanent fixturing are loaded to a CNC system, the program will execute immediately after the user confirms that stage movements are safe by responding to the CNC prompt message. The program run will fill the Feature List with fresh measurement data.



NOTE
Permanent fixturing is designated for a part program in the edit mode and is discussed later in this section.

Playing Programs with a Temporary or no Fixture

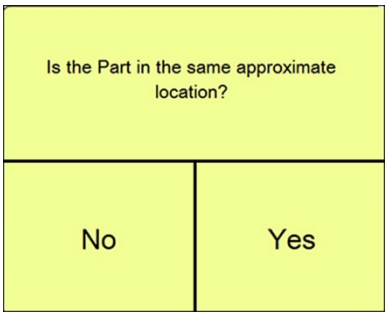
The choice between temporary fixturing and no fixturing is made by responding to a user prompt that appears when the user plays the loaded program. The prompt asks if the part is in the same location as the last run. To play a part with temporary or no fixture:

1 Press Yes if the fixture is temporary.

OR

2 Press No if there is no fixture
   - No fixture: The user must register each part before measuring
   - Temporary fixture: When the user runs the program, the system prompts the user to determine if the same part registration can be used again. If the part will be in the same location, the system will run the program without requiring a new part registration. If the part location is different, the system will require a new registration.

3 Follow the red arrow into green circle and part view guidance to perform measurements.



NOTE

When stage positions present more than one target feature in the field of view, all of the features in the field of view will be measured in the correct playback sequence.

Editing programs

All part programs can be edited unless the program author locked editing in the Allow Program Editing settings of the Program Properties step.

NOTE

Program Properties can be edited again to restore editing capabilities if required.

Changes made to an existing program will be reflected in subsequent program runs for the current program file session. If the file is reloaded, changes will only be reflected if the program file was saved. The user will be prompted upon exit to save if desired.

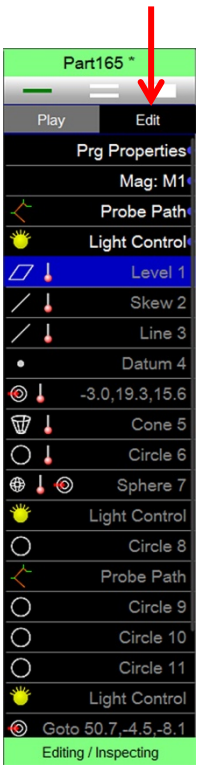
Program editing is performed using tools in the Feature (program steps) list and in the Edit toolbar: Editing functions include:

- Change program properties
- Change CNC probe path data
- Edit CNC Goto moves
- Change lighting
- Change Magnifications
- Change feature tolerances
- Play program
- Play from here
- Add program steps
- Enable or disable probe CNC approach and clearance moves
- Re-measure features
- Add CNC step and repeat measurements
- Add user messages or request data
- Delete program steps

Initiate Edit Mode

To edit a part program:

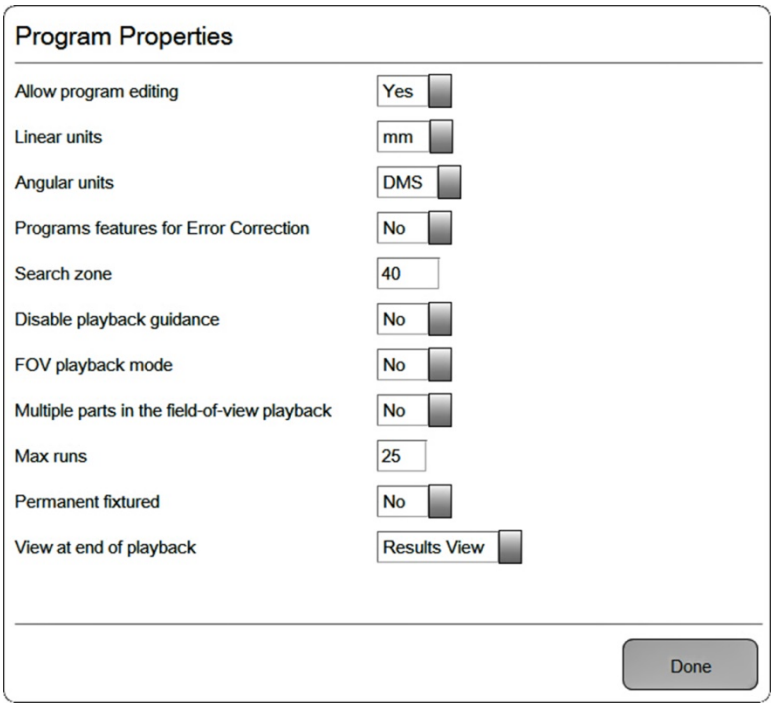
1. Load the program file as explained earlier if necessary.
2. Press the Edit tab at the top of the Feature List to display the detailed view of program steps and the Program Edit toolbar.



Change Program Properties

To edit Program Properties:

1 Press the Program Properties program step to display the Program Properties dialog. Press drop-down boxes and make selections to satisfy the needs of your application:



- **Allow program editing**: No to lock out program edits
- **Linear units:** Default linear units
- **Angular units:** Default angular units
- **Program features for error correction:** Restricted
- **Search zone:** Sets the default VED edge finding zone, in pixels, for measurements in program playback. Applies to all VED measurements.
- **Disable playback guidance:** Yes to disable graphic guidance for manual users
- **FOV playback mode:** Yes for field of view systems (video only, no encoders)
- **Multiple parts in FOV playback:** Yes to measure multiple identical parts in the FOV
- **Max runs:** Limit of runs in the runs database (push-down stack)
- **Permanent fixture:** Yes to designate permanent fixturing for CNC system
- **View at end of playback:** Replace default (Home) view of tabbed runs results database with different view upon program completion

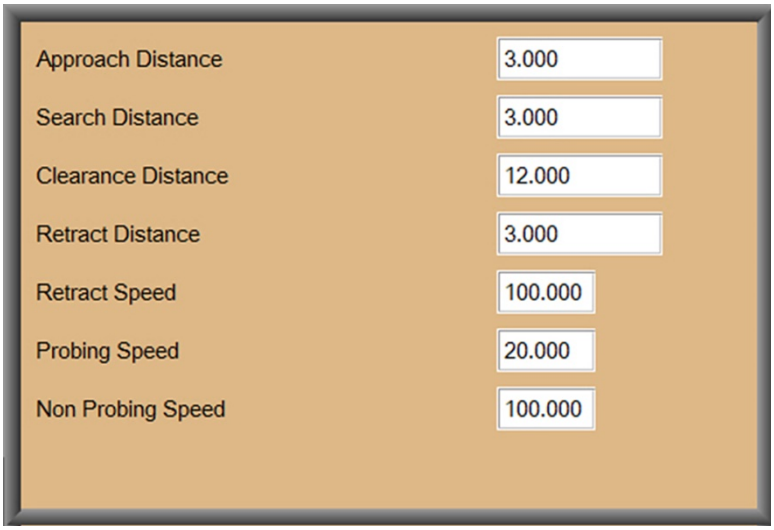
2 Press done to return to the Feature (program steps) list.

Change CNC Probe Path Data

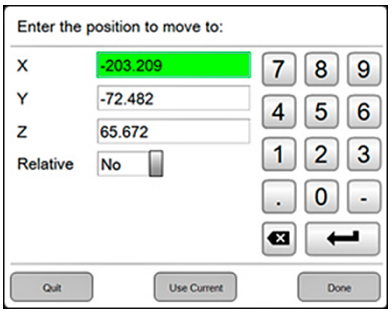
To edit probe path data for touch probe enabled systems:

1 Long-press the Probe Path Data step to display the Probe Path Data dialog.

2 Refer to *Probe Path Data* in Section 3: Probes for configuration information.



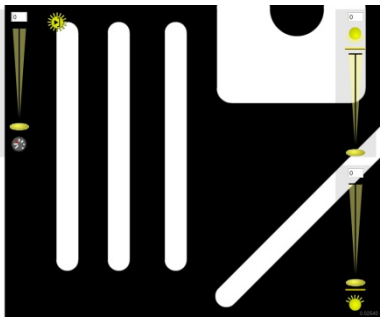
Edit Goto Moves



To edit a Goto move:

1 Long-press the Goto Move step to display the Goto dialog.
   - **X,Y,Z:** Change position
   - **Relative:** No for absolute position
   - **Use Current:** Press to zero at current position
   - **Quit:** Press to cancel edit
2 Press done to return to the Feature (program steps) list.

Change Lighting



To change light settings:

1 Long-press the lighting step to display the lighting adjustments.
2 Press and drag the lighting sliders to adjust the lighting.
3 Press in the screen to close the light adjustment function.

Change Magnification

To change the magnification:

1 Long-press the Mag step to display the magnification menu.
2 Select the desired magnification.
3 Press the magnification menu to close it.

Change Feature Tolerances

To change a feature tolerance:

1 Select the feature step.
2 Press in the small Feature Detail in the top-left viewport to display the Feature Detail screen.
3 Apply or change tolerance parameters as described in Section 8: Tolerancing.
4 Press Done twice to return to the Feature (program steps) list.

Play Program

To play the program:

1 Press the Play button.

Edit toolbar > Play button or tab

OR

System toolbar > M3 System menu > Play button

OR

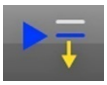
Feature list > Play tab

Play from Here

Program playback can begin at a specified program step. When playing from a selected step, the software will determine the requirements for generating feature context. If context dependent features have already been measured, the system will advance immediately to the selected step. If it is determined that additional context information is required, the operator will be prompted to measure the feature or features required to do so.

To play from a selected step:

1 Press the Play from Here button
Edit toolbar > Play from Here button

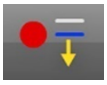


As mentioned earlier, context dependent steps will be executed automatically or prompted for manually and then playback will begin from the selected step.

Add Program Steps

New program steps can be added to existing programs. To add a program step:

1 Select the program step where you would like your new program operations to be added. These new program operations will be inserted after the selected program step. If no program step is selected after entering the Edit mode, program steps are added to the end of the program.
2 Press the Record Into button. A blue Edit Pin will be positioned at the point of new step insertion.
Edit toolbar > Record Into button



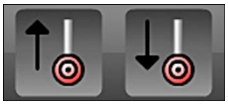
NOTE
When editing a program that has already been played back, the context generation will occur automatically, filling in context dependent program steps automatically from the previously acquired program data. For programs that have not yet been played back, the software will prompt the operator to manually measure all context dependent features prior to completing the context generation.

3 Once the Record Into edit command has been initiated, the user can measure or construct new features or insert export or print command steps according to the standard software procedures discussed earlier.
4 When the step or steps have been added, press the Edit tab to leave the Record Into mode. The added program steps will be shown in the current program edit list.
5 Press the save button to save the program with the new steps included.
System toolbar > M3 System menu > Save button

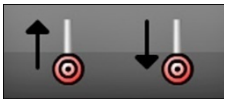
Enable or Disable CNC Clearance Moves

Clearance moves are made before and after probe contact with the target surface and are described in Probe Path Data in Section 3: Probes. Clearance moves are enabled by default but can be disabled together or independently when they are not adequate to prevent probe collisions during CNC moves in and around complex feature geometry and must be replaced with Goto moves.
To enable or disable clearance moves:

Press the (retreat ↑) Clearance or (approach ↓) Clearance button to toggle between enabled and disabled.



Enabled



Disabled

Re-measure Feature Steps

Program feature steps can be re-measured in the Edit mode. To re-measure a step:

1 Select the program step of the feature that you would like to re-measure.

2 Press the Re-measure Feature edit button.

Edit toolbar > Re-measure Feature button

3 You will be prompted to re-measure the selected feature. Re-measure the feature and then press Done when finished. The fresh feature data will replace the existing data within the program.

NOTE

The Re-measure edit command is commonly used to record new environment variables along with the feature measurement for a given step within the program. The new feature program step will contain all program variable information that exists at the time that the Re-measure command was executed. Information such as point spacing and light levels will be changed to reflect the context of the new measurement.

4 Press Save to save the revised program steps to the currently loaded program.

System toolbar > System menu > Save button

Add CNC Step and Repeat steps

The addition of CNC step and repeat steps greatly simplifies the measurement of features that appear in an organized matrix on the surface of a part, such as a matrix of drilled holes in rows and columns. The step and repeat function duplicates one or more feature program steps (excluding datum steps) in a matrix that is defined by the user. To create a matrix of duplicate program steps:

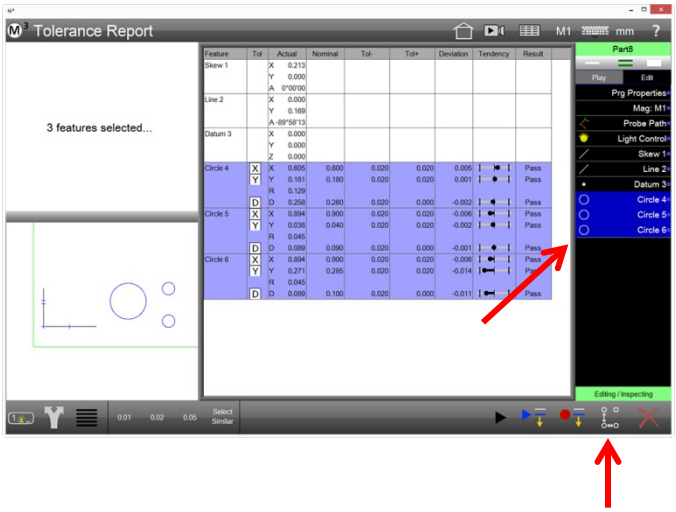
1 Select the desired program feature steps in the Feature (program steps) list. Use a multi-select tool at the top of the feature list if necessary.

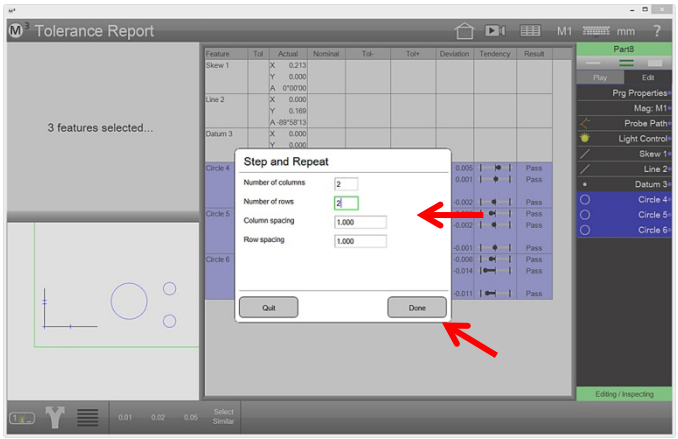
2 Press the Step and Repeat button to display the Step and Repeat dialogue.

Edit toolbar > Step and Repeat button

3 Enter the desired:

- Number of columns
- Number of rows
- Column spacing in the current unit of measure
- Row spacing in the current unit of measure

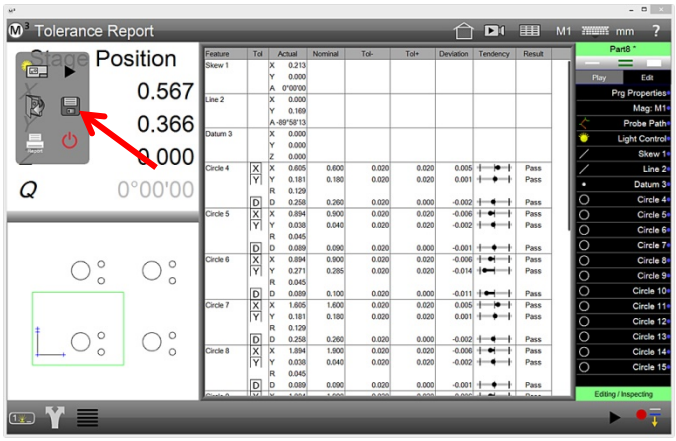


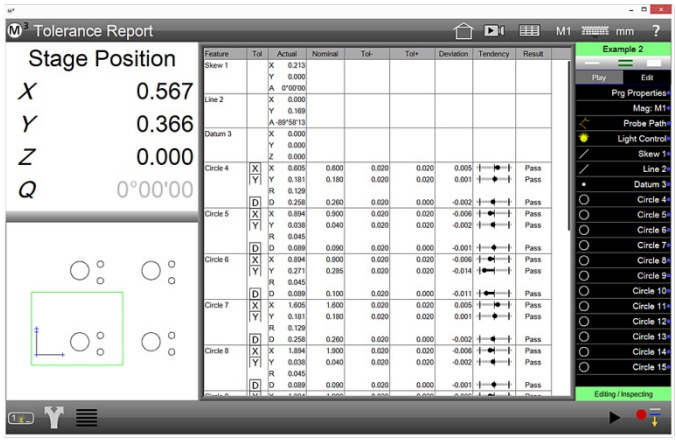


4 Press Done. The new program steps will be added to the feature list and shown in the Part View viewport.

5 Save the part.

System toolbar > M3 System menu > Save





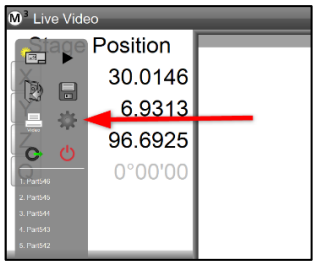
Polar Mode Step and Repeat

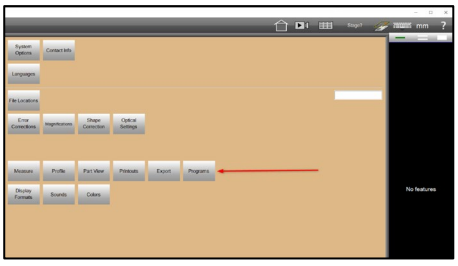
-Simplify creation of part programs containing large quantities of features by duplicating program step(s) automatically.

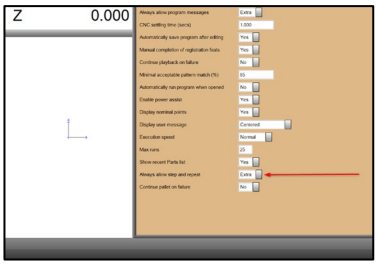
-Feature step and repeat functionality has been enhanced to support polar coordinate positions and offsets.

Setup:

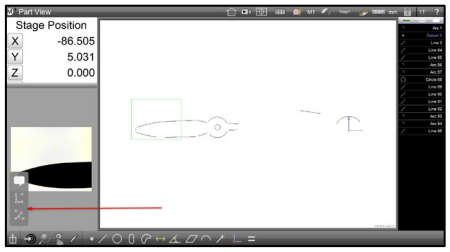
IMPORTANT: Cartesian and Polar step and repeat can now be used in both program creation mode(interactively) and in program edit mode. To enable the use of step and repeat mode while creating a part program navigate to the Program Settings screen and set the "Always allow step and repeat" flag to Extra. The Cartesian and Polar step and repeat buttons will be added to the Extra menu in the main M3 toolbar.

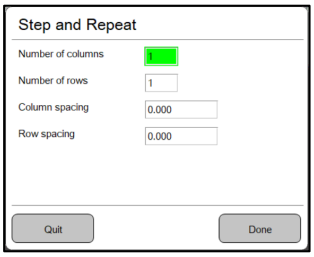


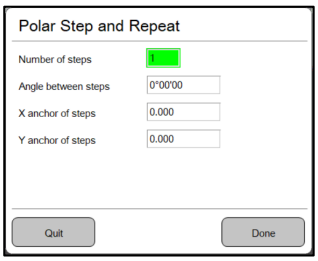




Operation:
The Step and Repeat function is used to duplicate measurements and constructions in your M3 part program, increasing programming speed for inspection tasks with large numbers of features.
The operator would first execute the source measurement(s) to be duplicated, select them from the feature list, and then press one of the step and repeat buttons to open the Step and Repeat Dialog. Descriptions of the dialog fields can be found on the next page.







Cartesian Step and Repeat:

Number of Columns – Designates the number of columns to be created in the step and repeat grid.

Number of Rows - Designates the number of rows to be created in the step and repeat grid.

Column Spacing – Designates the distance to be used between columns in the step and repeat grid.

Row Spacing - Designates the distance to be used between rows in the step and repeat grid.

Polar Step and Repeat:

Number of Steps – Designates the number of instances of the selected features to be created in the polar array.

Angle Between Steps – Designates the angle to be created between instances of the selected features in the polar array.

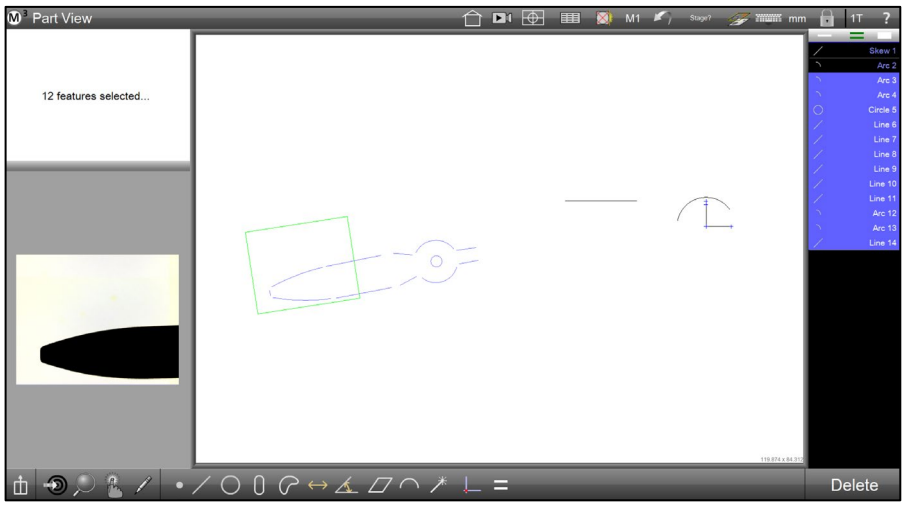
X Anchor of Steps – Designates the X position to be used for the origin of the polar array.

Y Anchor of Steps - Designates the Y position to be used for the origin of the polar array.

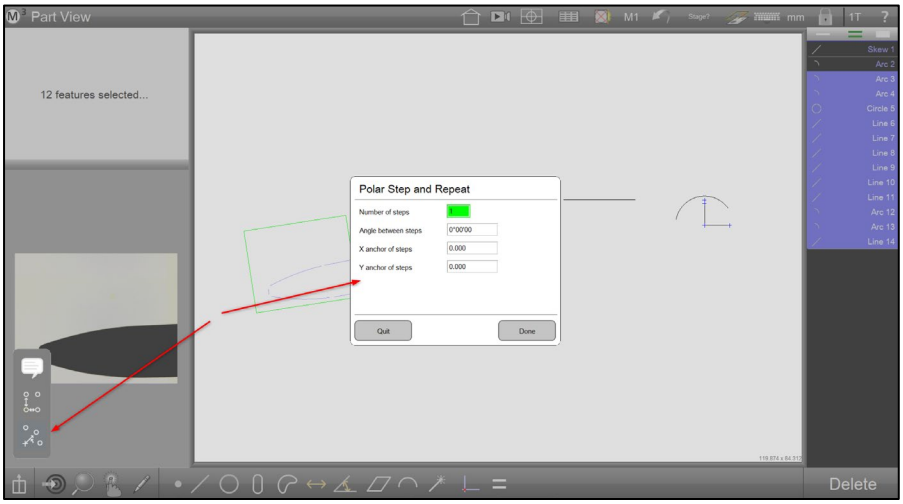
Sample Procedure Steps for using the "Polar Step and Repeat" functionality:

1. Author an M3 part program up to the point where the features to be used with Step and Repeat have just been measured.
2. Select the features from the feature list that you would like to duplicate for your polar array.

IMPORTANT: When using Step and Repeat in interactive mode, you can only choose steps that lead up to, and include, the most recently measured(or constructed) feature in the feature list. If you attempt to use a feature measured prior to the most recent an error message will be displayed.

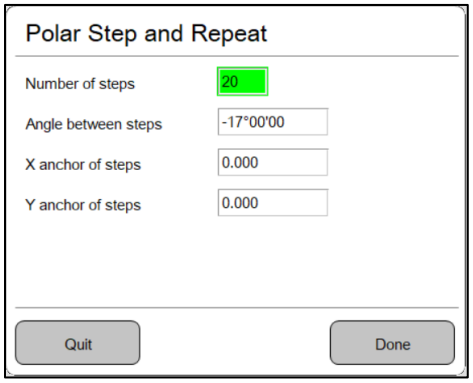


3. Press the Polar Step and Repeat button, located in the Extra menu in the bottom toolbar. The Polar Step and Repeat Dialog will be displayed.

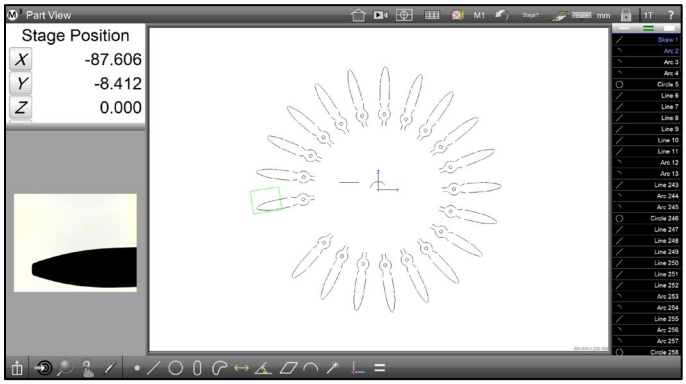


4. Enter the desired parameters into the (4) Polar Step and Repeat fields. For our example we will create a polar array with 20 instances of feature duplication, centered around the X/Y zero position, and at 17 degree increments.

   Note: Set the Angle Between Steps to a negative(-) number to step clockwise and a positive number(+) to step counter-clockwise.



5. Press the Done button to complete the Polar Step and Repeat. See resulting image below.



Adding a User Message or Request Data

User messages and images can provide useful information or instructions during program playback. Data provided by the user can enhance information provided on reports. The user message function can be included in programs during program creation or program editing for:

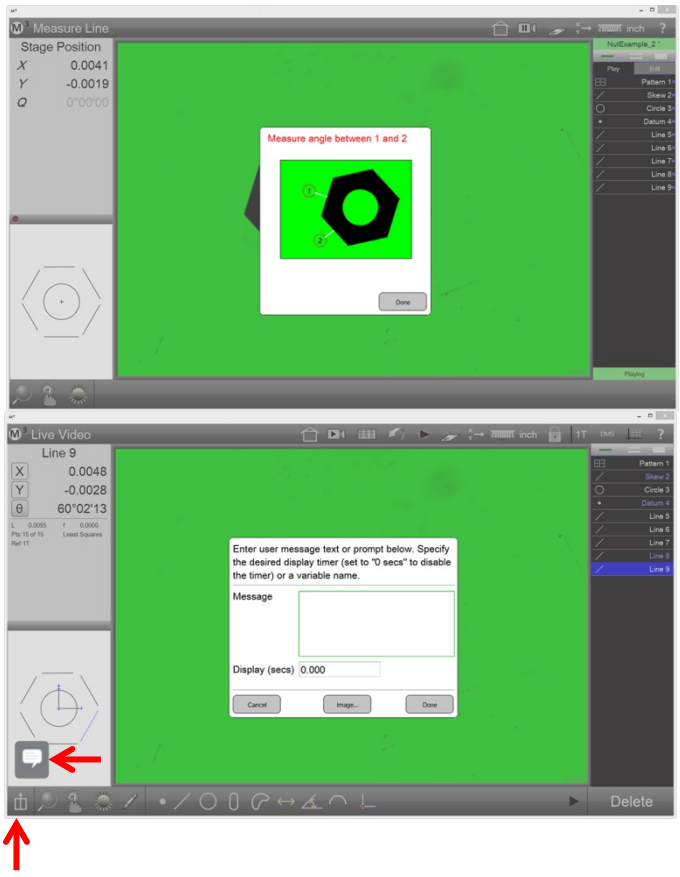
Text message during program playback

Adding User Messages and Images

To include user messages and images in a part program:

1 During program creation: If your system is configured to include the Ask button in the Extra menu, press the Extra menu button and then press the Ask button to display the User Message dialog.

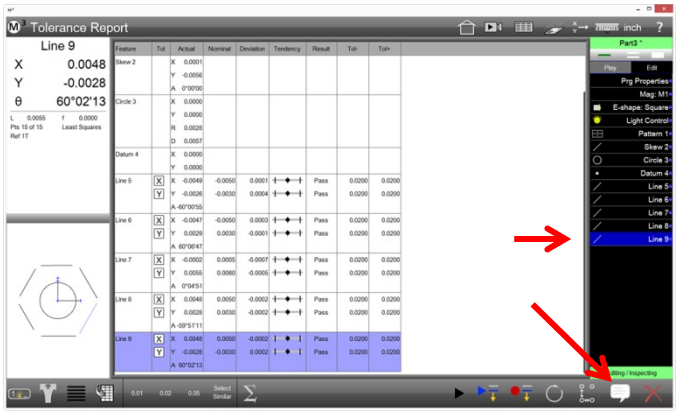
Measure toolbar > Extra Menu button > Ask button

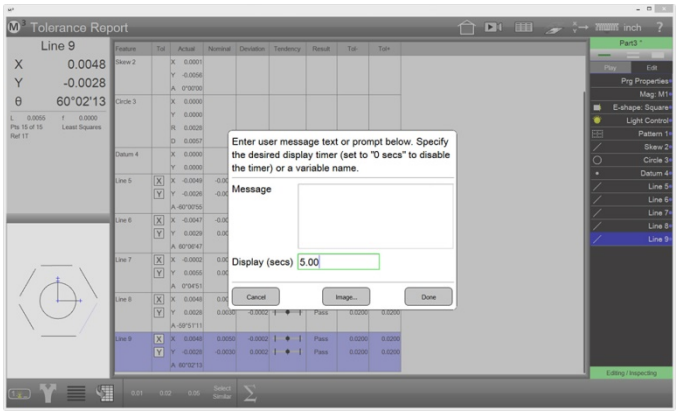


OR

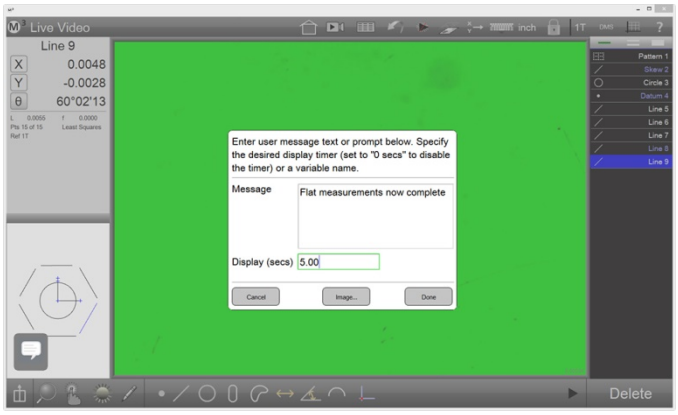
1 During program editing: Select the program step before the desired insertion point and then press the Ask button to display the User Message dialog.

Program Edit toolbar > Ask button

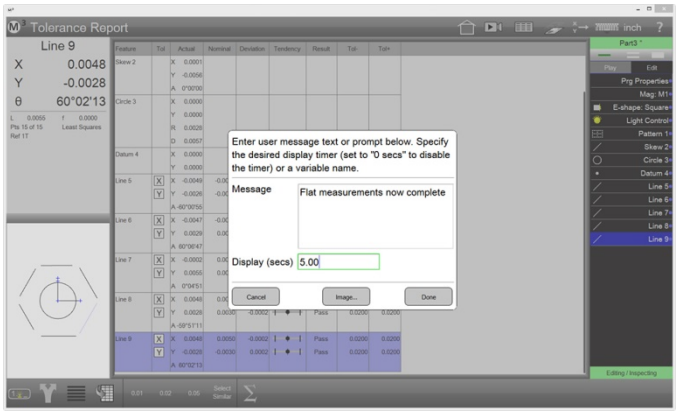




2 Enter a message into the Message field and then enter a message display period Into the Display field.



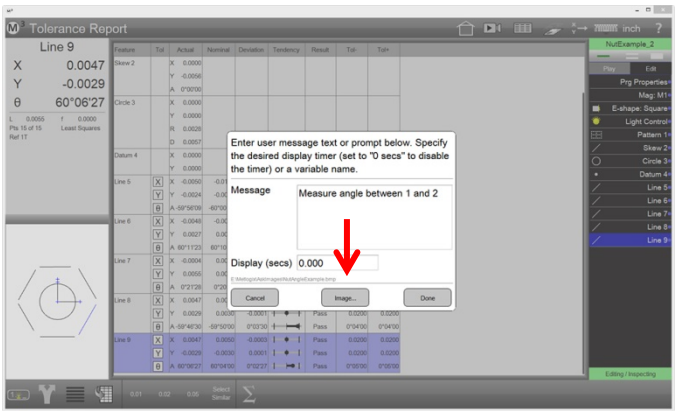
Message entered while programming

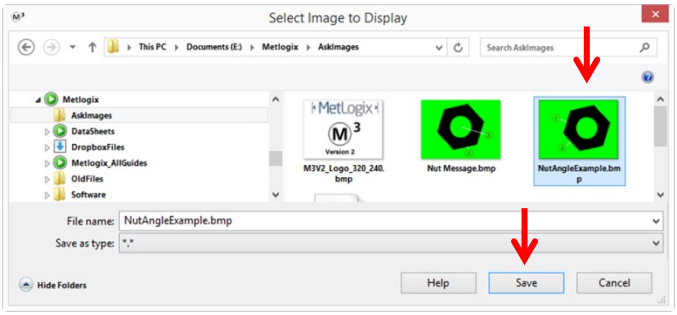


Message entered while editing

3 Press the Image button to include an image with the message. A browse dialog will be displayed. ***If no message is desired proceed to step 5.***

4 Navigate to and select the desired bitmap (.bmp) image and press Save. The image should be no more than approximately 320 by 240 pixels.





5 Press the Done button to insert the message. The message will be displayed at the appropriate point when the program runs.

Requesting User Data (Variables)

Variable data provided by the user at run time is shown in report template headers and reports. To include requests for user data in a part program:

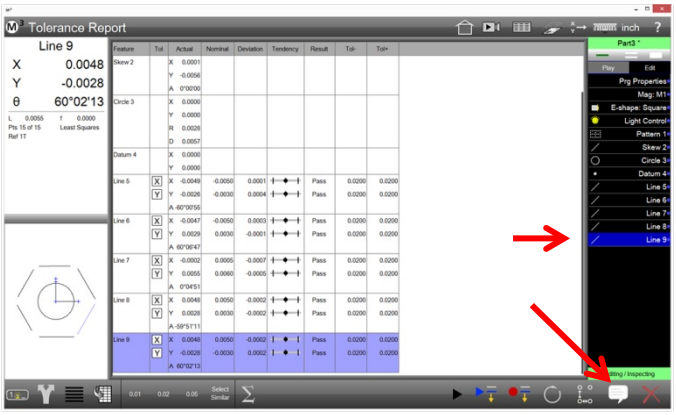
1 During program creation: If your system is configured to include the Ask button in the Extra menu, press the Extra menu button and then press the Ask button to add a request for data at the desired point in the program execution.

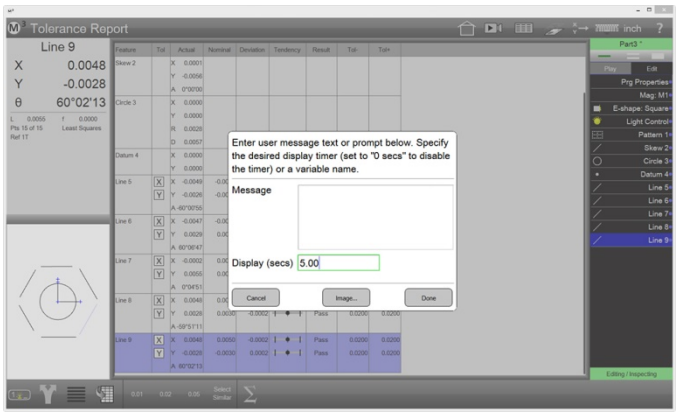
Measure toolbar > Extra Menu button > Ask button

OR

1 During program editing: Select the program step before the desired insertion point and then press the Ask button.

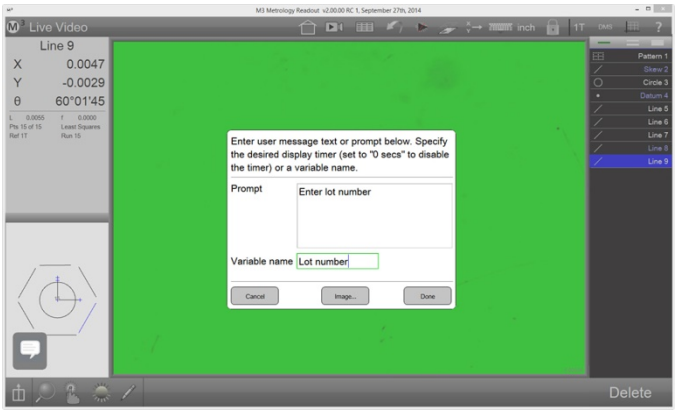
Program Edit toolbar > Ask button



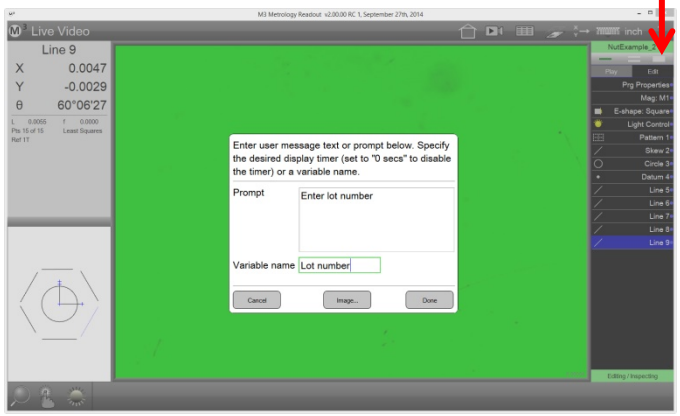


2 Enter the data request message into the Message field.

3 Enter the variable name that will be assigned to the data on template headers and reports into the Display field. The field title will change from Display to Variable Name. The variable name should reflect the reason for the data. For example, if the data request message asks the user to "Enter the lot number", then the variable name should be "Lot number".



Data request entered while programming



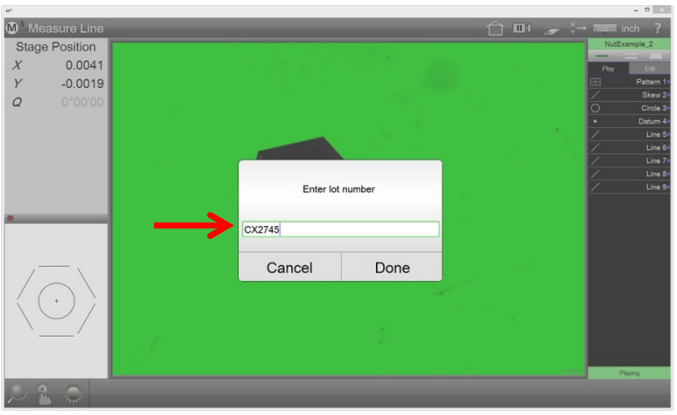
Data request entered while editing

NOTES

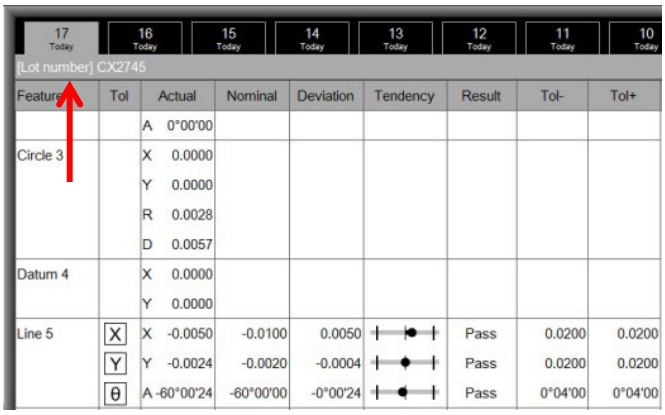
The leading character of the variable name must be an alpha character.

Do not include an image when creating a request for data.

The system will request user data at the appropriate point when the program runs. The data variable name and user-provided data will be displayed on template headers and on reports.



Data provided during run time



Variable name and Data shown on template header

Deleting Program Steps

To delete a program step:

1 Select the program step that you would like to delete.

2 Press the Delete Program Step button. The selected program step will be deleted.

Edit toolbar > Delete Program Step button

3 The step is deleted from the current program playback session. Press the Save button to permanently apply the deletion to the part program.

System toolbar > System menu > Save button

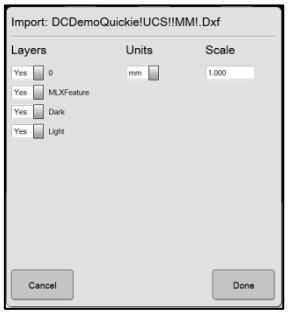
DXF Import and Auto Programming

The M3 system will convert DXF part drawing files into part programs using the DXF import and Auto Program module. To convert a DXF drawing file into a part program:

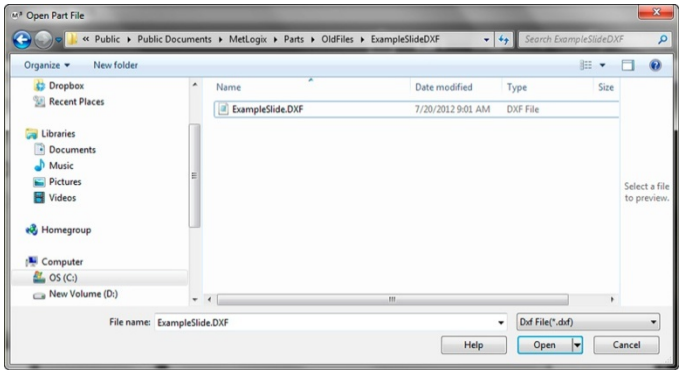
1 While the M3 program is in the interactive mode, press the Open Part button to display the Open Part dialog.

System toolbar > System menu > Open Part button

2 Select DXF from the File Type menu, select the desired file and press Open. The import DXF Dialog will be displayed.



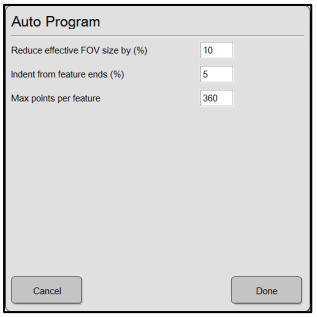
Choose DXF parameters



Import DXF part file

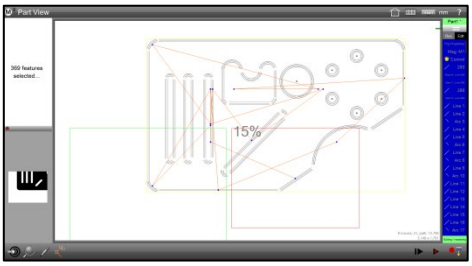
3 Choose the desired layers for import, the unit type for import, and specify the desired scale factor to be used for import and then press "Done" to continue. The DXF features will be added to the Feature List with DXF tags,

4 Trim (delete) features from the imported DXF file that you do not want to be part of the resulting auto-program.
5 Specify registration (datum) features to be used for the auto program registration by performing datum operations directly on the DXF features using reference frame operations described earlier in Section 7: Reference Frames. The specified registration features will be moved to the top of the Feature List.
6 Press the Play Button to display the Auto Program dialog and initiate the auto program process.
System toolbar > System menu > Play button
7 Specify the desired number of points per feature to be programmed, the desired FOV size to be used at each stage location, and the indent margin to be used for positioning point clouds from the DXF element feature ends, and then press Done to continue.



Specify auto program parameters

8 The auto program path will be generated by the system and program playback will be initiated. Probe the registration features specified on the DXF part image. Program playback will proceed once registration is complete.
9 Press the Single Step button to initiate single step playback.
Edit toolbar > Single Step button
10 In the single step mode, use the left and right step buttons, or keyboard arrow buttons to advance or rewind the program, one step at a time.
11 Perform program edit commands as needed to complete the playback of the auto programmed part.
12 Press the Save button to save changes when the program is complete to store changes made to the program.
System toolbar > System menu > Save button

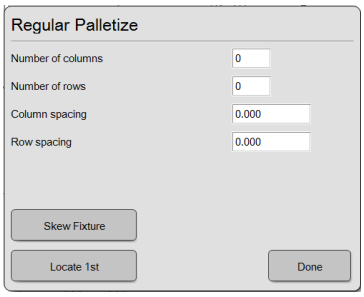


Auto program path generated

Pallet Program Creation

Quickly duplicate part programs for CNC playback within a palletized fixture. Specify the desired number of pallet rows and columns and their spacing, then create your pallet program in one click.
To create a pallet program:

1 Generate the part program to be used for pallet program duplication.
2 From the program edit mode, select the Program Properties step and press the Palletize button to access the Regular Palletize screen.
3 Specify the number of pallet columns and rows, and their spacing (in the currently selected units) to be used for the pallet.
4 Press the Skew Fixture button to assign a skew adjustment for the pallet fixture being used. The Fixture Skew will be associated with this pallet program until it is updated by a new Skew Fixture operation.



NOTE
The Skew Fixture operation initiates an interactive feature measurement session. Probe all of the features necessary to generate the skew alignment for the fixture, or explicitly probe a skew feature.

5 Press the Pallet Done button from the bottom toolbar once the skew has been generated.

6 After generating a Skew for your pallet program, press the Locate 1st button . The system will prompt the operator to probe the registration features of the first part to be used for the pallet program. This sets the origin part program for the pallet program.

IMPORTANT NOTE

Skew and Locate First for pallet programs must always be performed as a pair, and in that order. The skew and origin part for a pallet program can be updated by re-executing the above two steps, skew first, then locate first.

7 After establishing skew and origin part for the pallet, press play to begin pallet playback. The pallet pocket selection dialog will be displayed.Toggle the pocket locations to specify them as active or inactive for the subsequent program playback. Enabled pockets will be measured, disabled pockets will be skipped. Press Done to accept the selections.

The system will playback the part program at each enabled pallet location. At the end of playback, the pocket selection dialog will be displayed again indicating pockets that contain features with failed tolerances.

Palletizing and Support for Stitched Images

-Applications with pallet based fixturing requirements can now make use of stitched images.

-Part position and orientation can be quickly identified by using pattern recognition within the super-image source of the pallet program.

-Using this method greater inspection throughput can be realized by overall reductions in pallet loading times and improved playback reliability.

Setup:

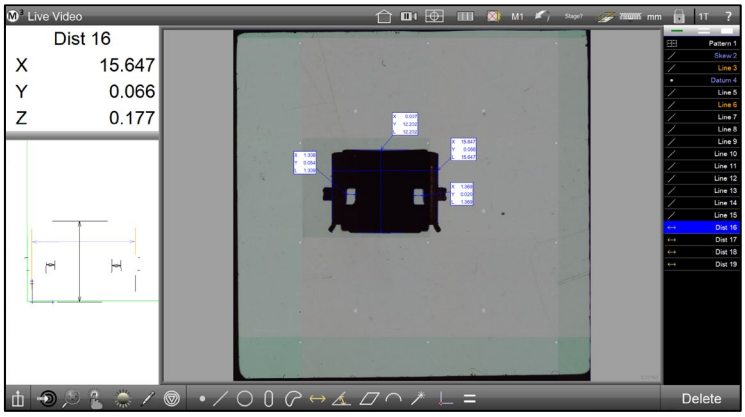
IMPORTANT: There are no additional software setup requirements to make use of stitched images within pallet programming. However, the measuring system must support image stitching, and pallet programming/permanent fixture modes.

IMPORTANT: The following sample procedure will palletize an existing M3 program that was authored using a stitched image of a "pocket" in a target fixture, and pattern recognition to identify the location of parts within the fixture. This sequence can also be used with M3 part programs using more traditional datuming and registration techniques.

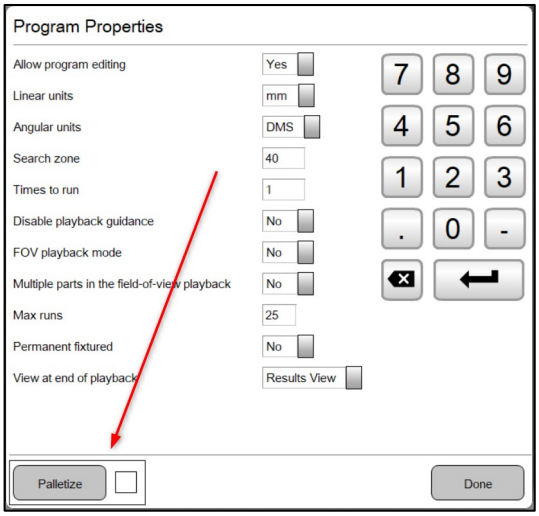
Sample Procedure Steps for "Palletizing a Program":

1. Author the sample part program using image stitching to create a super-image of the pallet pocket containing a part sample. Use pattern recognition to establish part program registration as would be typical in a Field of View(FOV) system application.

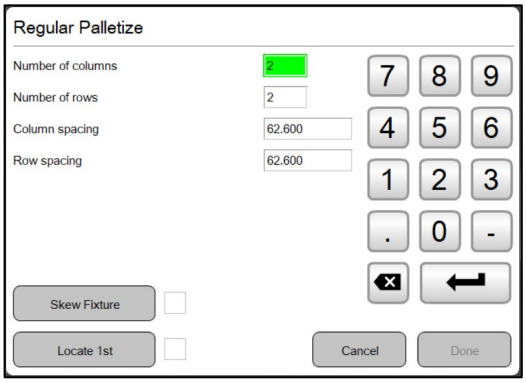
2. Complete the remainder of the program authoring process, including the application of tolerances, reporting requirements(export and print), etc.



3. Press the Play button in the M3 menu to convert your measurement routine to a part program. Play the program back once to test playback functionality.
4. Go into Edit mode and press the Program Properties Step to access the Program Properties dialog, as seen below. You should notice that the Pallet Status box in the lower left corner is empty, indicating the program is not currently palletized.



5. Press the Palletize button in the lower left corner to access the Pallet Setup dialog.
6. Enter the number of Rows and Columns for your pallet, as well as the spacing between each row and column into the 4 numeric entry fields, as seen below.



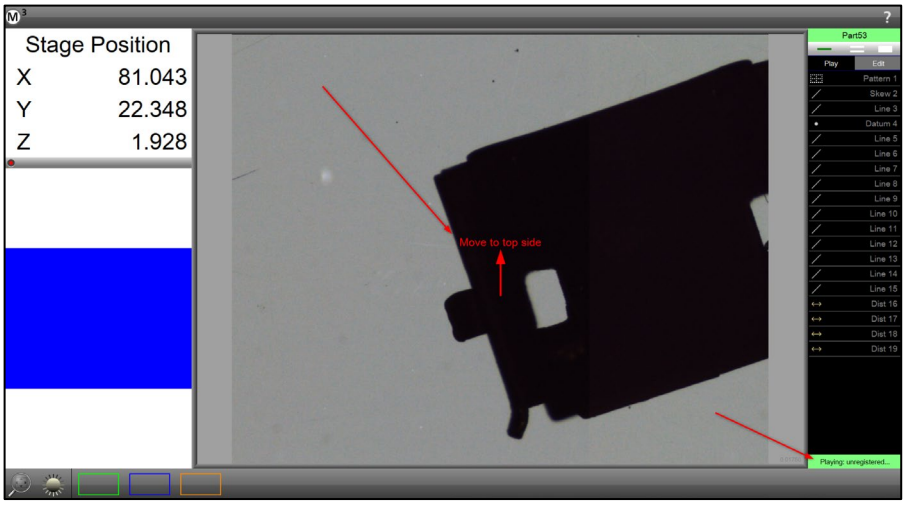
7. IMPORTANT: The Skew Fixture and Locate 1st buttons will be used to configure the skew orientation of the pallet fixture and identify the origin pocket for pallet program playback. Both of these steps are required to successfully palletize a program and must be performed in order, Skew Fixture first followed by Locate 1st. The small status boxes to the right of the buttons indicate when the Skew or Origin have been successfully set. The box will fill green when successfully established.

   It should also be noted that any changes to the row and column count, or spacing, will result in a de-registration of the above pallet parameters, indicated by the status boxes no longer being "green filled". Simply re-execute the Skew Pallet and Locate first to re-palletize the program using the new pallet row/column count or spacing.

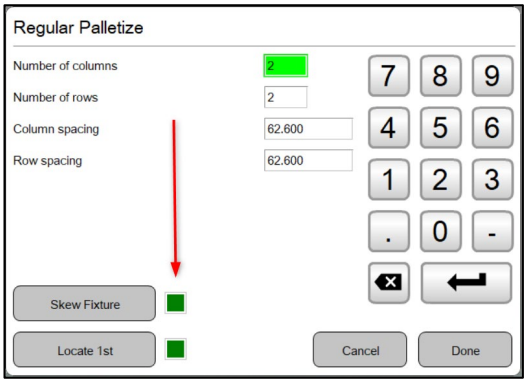
8. Press the Skew Fixture button. The M3 will return to video measuring mode. Skew the fixture by performing a skew measurement or construction on the fixture. Once a valid skew is detected by the M3 the Done Pallet button will appear in the bottom toolbar. Press the Done Pallet button to complete the process. Upon completion of the Skew Fixture routine the system will return to the Pallet Setup dialog.

9. Press the Locate First button. The system will switch to program playback mode for the part program you are palletizing. Execute the Registration program steps required for the part program.

   IMPORTANT: For programs that begin with an image stitch step, as is the case for our example, the program registration comes from the stitched image position and orientation itself. As such the Locate First operation described above will comprise of setting the rectangular bounds(top, right, bottom , and left) for the origin pallet pocket. This is accomplished using the same method as any other rectangular stitch operation, following the on-screen instruction text and arrow.



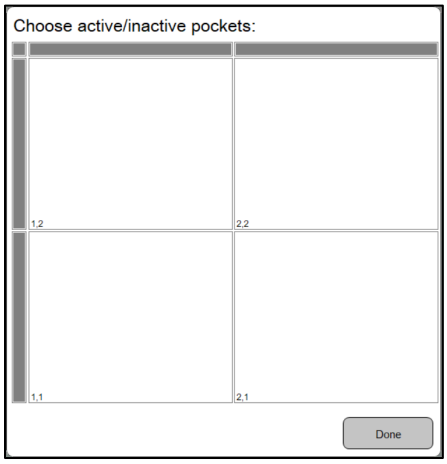
10. Upon registration complete the system will return to the Pallet Setup Dialog. You should see both status boxes filled green indicating successful pallet setup, as seen below.

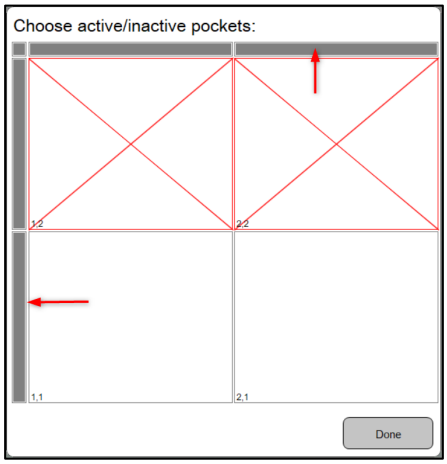


11. Press the Done button to close the Pallet Setup dialog and the Done again to close the Program Properties dialog. The program should now be successfully palletized.

Sample Procedure Steps for playing back a "Palletized Program":

1. Open a pallet part program and press play from the M3 menu, or if continuing from the procedure above, just press play. Playing back a palletized part program will always display a dialog similar to the one seen below.





2. The dialog will display a square pocket "button" for each pocket established in your pallet setup. Click any button to put a large red "X" in the button, as seen in the image to the right above. Use the grey row and column Select All buttons, located along the top and side of the pocket grid to enable/disable entire rows or columns of pockets. Pockets with a red "X" will be skipped in program playback. Press Done to proceed with pallet program playback.

Section 12: Field of View (FOV)

Field of view systems perform measurements on parts located entirely within a fixed video field of view. Feature dimensions and locations are determined by the pixel resolution of the video system. Measurement activities are automatically recorded as part programs in all M3 systems. Part programs can be played back manually or automatically without the need for part registration measurements performed by the user. Programs can measure single or multiple identical parts placed within the field of view.

M3 Measurement and Other Functions

Part measurement sessions can include measuring features, constructing features, applying tolerances, reporting results and other activities. The probes, user interface screens and other M3 functions described earlier in this guide serve as a reference for field of view system operation. Please familiarize yourself with M3 functions before proceeding. Discussions in the remainder of this section assume a basic working knowledge of the M3 system.

Functions unique to field of view systems

A few important functions are unique to field of view systems:

- Auto Play
- Program Finder
- Motion Monitoring

NOTE

Program finder can also be used in non-FOV systems, but not in conjunction with the Auto Play feature.

Auto Play

Field of view measurement sessions can be as simple as a quick diameter check, or can include a coordinate system, tolerances, markups, reports and other M3 functions. More complex measurement sessions usually begin with a datum pattern measurement of a part. The datum pattern registers the part and when the completed program is played back using Auto Play, the pattern is recognized and part registration operations by the user are unnecessary.

Automatic program playback is only provided by the Auto Play function when Auto Play recognizes the part pattern and the program is contained in the Program Finder list.

Program Finder

When Auto Play is not enabled, the user performs a pattern measurement on a part and when the part is contained in the Program Finder list, the program is loaded. The user can then simply run the program. When Auto Play is enabled, the program is played automatically. Only programs contained in the Program Finder list can be loaded or played automatically. Program Finder lists can contain many programs for widely varying parts. Programs can be added to or deleted from the Program Finder list at the user's discretion.

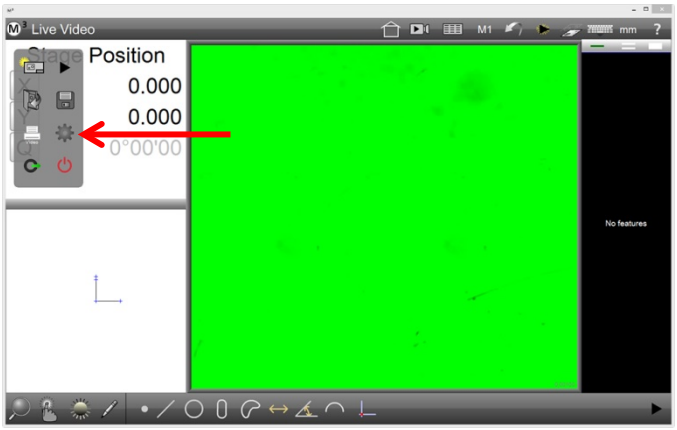
Auto Play Use of the Program Finder

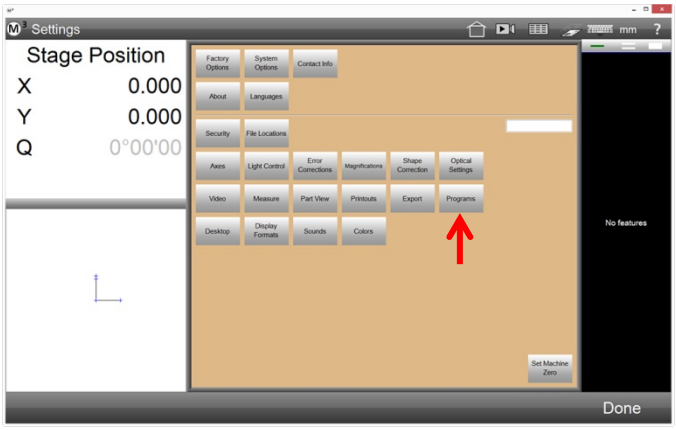
When a part is placed in the field of view, if its part program was registered with the pattern, Auto Play recognizes the pattern and looks for the corresponding part file in the Program Finder list. If the program is in the list, the part file and the associated runs database are loaded. The program is then played automatically.

Verifying Auto Play Use of Program Finder

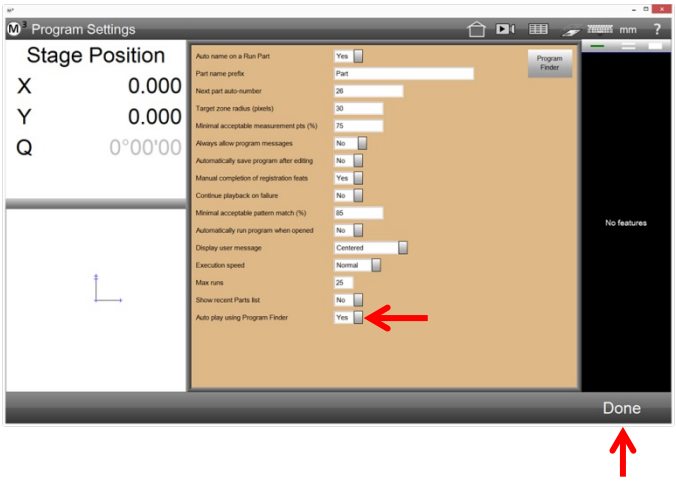
Auto Play must use the Program Finder list if Auto Play will be used to load or play programs. To confirm the use of Program Finder:

1 Open a new part and then open the Programs Settings screen.
System toolbar > M3 menu > Settings > Programs





2 Verify that the Auto Play Using Program Finder is set to Yes.
3 Press Done twice to exit the Settings screens.



Motion Monitor

The Motion Monitor launches the Auto Play function when any motion is detected in the field of view:

- One or more parts are placed in the field of view
- Part positions are changed

The Auto Play function then performs pattern recognition to identify a familiar part.

Operation Example

The following example includes:

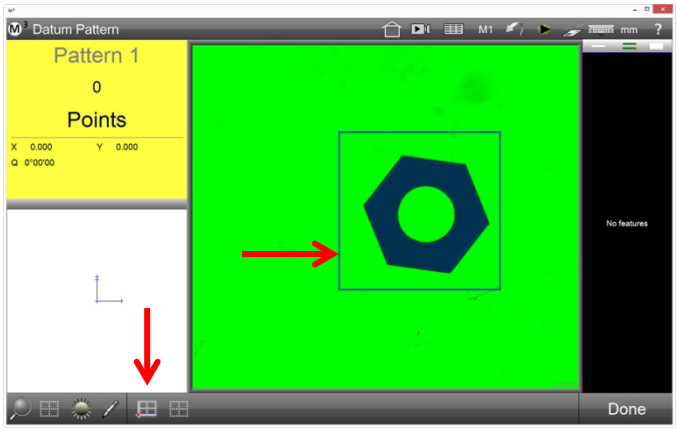
- Measuring and saving a part program
- Playing a part program without Program Finder
- Adding a part to the Program Finder list
- Playing programs using Program Finder and Auto Play
- Playing Programs to Measure Multiple Identical Parts

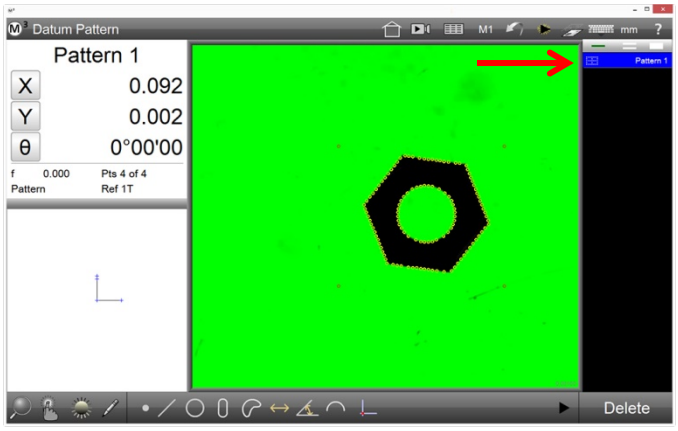
Measuring and Saving A Part Program

To measure and save a part:

1 Place a part in the field of view and measure (teach) a datum pattern. This pattern will be used in program playback to register the part.

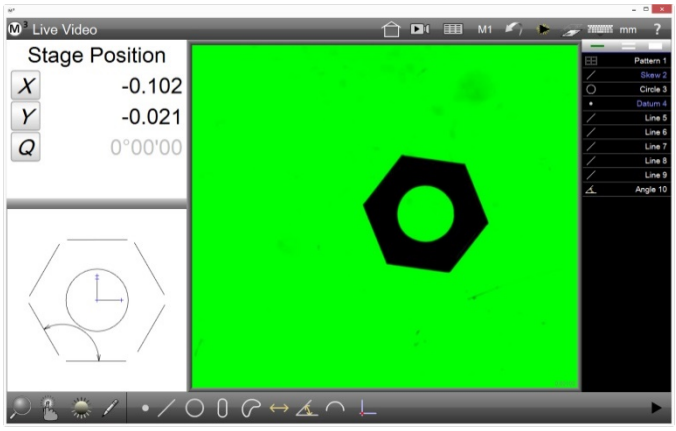
Measure toolbar > Datum > Datum pattern

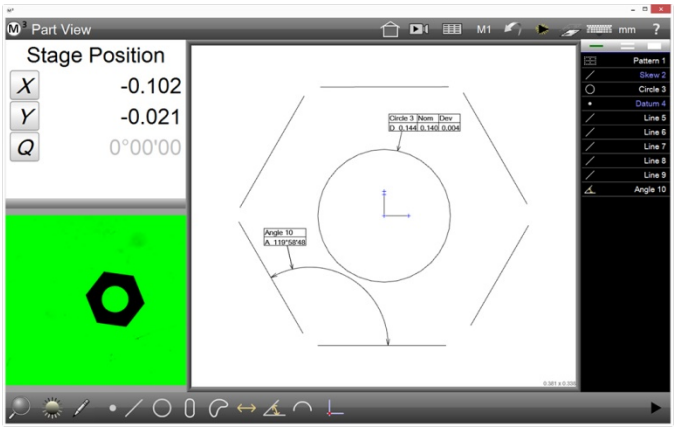




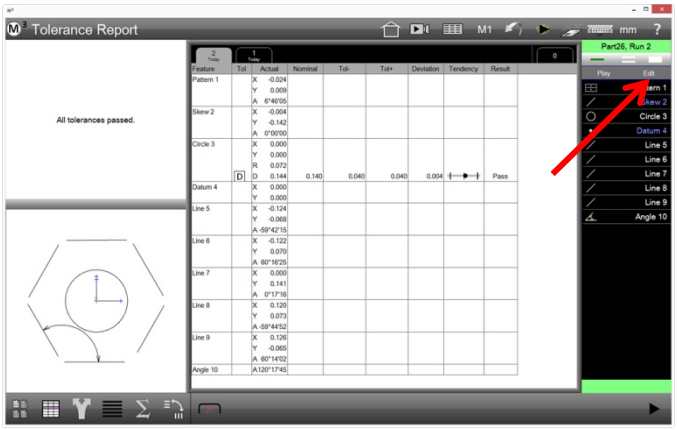
2 Measure part features and perform functions as desired. In this example:

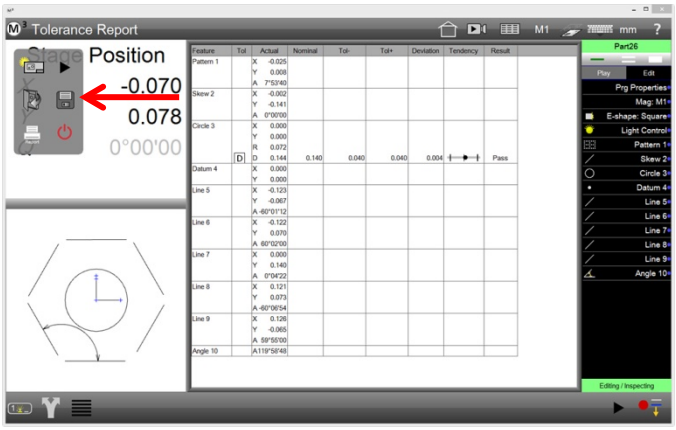
- The flat closest to the horizontal was used to skew the part
- The whole center was used as a zero datum
- An angle was constructed
- A tolerance was applied to the circle
- The part view was marked up



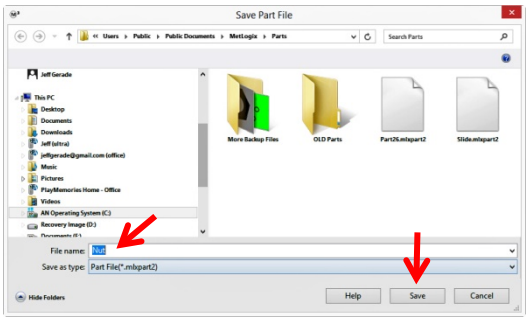


3 Play the program to verify its correct operation and save it.
Measure toolbar > Play button
Feature list > Edit tab
System toolbar > M3 menu > Save button





4 The Save Part File dialog will be displayed. Name the part file and press Save.



Playing a Part Program without Program Finder

Programs, in the form of measurement, tolerance and other software operations are played as a sequence of program steps. When a saved part is opened, the Feature List will become populated with features that were stored in the part file the last time the file was used and either:

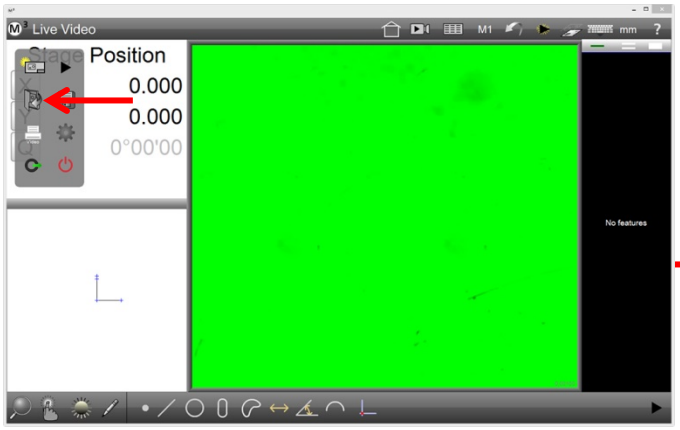
- A tabbed database of measurement results will be shown for a program that has been run to measure features

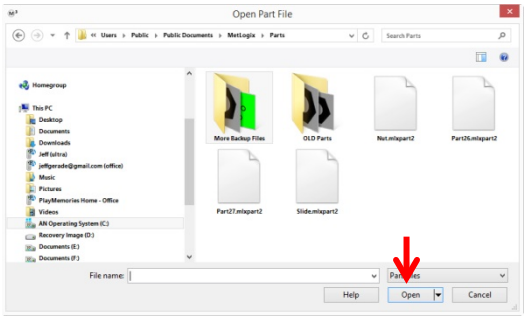
OR

- The original program feature data will be displayed for a program that was never run to measure features

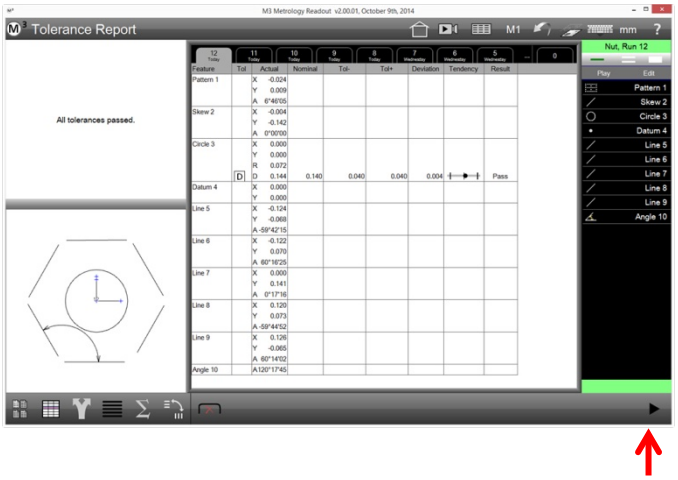
To load and play a part program:

1 Press Open Part File to display the Open Part File dialog.
2 Select the desired part file and press Open.
System menu > M3 menu > Open Part File





3 Press Play to play the part program file.



Adding a Part to the Program Finder

The part program must be added to the Program Finder list before the system can load the program automatically when the part pattern is recognized in the field of view. The part can be added:

- With Auto Play button displayed

OR

- Without Auto Play button displayed

Adding a Part Program with Auto Play

Auto Play runs a part program automatically when the part pattern is recognized in the field of view. When Auto Play is displayed in the System toolbar, the Auto Play function is shown either enabled or disabled.

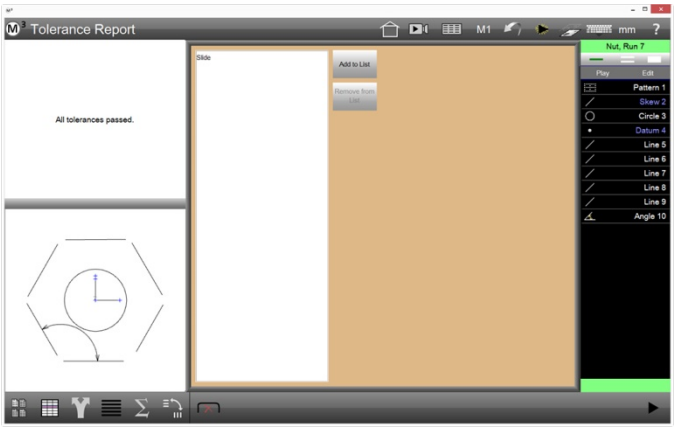
Enabled

Disabled

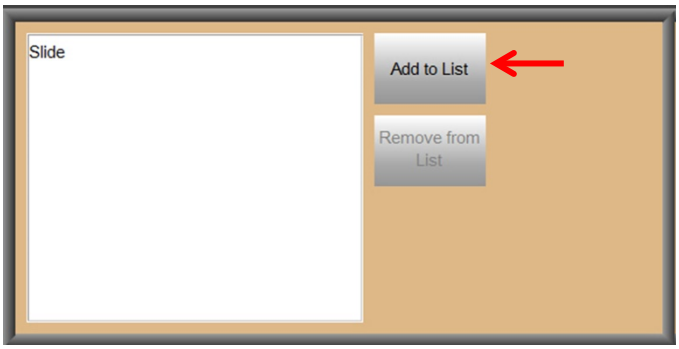
If the Auto Play button is not displayed, use the *Adding a Part Without Auto Play* procedure discussed in the next paragraphs. If the Auto Play button is displayed, in either enabled or disabled mode, use this procedure to add a part to the Program Finder. To add the part:

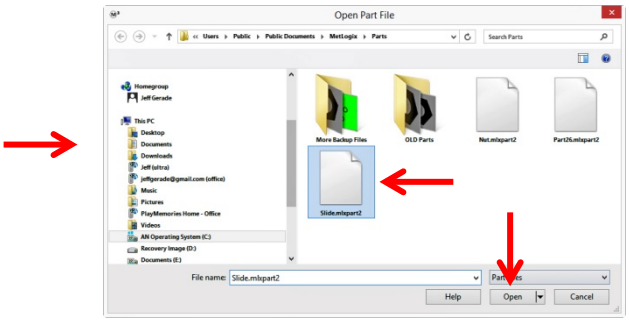
1 Long-press either Auto Play button to display the Program Finder Settings screen.

System toolbar > Long-press Auto Play



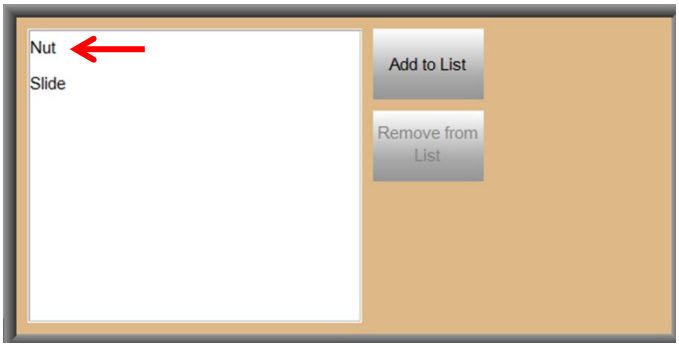
2 Press the Add button to display the Open Part File dialog, select the program and press Open. The program will be added to the Program Finder list.





3 Press Done to exit from the Settings screens

Measure toolbar > Done



Adding a Part Program without Auto Play

Auto Play runs a part program automatically when the part pattern is recognized in the field of view. When Auto Play is displayed in the System toolbar, the Auto Play function is shown either enabled or disabled.

Enabled

Disabled

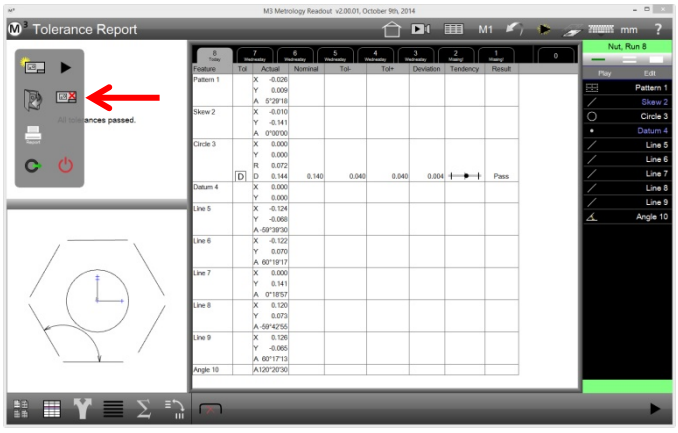
To add a part to the Part Finder when the Auto Play button is not displayed in either enabled or disabled modes:

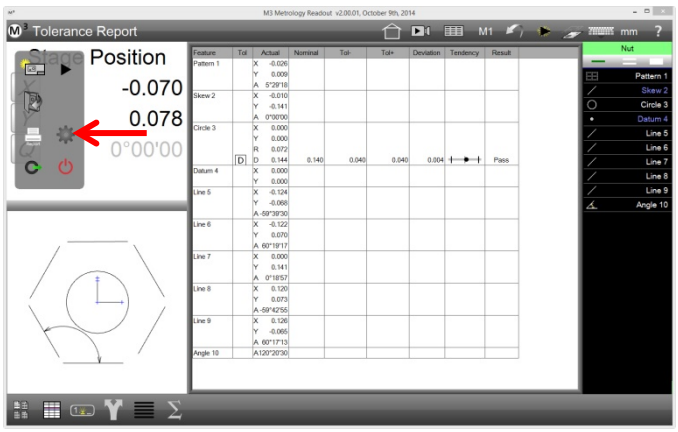
1 Press the Exit Part button to close the current part file. The file will be retained in system memory.

2 Press the Settings button to display the Settings screen.

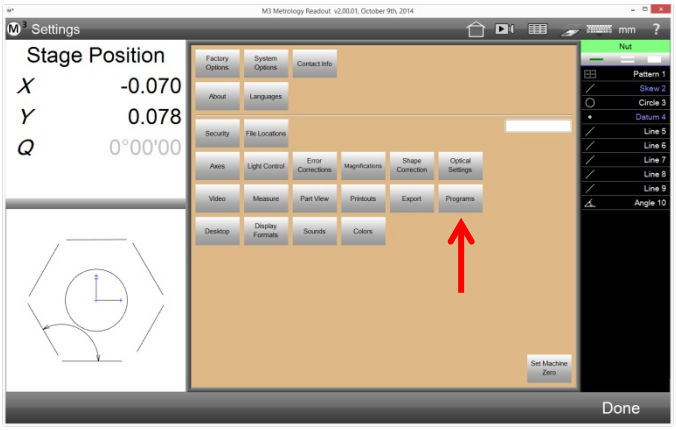
System toolbar > M3 menu > Close Part button

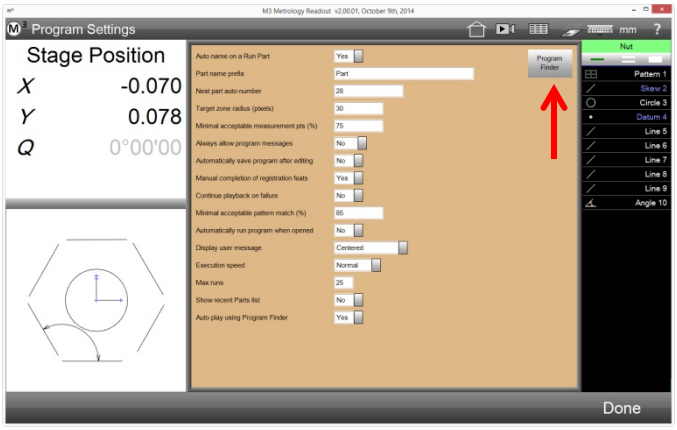
System toolbar > M3 menu > Settings button





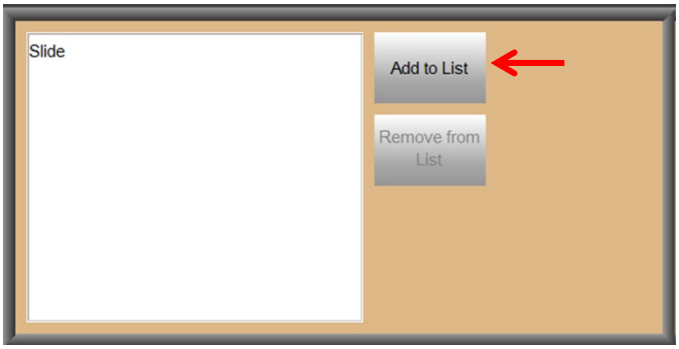
3 Press the Programs button to display the Programs Settings screen.

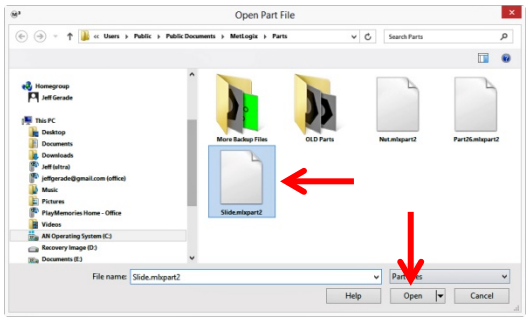




4 Press the Program Finder button to display the Program Finder list.

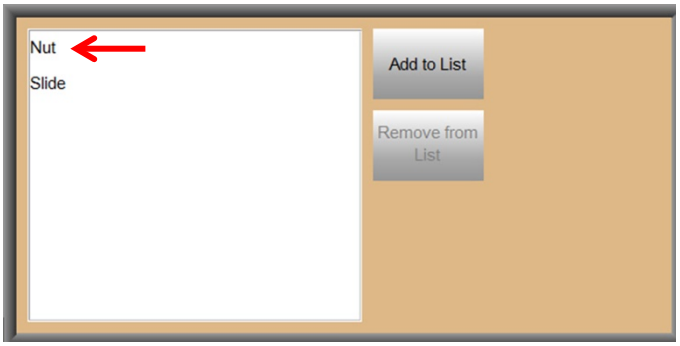
5 Press the Add button to display the Open Part File dialog, select the program and press Open. The program will be added to the Program Finder list.





6 Press Done to exit from the Settings screens

Measure toolbar > Done



Playing Programs using Program Finder and Auto Play

Programs played with Program Finder can be played without or with the use of the Auto Play function.

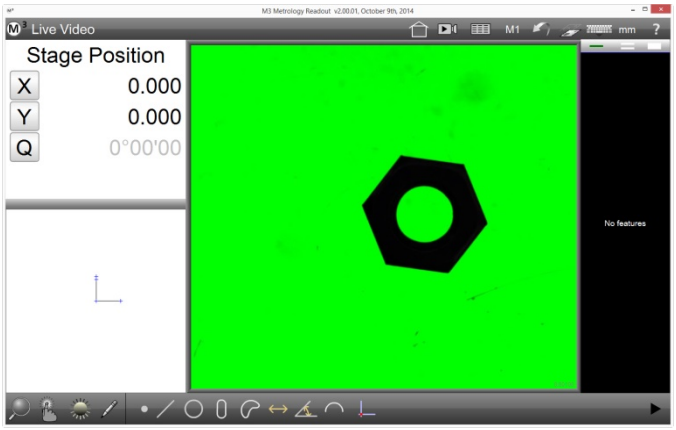
Playing a Program Without Auto Play

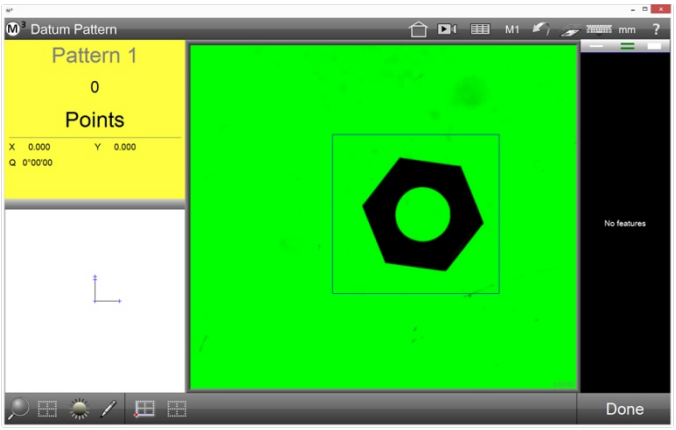
When programs are played using Program Finder but not using Auto Play:

- User places part in the field of view
- User probes the part with the Pattern Datum Probe
- System recognizes pattern and loads the program with the runs database
- User runs the program

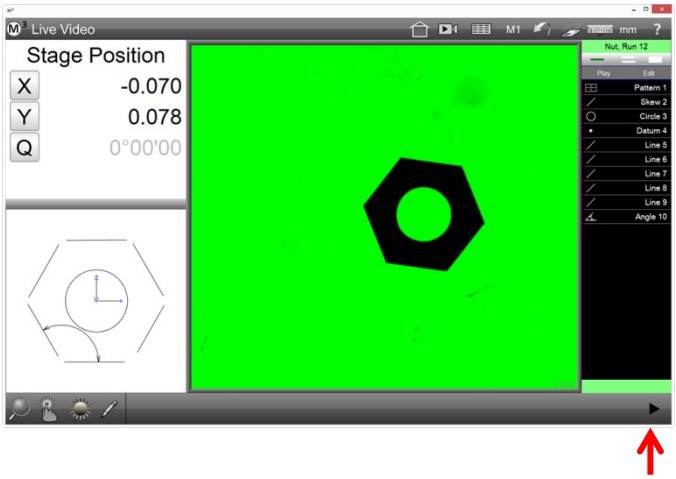
To play a program without Auto Play:

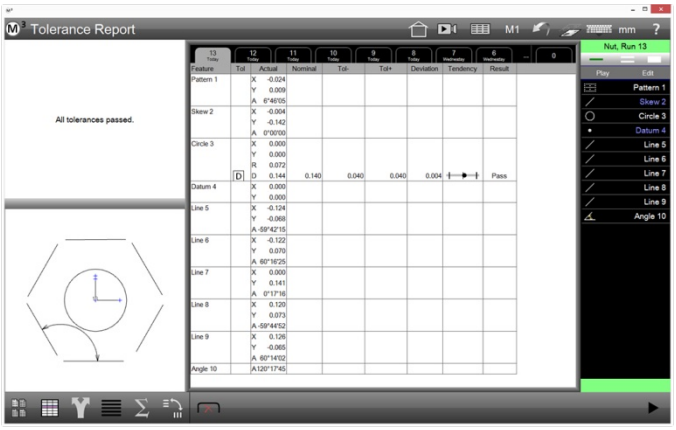
1 Place the part into the field of view and probe the part with the Pattern Datum probe.





2 The pattern will be recognized and the appropriate program will be loaded. Press Play to play the program.





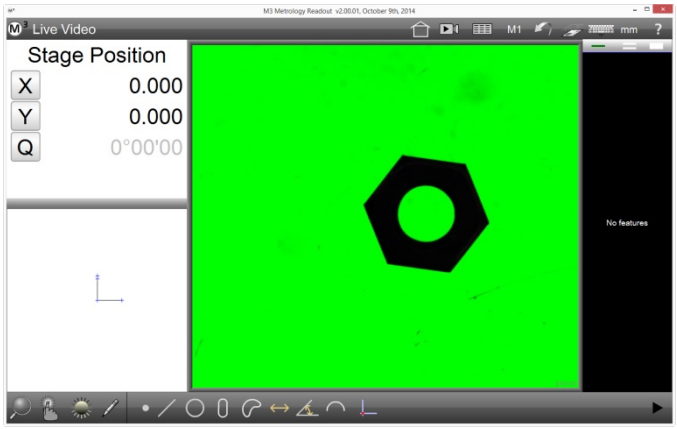
Playing a Program with Auto Play

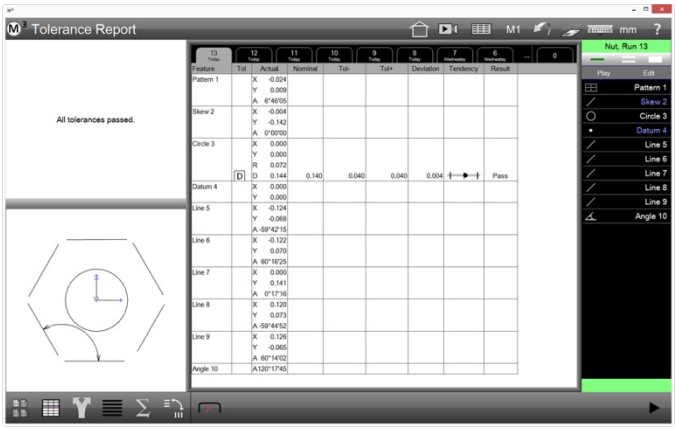
When programs are played using Program Finder and Auto Play:

- User places part in the field of view
- System recognizes pattern after Motion Monitor detects movement in the field of view, loads program with runs database and runs the program.

To play a program using Auto Play:

1 Place the part into the field of view. The pattern will be recognized and the program will be played.





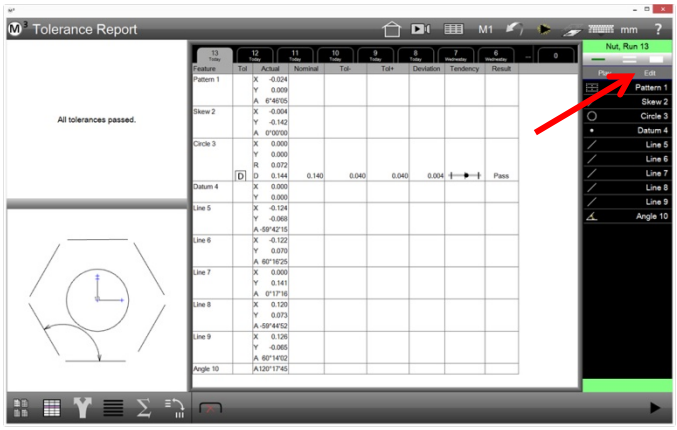
Playing Programs to Measure Multiple Identical Parts

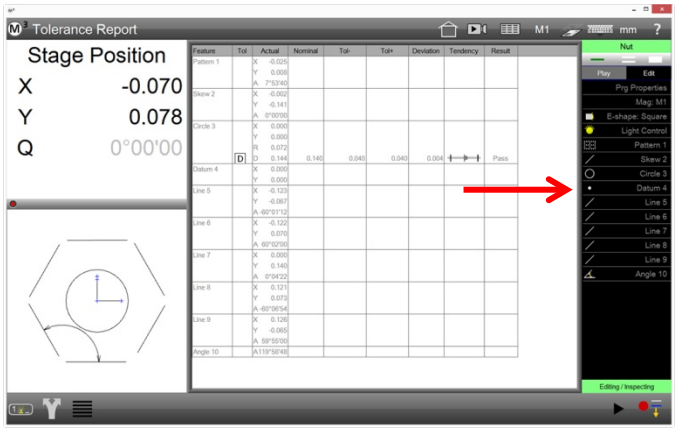
The concepts and procedures described for playing programs to measure single parts using the Program Finder and Auto Play also apply to the measurement of multiple identical parts. However, before playing a program to measure multiple parts, the program properties configuration settings must be edited.

Editing Program Properties to Measure Multiple Parts

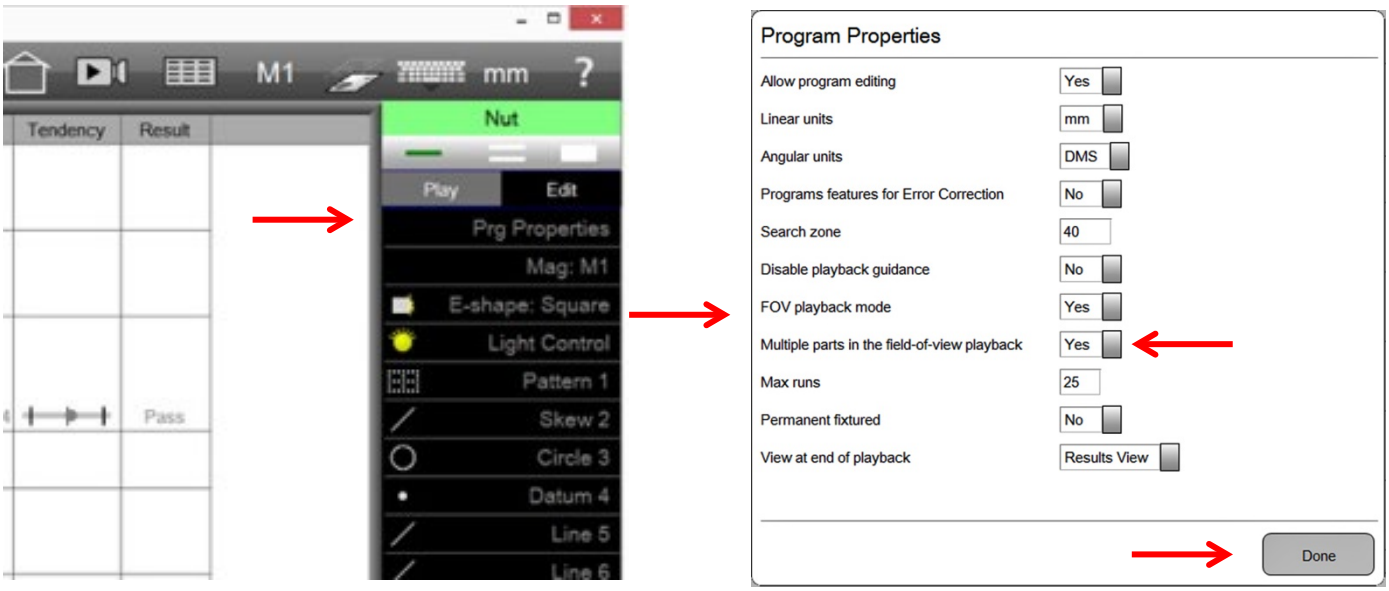
To edit program properties:

1 Load the desired program and then press the Edit tab to display the program steps.



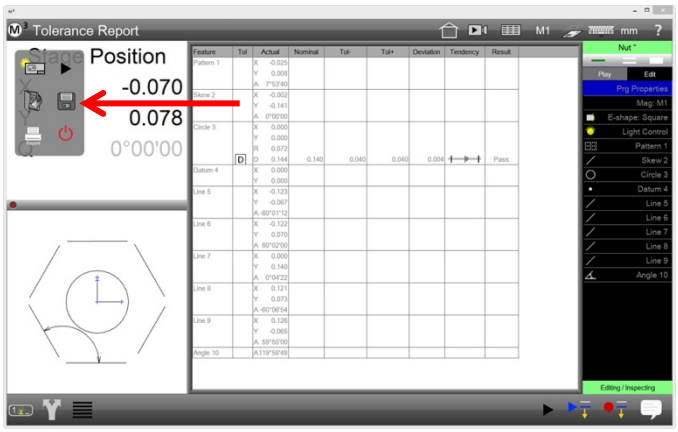


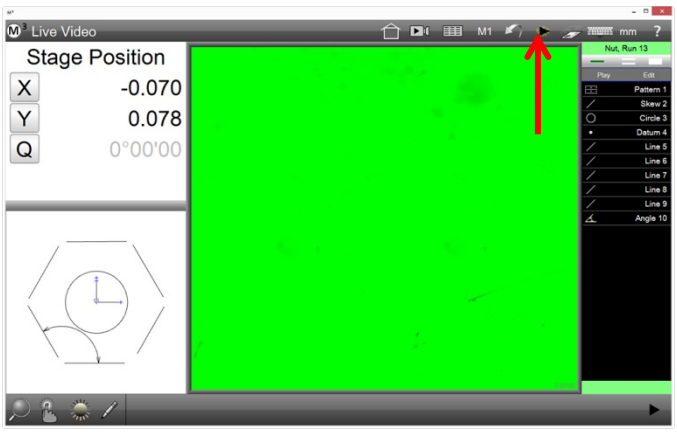
2 Press Program Properties to display the program configuration settings and then set the Multiple Parts in Field of View to Yes.



3 Press done to exit Program Properties and then press Save to save the configuration change.

4 Press the Edit tab again to Exit the edit mode and then verify that Auto Play is toggled to enabled.

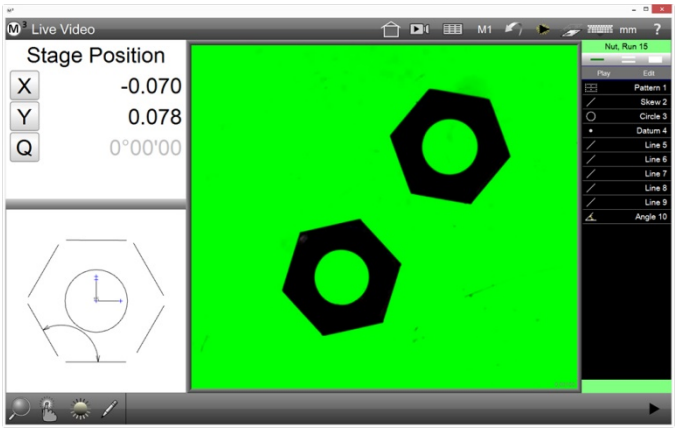


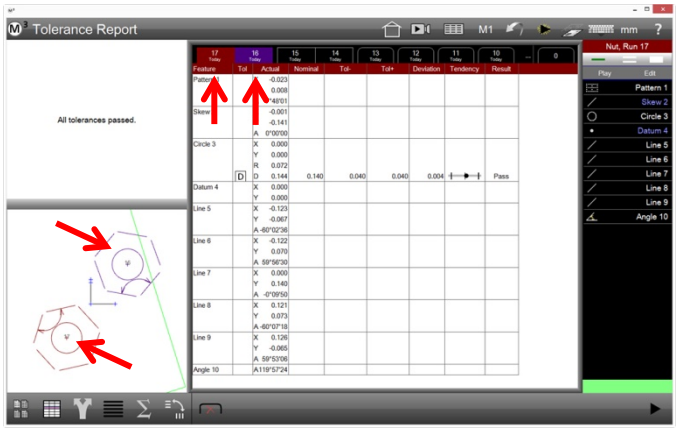


Playing the Program with Multiple Parts

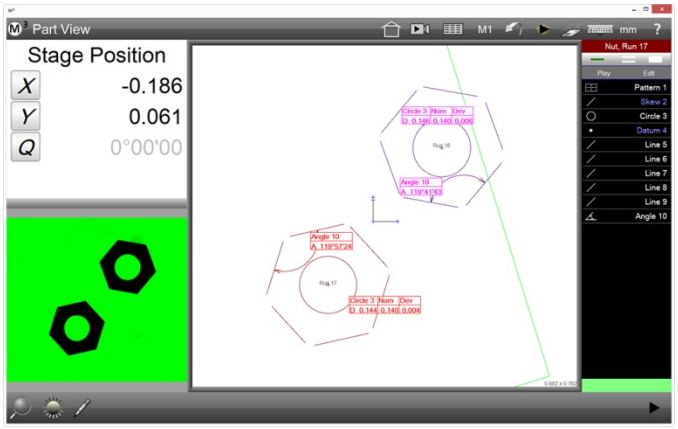
To play a program to measure multiple parts:

1 Place the parts into the field of view. The pattern will be recognized and the program will be played. The tabs in the Results view for the run of multiple parts will be connected. The run identification numbers will be shown on the run tab and in the part view.





Run identification numbers and markups created in the program will be shown in the enlarged Part View screen.



Product Warranty

This product is warranted against defects in workmanship, material and design for two (2) years from date of delivery to the extent that METLOGIX will, at its sole option, repair or replace the product or any part thereof which is defective, provided, however, that this warranty shall not apply to products subjected to tampering or, abuse, or exposed to highly corrosive conditions.

THIS WARRANTY IS IN LIEU OF ALL OTHER WARRANTIES WHETHER EXPRESS OR IMPLIED AND METLOGIX HEREBY DISCLAIMS ALL OTHER WARRANTIES, INCLUDING, WITHOUT LIMITATION, ANY WARRANTY OF FITNESS FOR A PARTICULAR PURPOSE OR MERCHANTABILITY. METLOGIX SHALL NOT BE LIABLE FOR ANY INCIDENTAL OR CONSEQUENTIAL DAMAGES, INCLUDING, BUT NOT LIMITED TO, ANY ANTICIPATED OR LOST PROFITS.

This warranty is voidable if the purchaser fails to follow any and all instructions, warnings or cautions in the product's Instruction Manual.

If a manufacturing defect is found, METLOGIX will replace or repair the product or replace any defective part thereof without charge; however, METLOGIX's obligation hereunder does not include the cost of transportation which must be borne by the customer. METLOGIX assumes no responsibility for damage in transit, and any claims for such damage should be presented to the carrier by the purchaser.

SOFTWARE LICENSE AGREEMENT

NOTICE TO USER: PLEASE READ THIS AGREEMENT CAREFULLY. BY COPYING, INSTALLING, OR USING ALL OR ANY PORTION OF THE SOFTWARE, YOU ACCEPT ALL TERMS AND CONDITIONS OF THIS AGREEMENT, INCLUDING, IN PARTICULAR THE PROVISIONS ON: TRANSFERABILITY IN SECTION 4; WARRANTY IN SECTIONS 6 AND 7; AND LIABILITY IN SECTION 8. YOU AGREE THAT THIS AGREEMENT IS LIKE ANY WRITTEN NEGOTIATED AGREEMENT SIGNED BY YOU. THIS AGREEMENT IS ENFORCEABLE AGAINST YOU AND ANY LEGAL ENTITY THAT OBTAINED THE SOFTWARE AND ON WHOSE BEHALF IT IS USED; FOR EXAMPLE, IF APPLICABLE, YOUR EMPLOYER. IF YOU DO NOT AGREE TO THE TERMS OF THIS AGREEMENT, DO NOT USE THIS SOFTWARE.

METLOGIX AND ITS SUPPLIERS OWN ALL INTELLECTUAL PROPERTY IN THE SOFTWARE. THE SOFTWARE IS LICENSED, NOT SOLD. METLOGIX PERMITS YOU TO COPY, DOWNLOAD, INSTALL, USE, OR OTHERWISE BENEFIT FROM THE FUNCTIONALITY OR INTELLECTUAL PROPERTY OF THE SOFTWARE ONLY IN ACCORDANCE WITH THE TERMS OF THIS AGREEMENT. USE OF SOME METLOGIX AND SOME NON-METLOGIX MATERIALS AND SERVICES INCLUDED IN OR ACCESSED THROUGH THE SOFTWARE MAY BE SUBJECT TO OTHER TERMS AND CONDITIONS TYPICALLY FOUND IN A SEPARATE LICENSE AGREEMENT, TERM OF USE OR "READ ME" FILE LOCATED WITHIN OR NEAR SUCH MATERIALS AND SERVICES.

THE SOFTWARE MAY INCLUDE TECHNOLOGY DESIGNED TO PREVENT UNAUTHORIZED USE AND COPYING. THIS TECHNOLOGY MAY CAUSE YOUR COMPUTER TO CONNECT TO THE INTERNET WITHOUT NOTICE ON INSTALL, LAUNCH, AND ON AN INTERMITTENT OR REGULAR BASIS THEREAFTER. ONCE CONNECTED, THE SOFTWARE WILL TRANSMIT INFORMATION TO METLOGIX VIA AN INTERNET CONNECTION. METLOGIX MAY USE THE TRANSMITTED INFORMATION, AS WELL AS OTHER INFORMATION YOU OUR THIRD PARTIES MAY PROVIDE TO US, TO DETECT OR PREVENT USE OF METLOGIX SOFTWARE THIS IS FRAUDULENT, NOT VALIDLY LICENSED, OR NOT USED IN ACCORDANCE WITH A VALID LICENSE. SOFTWARE THAT FAILS TO CONNECTO TO METLOGIX AND SOFTWARE THAT CONNECTS TO METLOGIX WHOSE USE IS FOUND TO NOT TO BE PERMITTED MAY OFFER ONLY LIMITED FUNCTIONALITY OR MAY NOT OPERATE AT ALL.

1. Definitions

"MetLogix" means MetLogix, Incorporated, a Delaware corporation, 175 Canal Street, Manchester, New Hampshire 03101.

"MetLogix Runtime(s)" means MetLogix M3, MetLogix M2, MetLogix M1, MetLogix L3, MetLogix L2, MetLogix L1, MetLogix Test Builder, or MetLogix CFR21.

"Computer" means a virtual or physical computer device that accepts information in digital or similar form and manipulates it for a specific result based on a sequence of instructions.

"Internal Network" means a private, proprietary network resource accessible only by employees and individual contractors (i.e., temporary employees) or a specific corporation or similar business entity. Internal Network does not include portions of the Internet or any other network community open to the public, such as membership or subscription driven groups, associations, and similar organizations.

"Output File" means an output file you created with the Software.

"Permitted Number" means one (1) unless otherwise indicated under a valid license (e.g., volume license) granted by MetLogix.

"Software" means (a) all of the information with which this agreement is provided, including but not limited to: (i) all software files and other computer information.

2. Software License

If you obtained the Software and any required serial number(s) from MetLogix or one of its authorized licensees (resellers) and as long as you comply with the terms of this Agreement, MetLogix grants you a non-exclusive license to install and use the Software in a manner consistent with its design and Documentation and as further set forth below.

2.1 Limited Use

2.1.1

The Software, or portions of the Software, may allow installation and use without a serial number. If so, you may install but not use such non-serialized Software on any number of Computers as part of an organizational deployment plan. Further, you may install and use such non-serialized Software on any number of Computers for demonstration, evaluation, and training purposes only and only if any Output Files or other material produced through such use are used only for internal, non-commercial, and non-production purposes. You may not use such non-serialized Software after any applicable timeout period has ended, unless you input a valid serial number. ACCESS TO AND USE OF ANY OUTPUT FILES CREATED WITH SUCH NON-SERIALIZED SOFTWARE IS ENTIRELY AT YOUR OWN RISK.

2.1.2

The Software, or portions of the Software, that are provided with a serial number designated for "evaluation purposes" or other similar designation (such as Software or a serial number supplied as "EVAL" under a separate ordering document) and that includes an expiration date ("Evaluation Software") may only be installed and used on Computers for demonstration, evaluation, and training purposes only and only if Output Files or other materials produced through such use are used only for internal, non-commercial, and non-production purposes. The Evaluation Software will automatically time out with or without prior notice. You may not use such Evaluation Software after any applicable time-out period has ended, unless you input a valid serial number. USE OF ANY OUTPUT FILES CREATED WITH SUCH NON-SERIALIZED SOFTWARE IS ENTIRELY AT YOUR OWN RISK.

2.1.3 General Use

Except as described otherwise herein, you may install and use one (1) copy of the Software in accordance with the Documentation only on the Permitted Number of your compatible Computers into which you enter a valid serial number.

2.1.4 Distribution from a Server

The Software may not be distributed from a Server or an Internal Network.

2.1.5 Dual Boot Platform

The Software is licensed for use on a specific operating system platform. You must purchase a separate license for use of the Software on each operating system platform.

2.1.6 Documentation

You may make copies of the Documentation for your own internal use in connection with use of the Software in accordance with this Agreement but no more than the amount reasonably necessary. Any permitted copy of the Documentation that you make must contain the same copyright and other proprietary notices that appear on or in the Documentation.

3. Intellectual Property Ownership

The Software and any authorized copies that you make are the intellectual property of and are owned by MetLogix Incorporated. The structure, organization, and source code of the Software are the valuable trade secrets and confidential information of MetLogix Incorporated. The Software is protected by law, including but not limited to the copyright laws of the United States and other countries, and by international treaty provisions. Except as expressly stated herein, this Agreement does not grant you any intellectual property rights in the Software and all rights not expressly granted are reserved by MetLogix Incorporated.

4. Restrictions and Requirements

4.1 Notices

Any permitted copy of the Software that you make must contain the same copyright and other proprietary notices that appear on or in the Software.

4.2 Use Obligations

You agree that you will not use the Software other than as permitted by this Agreement and that you will not use the Software in a manner inconsistent with its design or Documentation.

4.3 No Modifications

You may not modify, port, adapt, or translate the Software.

4.4 No Reverse Engineering

You will not reverse engineer, decompile, disassemble, or otherwise attempt to discover the source code of the Software.

4.5 No Unbundling

The Software may include various applications, utilities, and components, may support multiple platforms and languages, and may be provided to you on multiple media or in multiple copies. Nonetheless, the Software is designed and provided to you as a single product to be used as a single produce on Computers permitted by Section 2. You are not required to install all component parts of the Software, but you may not unbundle the component parts of the Software for use on different Computers. You may not unbundle or repackage Software for distribution, transfer, or resale.

4.6 No Transfer

YOU WILL NOT RENT, LEASE, SELL, SUBLICENSE, ASSIGN OR TRANSFER YOUR RIGHTS IN THE SOFTWARE, OR AUTHORIZE ANY PORTION OF THE SOFTWARE TO BE COPIED ONTO ANOTHER INDIVIDUAL OR LEGAL ENTITY'S COMPUTER EXCEPT AS MAY BE EXPRESSLY PERMITTED HEREIN. You may, however, permanently transfer all your rights to use the Software to another individual or legal entity provided that: (a) you also transfer (i) this agreement, (ii) the serial number(s), the Software affixed to medial provided by MetLogix or its authorized distributor, and all other software or hardware bundled, packaged, or pre-installed with the Software, including all copies, Updates, and Prior Versions (as defined in Section 5); (b) you retain no Updates, Prior Versions, or copies, including backups and copies stored on a Computer; and (c) the receiving party accepts the terms and conditions of this agreement and any other terms and conditions under which you purchased a valid license to the Software. NOTWITHSTANDING THE FOREGOING, YOU MAY NOT TRANSFER EDUCATION, PRE-RELEASE, EVALUATION SOFTWARE, OR NOT FOR RESALE COPIES OF THE SOFTWARE OR SOFTWARE OBTAINS UNDER A METLOGIX VOLUME LICENSE PROGRAM EXCEPT AS MAY BE EXPRESSLY PERMITTED BY METLOGIX WITH THE TERMS OF A VOLUME LICENSE PROGRAM.

4.7 No Service Bureau

You will not use or offer the Software on a service bureau basis.

4.8 MetLogix M Series Restrictions

You may not use MetLogix M Series software on any non-PC device or with any embedded or device version of any operating system.

5. Updates

If the Software is an Update to a prior version of MetLogix software (the "Prior Version"), the following apply:

Your use of this Update is conditional upon your retention of the Prior Version. Therefore, if you validly transfer this Update pursuant to Section 4.6, you must transfer the Prior Version along with it. If you wish to use this Update in addition to the Prior Version, you may only do so on the same computer on which you have installed and are using the Prior Version. Any obligation that MetLogix may have to support Prior Versions may end upon the availability of this Update. No other use of the Update is permitted. Additional Updates may be licensed to you by MetLogix with additional or different items.

6. Limited Warranty

MetLogix warrants to the individual or entity that first purchases a license for the Software for use pursuant to the terms of this agreement that the Software will perform substantially in accordance with the Documentation for the ninety (90) day period following receipt of the Software when used on the recommended operating system and hardware configuration. Non-substantial variation of performance from the Documentation does not establish a warranty right. This limited warranty does not apply to the following which are made available "AS-IS" and without warranty from MetLogix: patches, font software; pre-release (beta), trial, starter, evaluation, product sampler, and not for resale (NFR) copies of the Software including but not limited to Evaluation Software. All warranty claims must be made, along with proof of purchase, to MetLogix within such ninety (90) day period. If the Software does not perform substantially in accordance with the Documentation, the entire liability of MetLogix and its affiliates and your exclusive remedy will be limited to either, at MetLogix' option, replacement of the Software or refund of the license fee you paid for the Software (if any). THE LIMITED WARRANTY SET FORTH IN THIS SECTION GIVES YOU SPECIFIC LEGAL RIGHTS. YOU MAY HAVE ADDITIONAL RIGHTS UNDER LAW WHICH VARY FROM JURISDICTION TO JURISDICTION. METLOGIX DOES NOT SEEK TO LIMIT YOUR WARRANTY RIGHTS TO ANY EXTENT NOT PERMITTED BY LAW.

7. Disclaimer

THE LIMITED WARRANTY IN SECTION 6 IS THE ONLY WARRANTY OFFERED BY METLOGIX, ITS AFFILIATES, AND SUPPLIERS AND IT STATES THE SOLE AND EXCLUSIVE REMEDIES FOR METLOGIX', ITS AFFILIATES', OR SUPPLIERS' BREACH OF THAT OFFERED WARRANTY. THE LIMITED WARRANTY IN SECTION 6 AND ANY STATUTORY WARRANTY AND REMEDY THAT CANNOT BE EXCLUDED OR LIMITED UNDER LAW ARE THE ONLY WARRANTIES APPLICBLE TO THE SOFTWARE. OTHER THAN THOSE OFFERED AND STATUTORY WARRANTIES APPLICABLE TO THE SOFTWARE, METLOGIX, ITS AFFILIATES, SUPPLIERS, AND CERTIFICATE AUTHORITIES DISCLAIM ALL WARRANTIES, CONDITIONS, REPRESENTATIONS, AND TERMS, EXPRESSED OR IMPLIED, WHETHER BY STATUTE, COMMON LAW, CUSTOM, USAGE, OR OTHERWISE AS TO ANY MATTER, INCLUDING BUT NOT LIMITED TO PERFORMANCE, SECURITY, NON-INFRINGEMENT OF THIRD PARTY RIGHTS, INTEGRATION, MERCHANTABILITY, SATISFACTORY QUALITY, AND FITNESS FOR ANY PARTICULAR PURPOSE. OTHER THAN SUCH OFFERED AND STATUTORY WARRANTIES AND REMEDIES, METLOGIX, ITS AFFILIATES, SUPPLIERS, AND CERTIFICATE AUTHORITIES PROVIDE THE SOFTWARE AND ACCESS TO ANY WEBSITES, METLOGIX OR THIRD PARTY ONLINE SERVICES, AND CERTIFICATE AUTHORITY SERVICES AS-IS AND WITH ALL FAULTS. THIS DISCLAIMER OF WARRANTY MAY NOT BE VALID IN SOME JURISDICTIONS. YOU MAY HAVE ADDITIONAL WARRANTY RIGHTS UNDER LAW WHICH MAY NOT BE WAIVED OR DISCLAIMED. METLOGIX DOES NOT SEEK TO LIMIT YOUR WARRANTY RIGHTS TO ANY EXTENT NOT PERMITTED BY LAW.

8. Limitation of Liability

EXCEPT FOR THE EXCLUSIVE REMEDY OFFERED BY METLOGIX ABOVE AND ANY REMEDIES THAT CANNOT BE EXLCUDED OR LIMITED UNDER LAW, METLOGIX, ITS AFFILIATES, SUPPLIERS AND CERTIFICATE AUTHORITIES WILL NOT BE LIABLE TO YOU FOR ANY LOSS, DAMAGES, CLAIMS, OR COSTS WHATSOEVER INCLUDING ANY CONSEQUENTIAL, INDIRECT OR INCIDENTAL DAMAGES, ANY LOST PROFITS OR LOST SAVINGS, ANY DAMAGES RESULTING FROM BUSINESS INTERRUPTION, PERSONAL INJURY OR FAILURE TO MEET ANY DUTY OF CARE, OR CLAIMS BY THIRD PARTY, EVEN IF ANY METLOGIX REPRESENTATIVE HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH LOSS, DAMAGES, CLAIMS, OR COSTS. IN ANY EVENT, METLOGIX' AGGREGATE LIABILITY AND THAT OF ITS AFFILIATES, SUPPLIERS, AND CERTIFICATE AUTHORITIES UNDER OR IN CONNECTION WITH THIS AGREEMENT WILL BE LIMITED TO THE AMOUNT PAID FOR THE SOFTWARE, IF ANY. THIS LIMITATION WILL APPLY EVENT IN THE EVENT OF A FUNDAMENTAL OR MATERIAL BREACH OR A BREACH OF THE FUNDAMENTAL OR MATERIAL TERMS OF THIS AGREEMENT

THE FORECOING LIMITATIONS AND EXCLUSIONS APPLY TO THE EXTENT PERMITTED BY APPLICABLE LAW IN YOUR JURISDICTION. THIS LIMITATION OF LIABILITY MAY NOT BE VALID IN SOME JURISDICTIONS. YOU MAY HAVE RIGHTS THAT CANNOT BE WAIVED UNDER CONSUMER PROTECTION AND OTHER LAWS. METLOGIX DOES NOT SEEK TO LIMIT YHOUR WARRANTY OR REMEDIES TO ANY EXTENT NOT PERMITTED BY LAW.

9. Export Rules

You acknowledge that the Software is subject to the U.S. Export Administration Regulations and other export laws, restrictions, and regulations (collectively the "Export Laws") and that you will comply with the Export Laws. You will not ship, transfer, export, or re-export the Software, directly or indirectly, to: (a) any countries that are subject to US export restrictions (currently including, but not necessarily limited to, Cuba, Iran, North Korea, Sudan, and Syria) (each, an "Embargoed Country"), (b) any end user who you know or have reason to know will utilize them in design, development or production of nuclear, chemical or biological weapons, or rocket systems, space launch vehicles, and sounding rockets, or unmanned air vehicle systems (each, a "Prohibited Use"), or (c) any end user who has been prohibited from participating in U.S. export transactions by any federal agency of the U.S. government (each, a "Sanctioned Party"). In addition, you are responsible for complying with any local laws in your jurisdiction which may impact your right to import, export or use the Software. You represent and warranty that (i) you are not a citizen of, or located within, an Embargoed Country, (ii) you will not use the Software for a Prohibited Use, and (iii) you are not a Sanctioned Party. All rights to use the Software are granted on condition that such rights are forfeited if you fail to comply with the terms of this agreement.

10. General Provisions

If any part of this Agreement is found void and unenforceable, it will not affect the validity of the balance of this Agreement, which will remain valid and enforceable according to its terms. This Agreement may only be modified in writing, signed by an officer of MetLogix, Inc. The English version of this Agreement will be the version used when interpreting or construing this Agreement. This is the entire Agreement between MetLogix and you relating to the Software and it supersedes any prior representations, discussion, undertakings, communications, or advertising relating to the Software.

11. Notice to U.S. Government End Users

For U.S. Government End Users, MetLogix agrees to comply with all applicable equal opportunity laws including, if appropriate, the provisions of Executive Order 11246, as amended, Section 402 of the Vietnam Era Veterans Readjustment Assistance Act of 1974 (38 USC 4212), and Section 503 of the Rehabilitation Act of 1973, as amended, and the regulations at 41 CFR Parts 60-1 through 60-60, 60-250, and 60-741. The affirmative action clause and regulations contained in the preceding sentence will be incorporated by reference in this Agreement.

12. Compliance with Licenses

If you are a business, company, or organization, you agree that, in addition to any license compliance checking performed by the Software, MetLogix or its authorized representative have the right, no more than once every twelve (12) months, upon ten (10) days' prior notice to you, to inspect your records, systems, and facilities to verify that your use of any and all MetLogix software is in conformity with your valid licenses from MetLogix. For example, MetLogix has the right to those of your records useful to determine whether installations of the Software have, or have not, been serialized, and you shall provide such records to MetLogix promptly upon request by MetLogix. Additionally, you shall provide MetLogix with all records and information requested by MetLogix in order to verify that your use of any and all MetLogix software is in conformity with your valid licenses from MetLogix within thirty (30) days of MetLogix' request.

13. Peer to Peer Communications

The Software may use your connection to a local area network, without additional notice, automatically connect to other MetLogix software and, in doing so, may indicate on the local area network that it is available for communication with other MetLogix software. These connections may transmit the IP Address of your connection to the local network but no personally identifiable information is ever transmitted or received through such network connections (except to the extent that IP addresses my be considered personally identifiable in some jurisdictions).