CAM2Q

FaroArm:
Training Workbook

Version 1.5 • October 2010

FARO

FARO® Institute

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Training Workbook

Version 1.5 • October 2010

CAM2®

CAM2Q

About this Training

Intended audience: any user who will use FARO CAM2 Q

Length of Training: Three (3) days

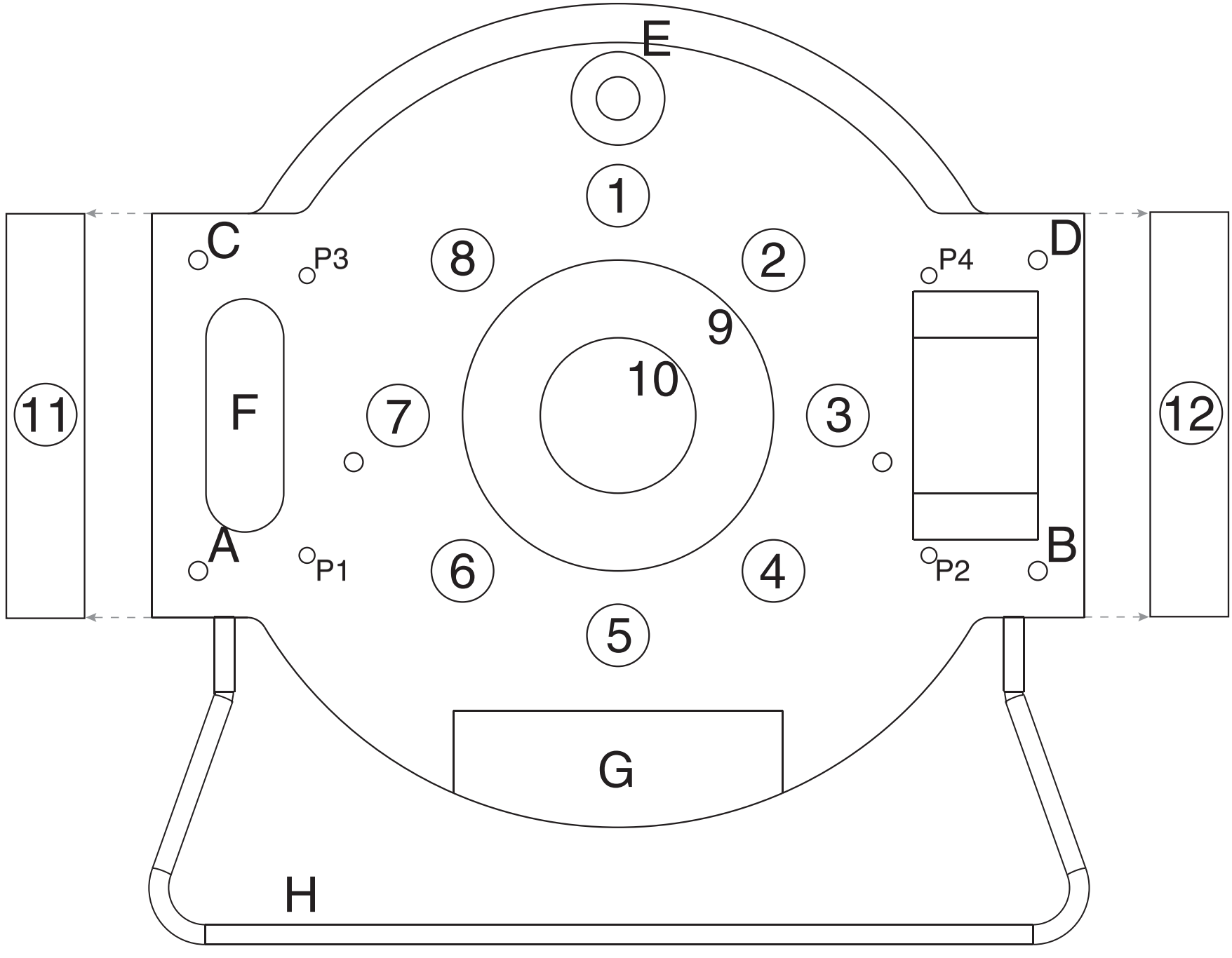
Structure: Modules and Lessons.

The training does not intend to cover every menu item of CAM2 Q. We have taken the most common applications of CAM2 Q and developed a training program that teaches you core principles, measuring strategies, and tips in a hands-on training format. Our intent is for this training to provide a strong foundation for you to start using your FARO system in your environment. Additionally, it will prepare you to learn more advanced features of the software.

Definition of Terms and styles:

	Information and tips
	Caution and Warning statements
	General information

Bold Italic Text	Indicates directory names, menu names, buttons, tabs, key names, acronyms, and modes.
`Monospaced` Text	Indicates alpha/numeric characters or values you enter in a field on the screen. For example, "Type `0.005` for the tolerance setting."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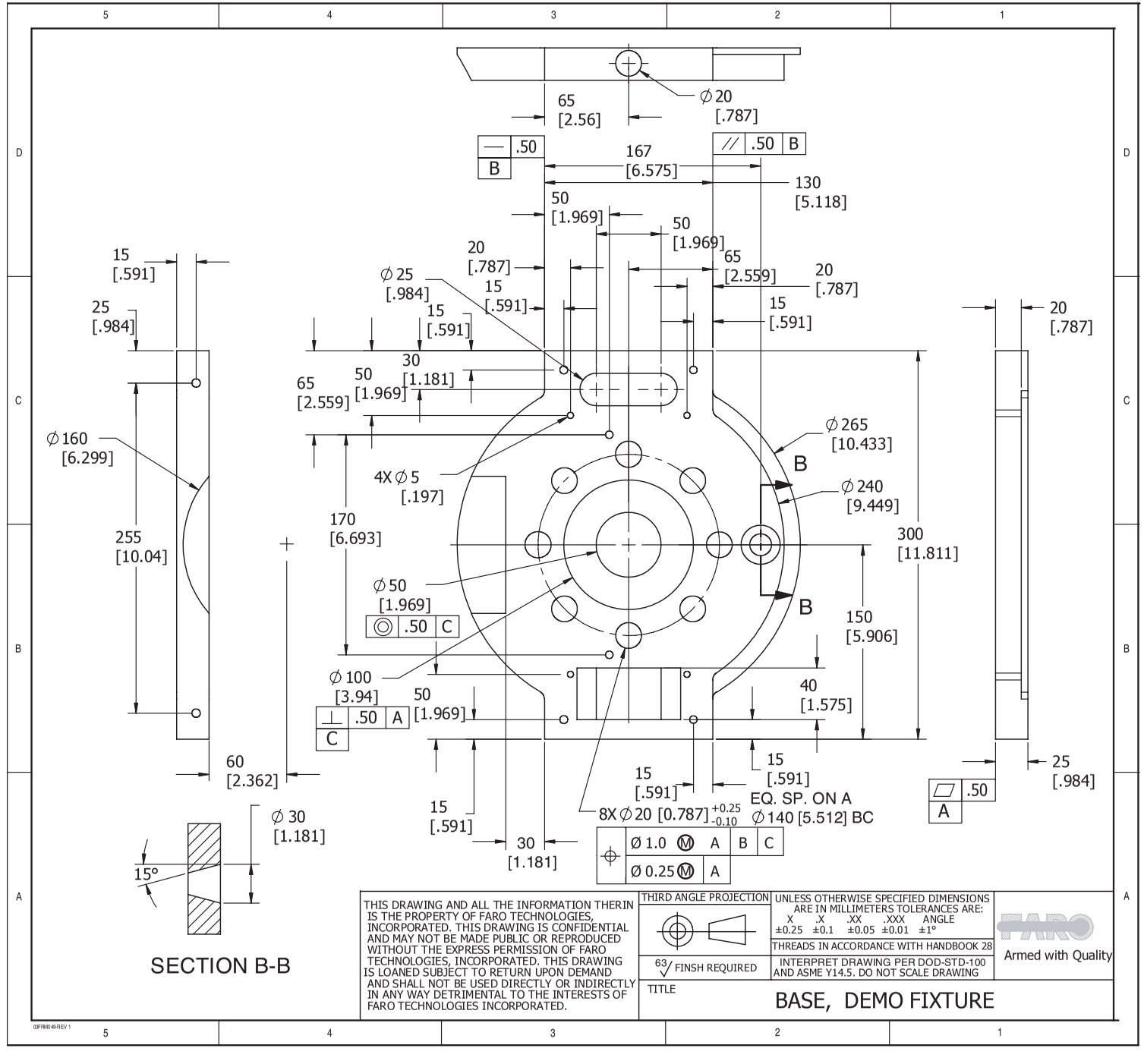
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SECTION B-B
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THIRD ANGLE PROJECTION
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UNLESS OTHERWISE SPECIFIED DIMENSIONS ARE IN MILLIMETERS TOLERANCES ARE:
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INTERPRET DRAWING PER DOD-STD-100 AND ASME Y14.5. DO NOT SCALE DRAWING
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Armed with Quality
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Module 1:
Measurement System Overview

In this module, you will become familiar with how the FARO measurement device and CAM2 Q software interact with one another to provide accurate 3D measurements.

MODULE OBJECTIVES

You will be able to:

- Set up the FARO measurement device
- Navigate the CAM2 Q user interface

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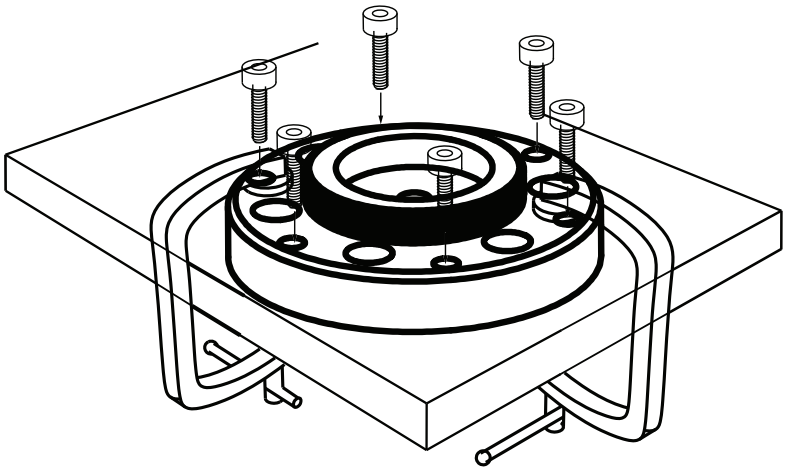
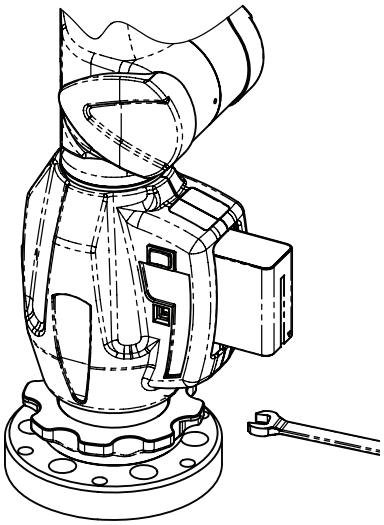
Lesson 1: Getting Started

Setting up the FaroArm

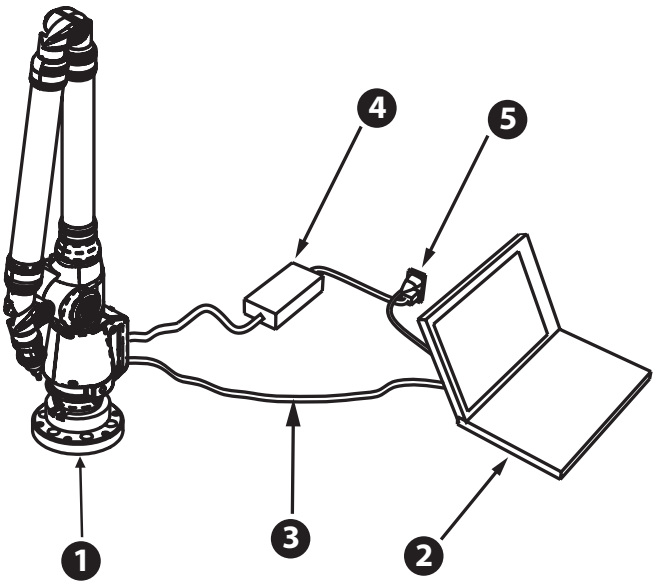
To set up the FaroArm, refer to the unpacking and setup instruction sheets included with the FaroArm. It is very important that you mount the FaroArm in a stable relationship to the part (the part should not move relative to the FaroArm).





- Attach the 3.5 inch threaded ring and surface mount plate to a stable location. A granite slab, or acceptably rigid surface, is recommended.
- Tighten all mounting bolts to 11.5 N-m (100-inch pounds).
- Place the FaroArm on top of the 3.5 inch threaded ring.
- Screw the threaded collar clamp onto the base of the FaroArm and the 3.5 inch threaded ring.
- Use the wrench to tighten the threaded collar clamp.

Poor mounting causes poor accuracy.



1. **FaroArm**
2. **Computer**
3. **USB Cable**
4. **FaroArm Power Supply**
5. **Power Outlet**

If an error occurs, contact FARO Customer Support. Please have a written description of the error, the order of your procedures just before the error occurred, and the FaroArm serial number before calling.

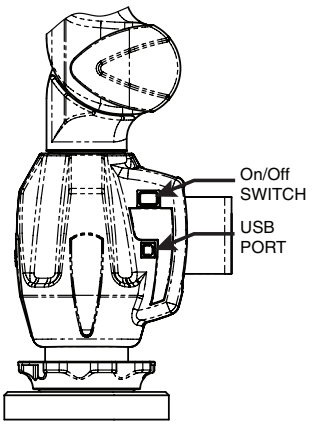
You can find the serial number on the base of the FaroArm.

If you are using a FaroArm with the FARO Wireless option, follow the procedures found in the ***FARO Wireless*** section of your FaroArm manual.

Connecting the FaroArm to the Computer

To connect the FaroArm to the computer:

1. Plug the port lock into any USB port on the computer (this authorizes CAM2 Q to operate). If you have a printed license key, you will not need a port lock.
2. Connect the FaroArm to the computer using the USB cable.
3. Connect the FaroArm to the Power Supply cable.
4. Using the ***On / Off*** switch, turn the device on.



For safety reasons, complete all cable connections before applying power to the computer and the FaroArm.

Starting CAM2 Q

You can start CAM2 Q using any of the following methods:

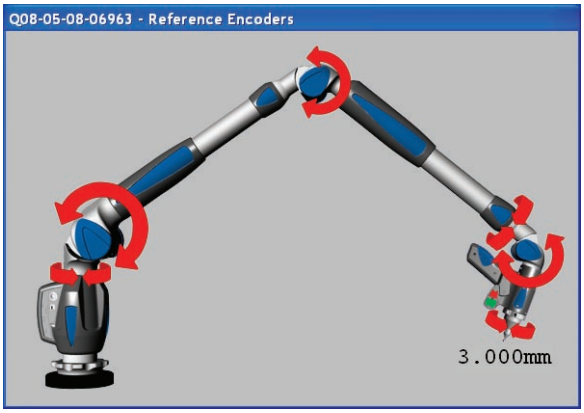
- Double left-click the ***Q*** icon on the Desktop; or
- Click the ***Start*** button, point to ***Programs***, point to ***FARO***, and then click ***CAM2 Q***; or
- Double left-click any previously saved CAM2 Q file.

User Profile

The very first time you start CAM2 you must choose a default profile. User profiles contain custom screen layout settings. You can change the default profile any time. On the ***Edit*** menu, select ***User Profiles***.

Referencing the FaroArm Encoders

Each joint of the FaroArm contains encoders. The FaroArm electronics must locate and recognize (reference) each encoder before you can collect data. This process is known as "referencing the encoders". Red arrows in the ***Reference Encoders*** window indicate that the encoders have not been referenced and you must rotate each axis until each warning clears.



You must properly connect the FaroArm to the computer running CAM2 Q to see the reference window.

When referencing the encoders and measuring, you may see the End Stop/Stress Warning window. This means that an encoder has reached one end of its rotation, or there is stress on one of the tubes or encoder joints.

Device Control Panel

The ***FARO Device Control Panel*** shows the connected device information:

The elements of the FaroArm ***Device Control Panel*** are:

- Add New Device
- Selected Device
- Probe information:
  - ➡ Current Probe
  - ➡ Calibration Error
  - ➡ Calibration Date
  - ➡ Temperature at Calibration
  - ➡ Probe Management
- Setup information:
  - ➡ Probe Calibration
  - ➡ Hardware Configuration
  - ➡ Diagnostics
  - ➡ Temperature
  - ➡ Measurement Mode Setting

To open the panel, click ***Device Control Panel*** on the ***Device*** menu, or press the ***P*** hot key. You can close this panel by clicking the ***X*** in the upper right corner of the panel.

FaroArm Buttons

The FaroArm has four buttons on the handle:

- The green ***FRONT*** buttons, nearest the probe, collect data.
- The red ***BACK*** buttons, furthest from the probe, accept data.

The buttons on either side of the handle accommodate left-handed and right-handed users.

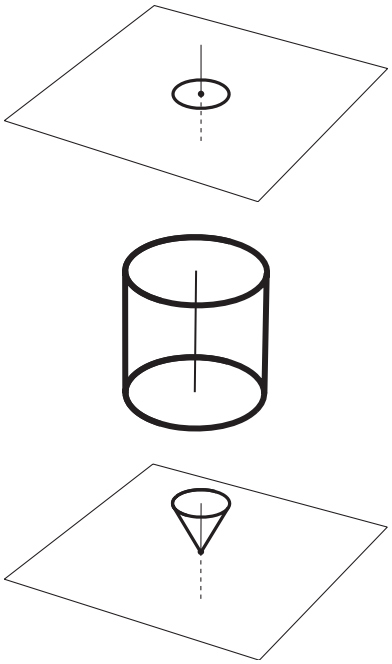
When you press a button, the LED next to that button lights and the computer sounds, to indicate the collecting or accepting of data.

Lesson 2: CAM2 Q Overview

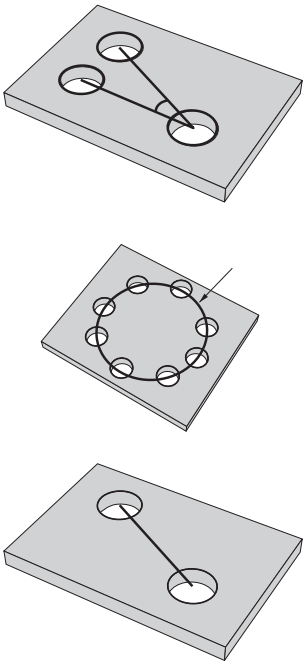
Feature-Based Measurement

CAM2 Q is feature-based measurement software. A feature is a geometric attribute or element of a part. You can directly measure some features (e.g. a circle, sphere, cone); others you cannot directly measure easily, accurately or repeatedly (e.g. an angle, bolt hole circle, length), and as a result you must construct or dimension them from elements that you can directly measure. CAM2 Q enables direct measurement, constructions, and dimensions. The ***Measure***, ***Construct***, and ***Dimension*** menus reflect these capabilities.

Features you can directly measure



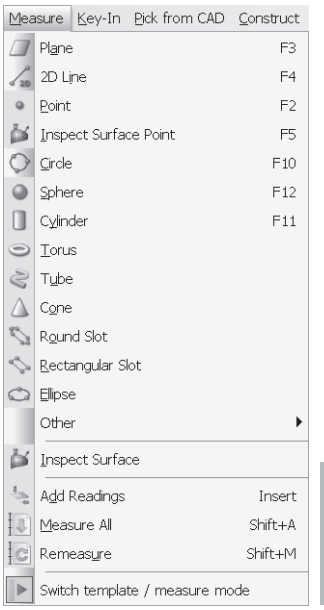
Features that you must construct

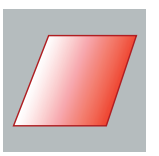


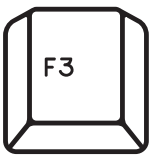
Commands in CAM2 Q – Methods to Access

You can access many commands using hot keys, which are a single keystroke method of choosing a command or accessing a panel. There is a list of all hot keys in the Appendix for your convenience.

Access commands in CAM2 Q either by menus, toolbar buttons or hot keys.



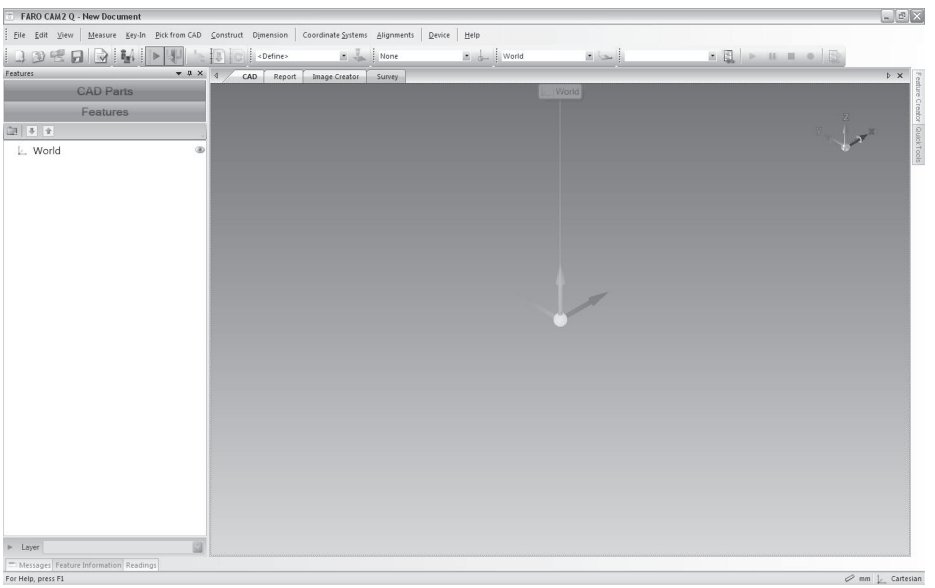




Screen Layout

Main Window

This area of the CAM2 Q screen shows your coordinate system, the results of your measurements, and any imported CAD files. When you first view the screen, there is an XYZ axis representing the three dimensions, which uniquely identifies the locations of points you collect.



Pull-Down Menus

The menu commands at the top of the screen, ***File***, ***Edit***, ***View***, ***Measure***, ***Key In***, ***Pick from CAD***, ***Construct***, ***Dimension***, etc. are pull-down menus which contain a variety of commands.

Control Panels

Control Panels are fields that you can bring to the front of the screen containing a variety of information. CAM2 Q includes many panels.

On the ***View*** menu, click ***Panels***. The following panels are the most common panels you will use in this training.

Features:	Displays the list of features you ***Measure***, ***Key In***, ***Pick from CAD***, ***Construct*** or ***Dimension***
Feature Information:	Displays the tolerances and nominals you input for specific features
Readings:	Displays the results for features you have measured
Feature Creator:	Enables you to select the features you want to measure immediately or at a later time
QuickTools:	Enables you to record, edit, and play a part inspection program

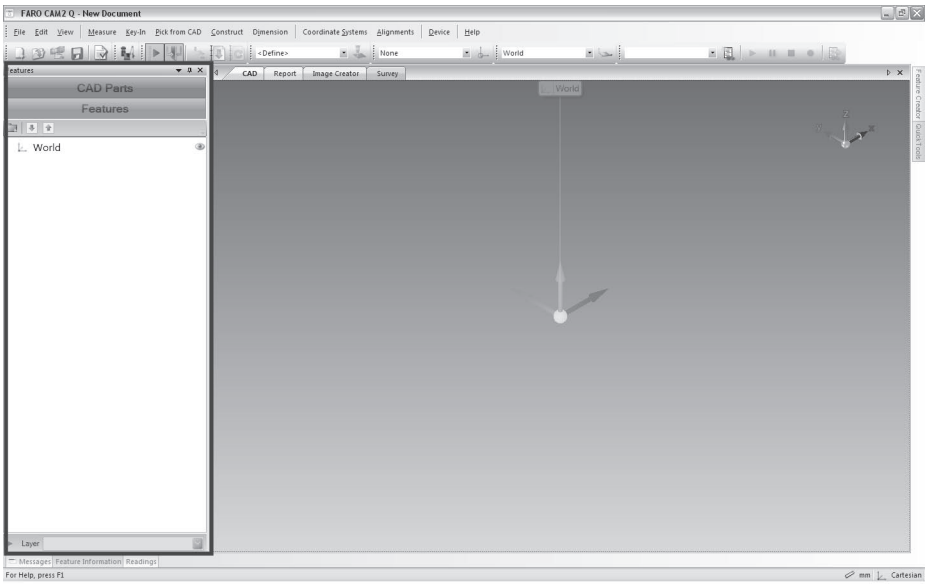
Some of these Control Panels are attached to the screen boundaries and are in "Auto Hide" mode. Look for a tab with the title of the Control Panel and click it to show that Control Panel. The Control Panel remains visible for a short time and automatically collapses back to the tab as you move to another area of the screen.

Click the AUTO HIDE button  in the upper right corner of the panel to keep the bar visible.

Click the CLOSE button  in the upper right corner to close a panel.

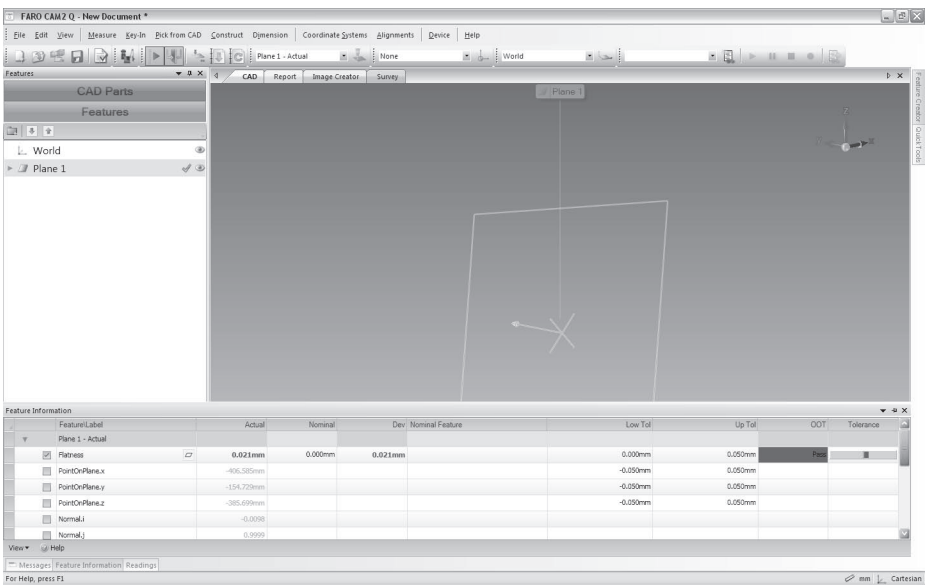
Features Panel

CAM2 Q provides a ***Features*** panel that shows all the features you measure, construct, or import with a CAD model. In this panel, the features are listed in chronological order of their creation.



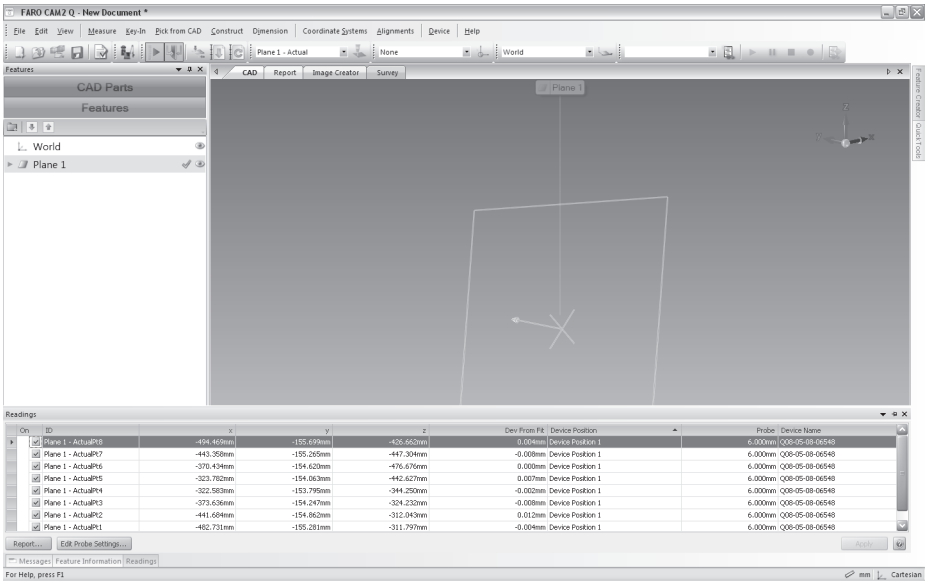
Feature Information Panel

Click the ***Feature Information*** tab at the bottom of the screen to show the ***Feature Information*** panel. CAM2 Q provides a ***Feature Information*** panel that shows tolerance values for an individual, or selected group, of features. You can choose which value of a feature to tolerance and edit the tolerance values.



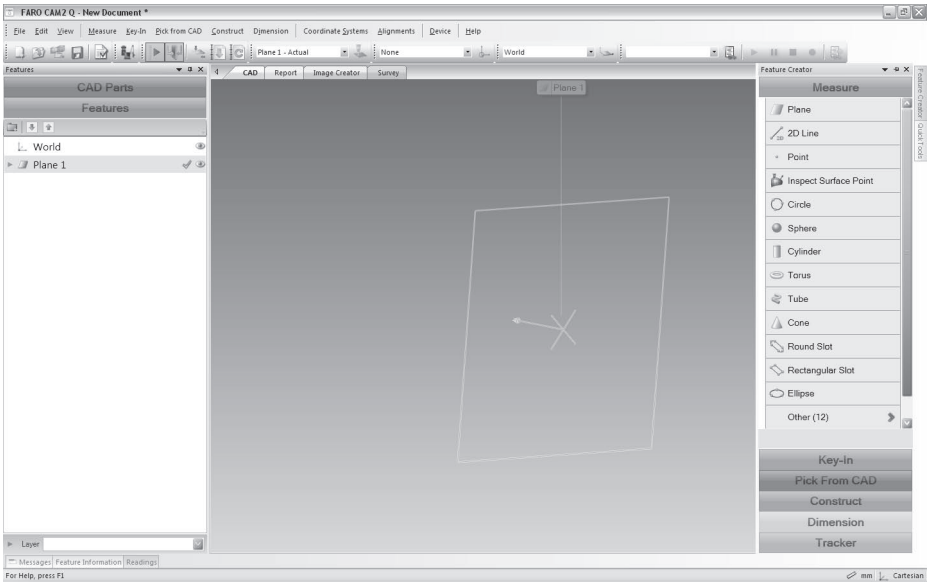
Readings Panel

Click the ***Readings*** tab at the bottom of the screen to show the ***Readings*** panel. CAM2 Q provides a ***Readings*** panel that shows a list of measurement reading and properties for a selected feature.



Feature Creator Panel

Click the ***Feature Creator*** tab at the right of the screen to show the ***Feature Creator*** panel. CAM2 Q provides a ***Feature Creator*** panel that shows feature creation commands grouped by ***Measure***, ***Key In***, ***Pick From CAD***, ***Construct***, ***Dimension***, and ***Tracker*** headings. This panel provides a quick method for creating features without using the menu commands.



These are the key elements of the CAM2 Q screen layout. This workbook guides you through the most commonly used menu items and commands

and prepares you to explore the full capabilities of the CAM2 Q software using the ***Help*** menu.

Customizing the Panels Layout

You can change the position of the panels on the screen. To move a panel, left-click and hold the title bar of any of the panels and drag the panel with the mouse. Once you start dragging a panel, the screen prompts you with icons to show where you can place the panel.

Creating a User Profile (Saving a Customized Screen Layout)

You can easily save a particular screen layout so you can easily recall it. Follow the instructions below:

1. On the ***Edit*** menu, click ***User Profile***.
2. When the ***User Profile Selection*** dialog box appears, click ***Save New Profile***.
3. Name your profile and click ***Close***.

This saves your current screen as a ***User Profile***. You can save any number of user profiles. For example, you can have many users, each with their own user profile, using the same computer. This can save time because each user does not have to setup the screen layout each time CAM2 Q opens.

Selecting/Activating a New User Profile

1. To select and activate a saved User Profile, click ***Profile*** on the ***Edit*** menu.
2. From the ***User Profile Selection*** dialog box, select the ***User Profile*** you want to load, and then click ***Load Profile***.

CAM2 Q Help File

The CAM2 Q Help file is the first place you can look for information.

- On the ***Help*** menu, select ***Contents***.
- Use the ***Contents***, ***Index*** and ***Search*** tabs, on the left side of the window, to navigate around the file and select and search for topics.
- Each time you receive an update to CAM2 Q, you should view the *What is New* topic. This topic describes new functions and commands that have been added to CAM2 Q, as well as the enhancements made to existing commands. The *What is New* topic is located at the end of Chapter 1: Introduction to CAM2 Q.

Module 2:
Measurement Considerations

Before you start measuring, you have to make sure outside factors such as temperature, vibrations, and mounting instability do not compromise your data. You also have to make sure that you calibrate the probe prior to every measurement session. In this module, you will learn how to compensate for environmental factors and prepare for measurement to get the accurate data that you need.

MODULE OBJECTIVE

You will be able to:

- Evaluate environmental variables that impact measurement accuracy and take the appropriate steps to avoid, minimize, or correct the impact.
- Identify when to use each probe calibration method and perform a probe calibration.

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Lesson 1: Environment

In addition to the calibration accuracy of your device, environmental factors play a major role in the accuracy of your measurements.

Temperature Effect On a Part

Changes in temperature may cause a part to expand or contract, affecting measurement accuracy.

Environment vs. Part Temperature: If the part is warmer or cooler than the surrounding environment, it will expand or contract while being measured. Measurements taken earlier may not equal measurements taken later. To minimize the impact of air temperature on the part:

- Let a part "soak" in the inspection environment to adjust to the air temperature prior to taking measurements. Generally, the larger the part, the longer it will take to adjust.
- Avoid taking measurements where the air temperature will vary widely; e.g. don't take measurements where doors will be opened to warmer or cooler air (near a furnace, garage door, etc.)

Software Temperature Compensation: CAM2 Q can compensate for the temperature of a part. The ability to scale for temperature allows you to measure parts adjusting for thermal expansion or contraction. For details on how to do this, look in the CAM2 Q ***Help*** for Material Temperature.

Temperature Effect On the Device

The FaroArm compensates for temperature variations.

Environment vs. Device Temperature: The FaroArm compensates for temperature variations; however, if the difference in temperature between the FaroArm and the air results in the FaroArm changing more than three degrees Celsius within five minutes, a warning appears on the computer screen. At this point, to ensure accuracy, you cannot take any readings until the FaroArm temperature has stabilized.

To avoid warnings: Always let the FaroArm temperature stabilize before taking measurements. As described above, avoid places where there will tend to be wide temperature variations.

Light: Light, especially sunlight, generates heat. When exposed to light unevenly, a large part can heat up and expand or contract unevenly. To ensure accurate readings, make sure the part you are measuring has even exposure to light.

Movement

Movement always affects accuracy. Movement resulting in no change in the FaroArm's position relative to the part will have the least impact on measurement accuracy. To avoid the impact of movement choose a measurement spot where the part and device can be fastened and stationary.

Vibration Effects

Vibration can cause inaccuracies. People putting heavy parts on your surface plate, forklifts driving by, and stamping machines are some of the causes of vibration.

Mounting Stability

Mounting stability is key to accurate measurement.

Tripods

FARO sells two types of tripods: Folding and Brunson. While tripods provide mobility and allow multiple device positions, you must be careful to ensure mounting stability. For detailed instructions, refer to the manual that came with the tripod you purchased.

Magnetic Mount

The Magnetic Mount is another portable option when mounting to most metal surfaces. To ensure proper mounting with the Magnetic Mount, follow the detailed instructions provided with the mount.

Vacuum Mount

You can use the Vacuum Mount to secure the FaroArm to a granite or machined metal surface. To ensure stability with the vacuum mount, follow the detailed instructions in the Accessory Manual provided with your device.

C-Clamps

You can use C-clamps to secure the FaroArm to a variety of surfaces. When using C-clamps to increase stability:

- Make sure you arrange the clamps on opposite sides of the base - 180 degrees from each other. Less than 180-degree separation creates the risk of flexing the bottom surface off the table/plate, which could cause the FaroArm to pivot on the plate.
- Clear the surface of any debris which can cause the base to rock.

Custom Plates

You can use custom plates to secure the FaroArm when standard mounts are not practical. To ensure stability with custom mounts:

- Make sure the plate is large enough to mount the FaroArm using the Magnetic Mount or C-clamps.
- Make sure the plate is thick enough to prevent flexing.
- Make sure to mount the plate securely to the part/fixture.
- While using the FaroArm, use a dial indicator to determine if the mount is flexing. If it is, use a thicker plate.

Lesson 2: Device

In addition to the environment and the device mounting, using proper methods and techniques to calibrate the probe ensures system accuracy.

Probe Calibration

The FaroArm collects data by touching the part with a probe attached to the end of the handle. Once the probe is attached, you must determine the X, Y, Z location of the probe tip, relative to the FaroArm's coordinate system, prior to measuring. Since all data is taken at the center of the probe, we have to "find" that center before measuring.

Check for proper probe installation and check the condition of the probe for cracks or chips.

Calibrate the probe prior to every measurement session; or, if you are using several probes, calibrate each probe any time you switch. While it is true that a straight probe can be removed and replaced very accurately, it is still good practice to perform the probe calibration.

Probe calibration is the most critical task performed during any measurement session. If the probe calibration passes, the measurements will be accurate. If the probe calibration fails, the measurements will not be accurate.

There are two acceptable methods to calibrate a probe:

- the ***Single Hole Calibration*** method, which is recommended for standard ball probes, and
- the ***Sphere Calibration*** method, which is recommended for touch trigger probes and point probes.

In this lesson, we will present the Single Hole Calibration method.

XYZ Location

The last axis of the FaroArm has its own coordinate system. The location of the center of the ball probe will be reported in this coordinate system. Using the same probe, the FaroArm should be able to repeat this value with better results than the single-point accuracy of the device.

Probe Calibration Error

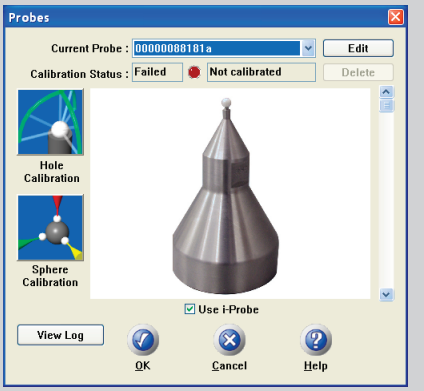
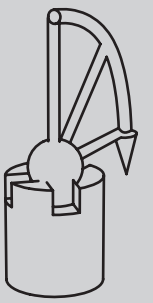
A PASS condition indicates a result with a low calibration error ((max-min)/2 value equals or below device specification). A FAIL condition indicates a result with a high calibration error ((max-min)/2) value above device specification).

Single Hole Calibration Method

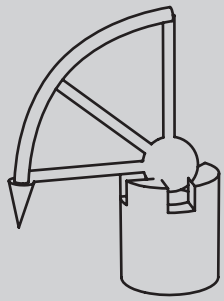
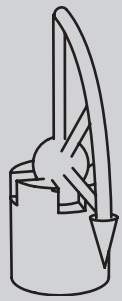
The Single Hole calibration is performed using the FARO probe calibration cone or a 5mm machine drilled hole. The hole does not have to be exactly 5mm, but it must be smaller than the probe's diameter with a smooth seat. All of the points in this method will be collected by holding down the ***FRONT*** button. The FaroArm will collect points as fast as possible (scanning) until you release the ***FRONT*** button.

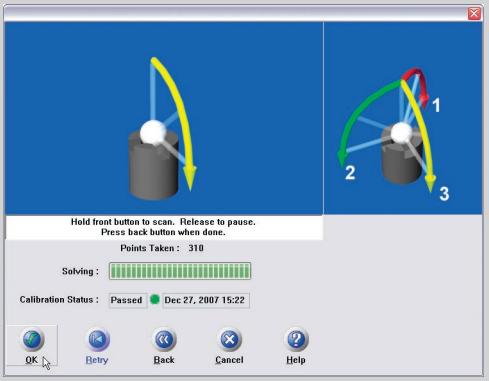
ACTIVITY: Probe Calibration

For this activity, you will calibrate a probe using the ***Single Hole Calibration*** method.

1. ***Probe Management***	Click the ***Device Control Panel*** icon. On the ***Device Control Panel***, click the ***Probe Management*** button. 
2. Choose probe	In the ***Current Probe*** list, select ***6mm Ball Probe***.
3. Guidance off	Click the ***Edit*** button and make sure that the ***Guidance*** check box is clear.
4. Start ***Hole Calibration***	Click the ***Hole Calibration*** button. 
5. Position #1	Place the ball probe in the cone in the upright position #1. 
6. Start point collection	Press and hold the ***FRONT*** button.

For best results, keep the elbow of the FaroArm on the same side for the entire calibration process.

7. Sweep down	Sweep the handle down to horizontal position. Be sure that the ball probe remains seated in the hole. Release the ***FRONT*** button. CAUTION: The probe must be well-seated in the hole when digitizing all calibration points. Even one or two poorly digitized points significantly affects the optimization process, which in turn effects the accuracy of the FaroArm.
8. Position #2	Place the ball probe in the cone in the upright position #2. 
9. Start point collection	Press and hold the ***FRONT*** button.
10. Sweep down	Sweep the handle down to horizontal position. Be sure that the ball probe remains seated in the hole. Release the ***FRONT*** button.
11. Position #3	Place the ball probe in the cone in the upright position #3. 
12. Start point collection	Press and hold the ***FRONT*** button.
13. Sweep down	Sweep the handle down to horizontal position. Be sure that the ball probe remains seated in the hole.
14. End point collection	Press the ***BACK*** button.

15. View results before clicking ***OK*** or ***Retry***	Check the Calibration Status. ● If Calibration Status = Passed, click ***OK***. ● If Calibration Status = Failed, click the ***Retry*** button and repeat the calibration.  ☛ *Ensure that you have collected at least 200 points.*

Single Point Articulation Test (SPAT)

The Single Point Articulation Test (SPAT) test checks the repeatability of the FaroArm's X, Y and Z coordinates. Refer to the FaroArm Help File for the procedure.

FaroArm Help File

You can access the FaroArm Help file by clicking the ***Help*** button in any FaroArm driver command. These commands are found in the ***Device Control Panel***.

- On the ***Device Control Panel***, click the ***Probe Management*** button, or any other command button.
- In the ***Probes*** dialog box, click the ***Help*** button.

This opens the FaroArm help file and automatically jumps to the Probes section of the file. Use the ***Contents***, ***Index*** and ***Search*** tabs, on the left side of the window, to navigate around the file.

Lesson 3: FaroArm Device Setup Checklist

Temperature Effect On A Part

- ❒ Part has been allowed to soak in the inspection environment to adjust to the air temperature.
- ❒ Measurements will be taken where the air temperature is relatively constant.
- ❒ CAM2 Q is set to appropriately adjust for part expansion/contraction (see software user guide).
- ❒ The part is evenly exposed to light.

Temperature Effect On The Device

- ❒ The device temperature has been allowed to stabilize.

Movement

- ❒ The device and part are fastened and stationary.

Vibration Effects

- ❒ Vibrations (forklifts driving by, stamping machine movement etc.) have been avoided.

Mounting Stability

- ❒ Detailed instructions provided with mount (tripod, magnetic, or vacuum) were carefully followed.

If Using C-Clamps:

- ❒ The surface is clear of any debris which can cause the base to rock.
- ❒ Clamps are arranged on opposite sides of the base - 180 degrees from each other.

If Using A Custom Mount:

- ❒ The plate is large enough to mount the FaroArm using the Magnetic Mount or C-clamps.
- ❒ The plate is thick enough to prevent flexing.
- ❒ The plate is securely mounted to the part/fixture.
- ❒ A dial indicator has verified the rigidity of the plate.

CAM2Q

Module 3:
Basic Measurement

In this module, you will become familiar with the CAM2 Q Measure, Construct, Dimension, and Report menus—the four steps common to every CAM2 Q measuring job.

You will also be introduced to a Basic Measurement Strategy, which will get you thinking about what you need to include in the final report and how you can obtain that information *before* you begin measuring. You will see the process of how you think about a project vs. how you actually do a project.

This module will first focus on each of the key menus. After you learn how to use each menu, you will apply the Basic Measurement Strategy to put them all together in the proper order to complete a measurement job.

Let's begin!

MODULE OBJECTIVE

You will be able to:

- Use the Basic Measurement Strategy to plan and complete a measurement job using the ***Measure***, ***Construct***, ***Dimension***, and ***Report*** menus.

LESSONS

CAM2Q

08m13e14 - CAM2 Q v1.5 - FaroArm Training Workbook - October 2010.pdf Part#12052-001

SCENARIO: FIRST ARTICLE INSPECTION

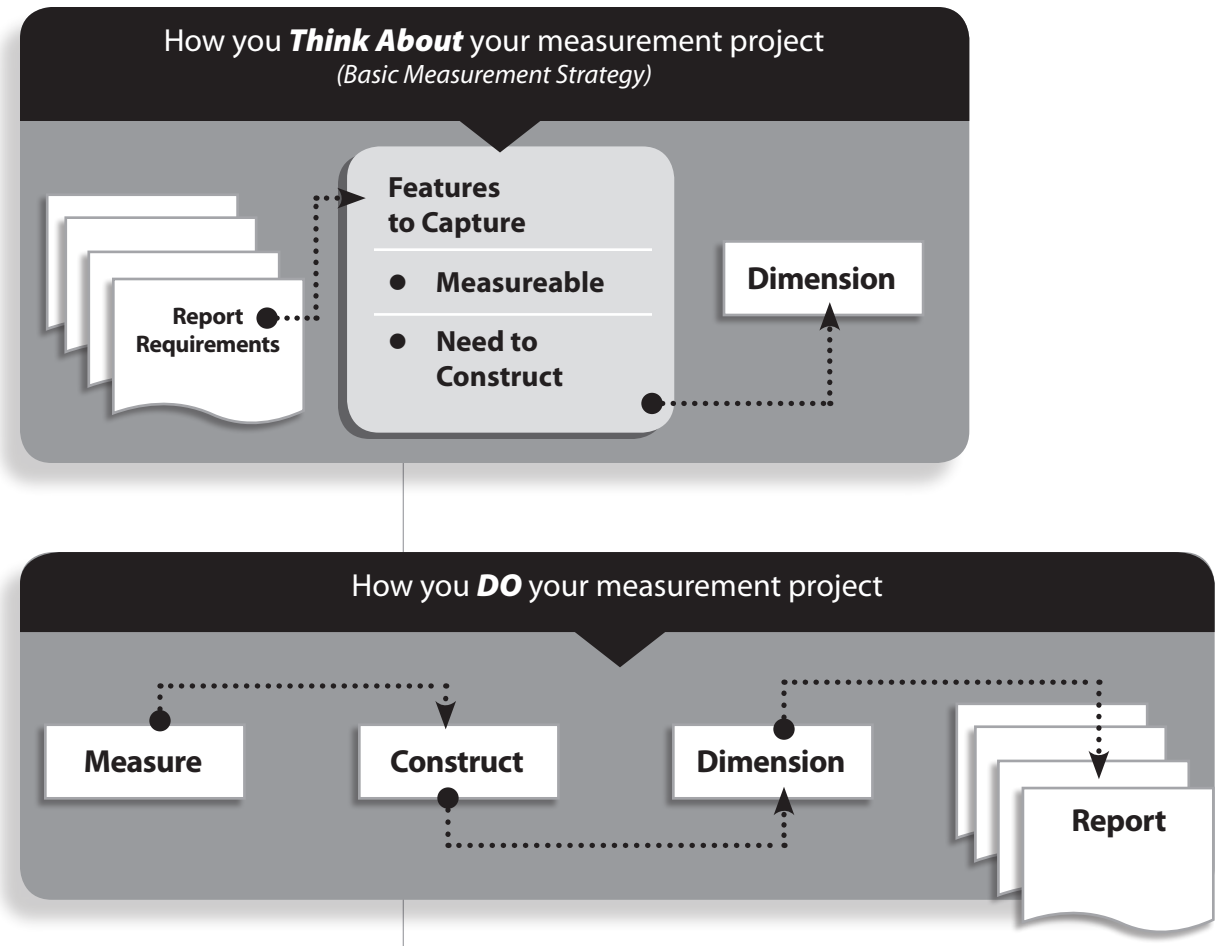
You enter your boss' office. The boss says, "Using our new FaroArm and CAM2 Q, I need you to do a first article inspection on this part. We need to finish our part inspection by the end of the day if we are going to meet our delivery schedule. Here is the blueprint… get to work." You return to your workstation, look at the blueprint and the part, and think about how to tackle this job.

"What is the most effective way to provide what the boss is asking for? I need to think about the measurement strategy and begin with the end in mind."

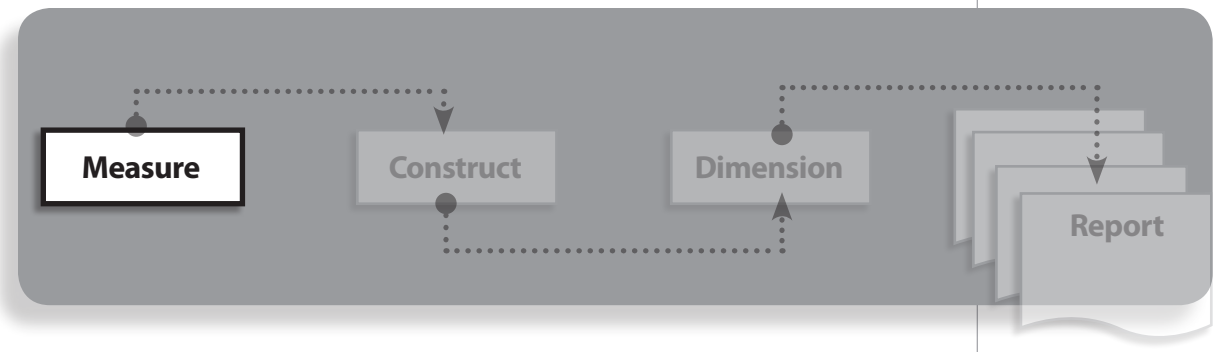
Strategy: Basic Measurement

Look at what you need to measure and consider the following questions:

- What should you include in the formal ***Report***?
- Which features (geometric elements) do you need to capture?
  - ➡ Which features can you ***Measure*** directly (e.g. planes, circles, cylinders, lines, etc.)?
  - ➡ Which features do you need to ***Construct*** (e.g. bolt hole circle)?
- Which ***Dimensions*** do you need? (e.g. circle center to circle center)
- What logical sequence should you use to move around the part and capture the features? (i.e., left to right, top to bottom, or measure all planes first, then circles, etc.)


How you **Think About** your measurement project
(Basic Measurement Strategy)
Report Requirements
Features to Capture
Measureable
Need to Construct
Dimension
How you **DO** your measurement project
Measure
Construct
Dimension
Report

Lesson 1: Measure



Before you can apply the ***Basic Measurement Strategy*** to a complete measurement job, you need to know the capabilities of the CAM2 Q menus (Measure, Construct, Dimension, Report). The first menu we will present is ***Measure***. You will learn to measure features correctly, analyze the data for the feature, and find and edit feature readings and names.

Lesson Objective

With the assistance of the ***Help*** menu, you will be able to use all of the functions within the ***Measure*** menu. This means you will be able to:

- Use the ***Measure*** menu to capture data.
- Review and evaluate measurement quality and edit data.
- Define projection planes for 2D measurements.
- Save and open files.
- Rename features.
- Determine the optimal number of points to collect and distribution of those points on a feature.

Before Measuring

Before you begin measuring features, make certain that you use the correct probe. On the ***Device*** menu, click ***Device Control Panel***. Check the ***Current Probe*** listed in the ***Probe*** section. If you need to change the probe, click the ***Probe Management*** button.

Once you determine that you are using the correct probe, make sure to calibrate the probe.

How To Measure Features

1. Choose a Part Measurement Workflow
2. Add Feature(s) to your file
3. Record Measurements

Part Measurement Workflow: Measure or Template Mode

The default mode is Measure Mode.

There are two workflows for measuring your part in CAM2 Q. Sometimes you may want to take measurements immediately as you are inspecting a part (***Measure Mode***); other times you may want to plan an entire measurement process before you actually take any measurements (***Template Mode***).

Measure/Template Mode Icon

In CAM2 Q, you can choose either approach by selecting or deselecting the ***Measure/Template Mode*** command. Select (highlight) the icon, or ***Measure*** menu command, for ***Measure Mode***, and deselect for ***Template Mode***. When ***Template Mode*** is active, a red border and "Template Mode" text appears on the Main Window.

Measure Mode

To measure features in ***Measure*** mode:

1. On the toolbar, select the ***Measure/Template Mode*** icon.
2. On the ***Measure*** menu, or from the ***Feature Creator***, click the feature you want to measure (e.g. a plane).
3. The ***Measurement Window*** appears and you can start adding readings (measurements).

Template Mode

The sequence in which you click the features is the sequence in which you are prompted to measure the features.

In ***Template*** mode, you first create the features that you will want to measure. To create these features:

1. On the toolbar, deselect the ***Measure/Template Mode*** icon.
2. On the ***Measure*** menu, or in the ***Feature Creator*** panel, select the feature you want to measure (e.g. a plane). The feature is added to the ***Features*** panel. The feature name appears in gray, indicating that the feature has not been measured and is not "solved" yet.

To measure in the ***Template*** mode:

1. In the ***Features*** panel, select an existing feature to measure.

2. You can select features one at a time, or, to select multiple features, press the ***CTRL*** key on the keyboard while clicking each feature with the mouse.

3. On the toolbar, click the ***Add Readings***  icon.

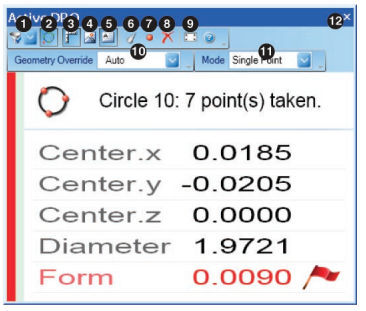
4. The ***Measurement Window*** appears and you can add readings (measurements) to each feature.

Measurement Window

The ***Measurement Window*** panel appears each time you choose a measurement command either from the ***Feature Creator*** panel, the drop-down menus, or by right-clicking a feature in the ***Features*** panel and selecting ***Add Readings***.

The ***Measurement Window*** consists of three panels:

- ***Prompt***—displays prompts in conjunction with the DRO.
- ***Device's Digital ReadOut (DRO)***—displays the position of the probe at any point in space using a three dimensional coordinate system and displays the Form.
- ***Image***—displays the image for the current measurement.



The ***Measurement Window*** panel consists of Setup, Measure, Sizing, and Scanning commands:

Setup

❶ ***Show Options:*** Allows you to select the coordinate system axes you want to display.

❷ ***Geometric Data:*** Geometric data and statistics associated with the current measurement.

❸ ***Digital ReadOut (DRO):*** The current location of the measurement device's probe that updates as the probe moves through space. If there is a communication or device error all of the coordinate values display as a line of asterisks (*****).

❹ ***Measurement Image:*** The image for the current measurement (QuickTools only).

❺ ***Show Prompt:*** The text area that contains informational messages relating to the current measurement.

Measuring

❻ ***Record Measurement:*** Enables you to take a single point reading.

❼ ***Record End Click:*** Completes the collection of readings.

❽ ***Remove Last Reading:*** Enables removal of the last reading taken.

Sizing

❾ **Full Screen**: Maximizes the Measurement Window to fill the entire computer screen.

Geometry Override

❿ **Geometry Override**: Allows you to override CAM2 Q's Automatic Feature Recognition by manually selecting a geometry type.

Scanning

⓫ ***Mode:*** Determines the way in which you collect readings. Single Point (the default setting) collects one reading per click of the ***FRONT*** button. Time Interval and Distance Interval are different methods of scanning points, which means that points are continuously taken as long as the ***FRONT*** button is pressed.

Recording Readings

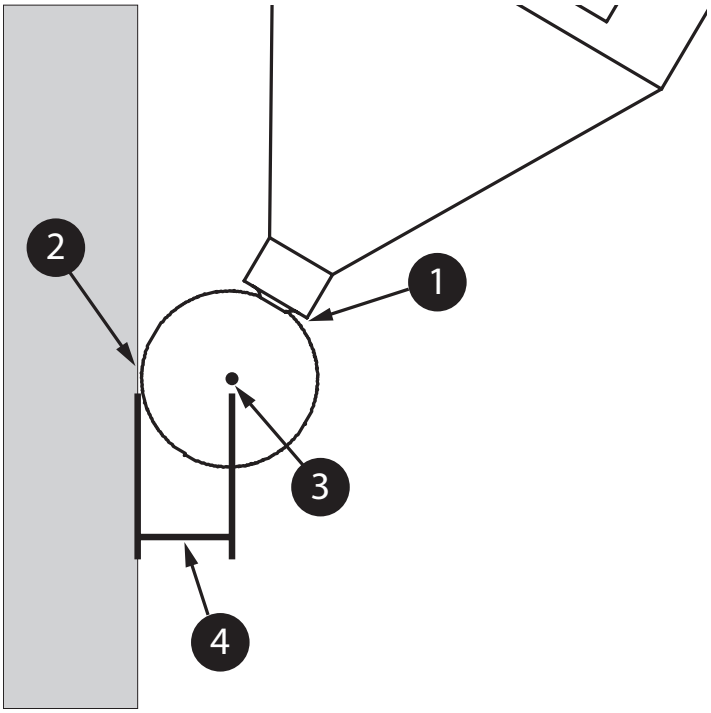
Click the green ***FRONT*** button to collect point readings. Click the red ***BACK*** button to accept the collected readings, which is the End Click.

The position of the probe at the End Click is critical for accurate readings.

Probe Compensation

When measuring with a ball probe ❶, the location of the center of the probe records as a point each time you press the ***FRONT*** button. Because each probe has a known diameter, there is a known distance between the center of the probe and the outer edge of the probe. This distance needs to be specified (projected) for the point to record in the correct location.

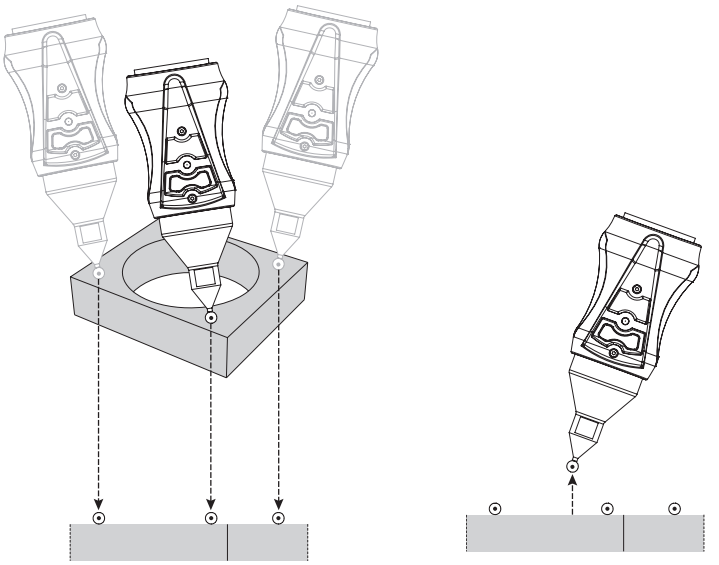
The distance between the probe point of contact ❷ with the surface being measured and the center of the probe ❸ is known as Probe Offset ❹. This transfer of the point from the center of the probe to the correct location is known as probe compensation.



To make sure the compensation for the probe radius calculates correctly, the location of the probe is extremely important when you press the BACK button.

Plane Compensation

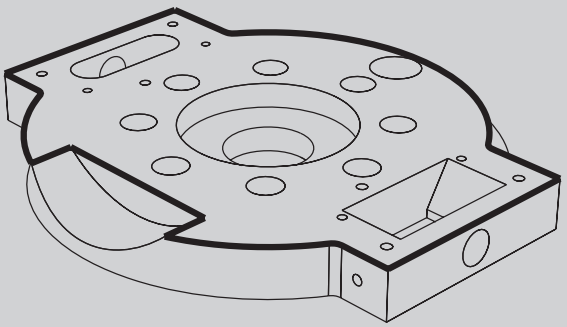
In this example, you are taking three points to identify a plane on a part. This plane can compensate in one of two opposite directions. The correct compensation depends on where you press the ***BACK*** button. In this case, press the ***BACK*** button above the plane. The plane then compensates in a downward direction the distance equal to the probe radius.



In this example, the correct location to accept points is above the plane.

ACTIVITY: Measure A Plane In The Measure Mode

For this activity, you will measure a plane on the top of the demo part with at least three points and review the Form data.



1.	Select Workflow Mode	If necessary, click the ***Measure/Template Mode*** icon so that you are in ***Measure*** mode.
2.	Select ***Plane***	On the ***Measure*** menu, click ***Plane***.
3.	Collect readings	Use the ***FRONT*** button to collect at least three readings.
4.	Compensate probe	Pull the probe up and away from the part when you finish collecting readings.

5. Accept readings	Press the ***BACK*** button to accept readings.
6. Cancel the measurement command	Press the ***BACK*** button again to cancel the ***Plane*** command.

When measuring a plane, probe location during the End Click is critical for correct placement of the plane.

7. View screen	On the ***View*** menu, click ***Zoom***, and ***All***. Or press the ***E*** hot key.

Now, remeasure the plane with more than three readings and review the Form data in the Measurement window.

8. ***Remeasure*** command	On the ***Measure*** menu, click ***Remeasure***. Use the ***FRONT*** button to collect more than three readings
9. Compensate probe	Pull the probe up and away from the part when you finish collecting readings.
10. Accept readings	Press the ***BACK*** button to accept readings.

Percentage Of Measured Feature Vs. Total Feature Size

Measuring a small percentage of a feature does not provide the most accurate representation of the feature. Since CAM2 Q constructs a best-fit feature from the points you collect, the number of points collected and the spacing of those points directly impact the accuracy of the measurement result.

A common principle applies regardless of the feature you measure. The more points you collect, and the more evenly you distribute them over the part, the more repeatable your measurement result will be. For optimal results, we recommend collecting two times the minimum points plus one. (Minimum number of points x 2 + 1).

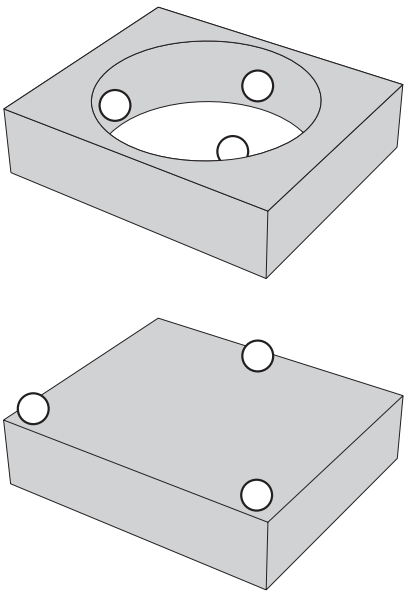
When considering the distribution of the points over the part, remember that even distribution is best. If there are times when only a portion of the feature is accessible, then take as many points as you can on that portion. While this is not the ideal situation, taking as many points as you can will help ensure a better representation of the feature.

Feature	To solve the feature: minimum number of points to collect	For optimal results: suggested minimum number of points to collect
 Plane	3	7
 Line	2	5

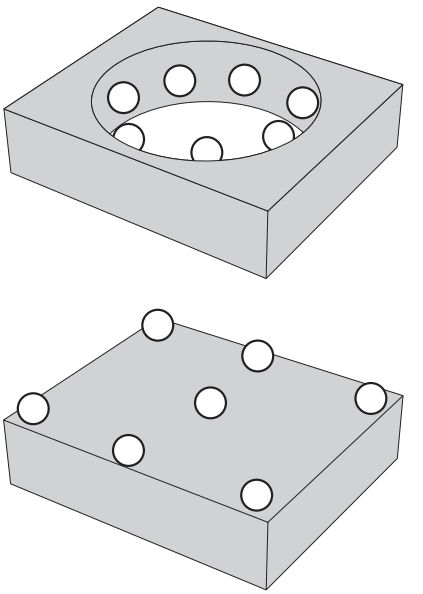
Circle	3	7
Sphere	4	9
Cylinder	6	13
Cone	6	13

Examples:

Minimum



Minimum: For Optimal Results



ACTIVITY: Measure Features In The Measure Mode

For this activity, you will add the following features to the ***Features*** panel to measure later:

- Four circles
- One sphere

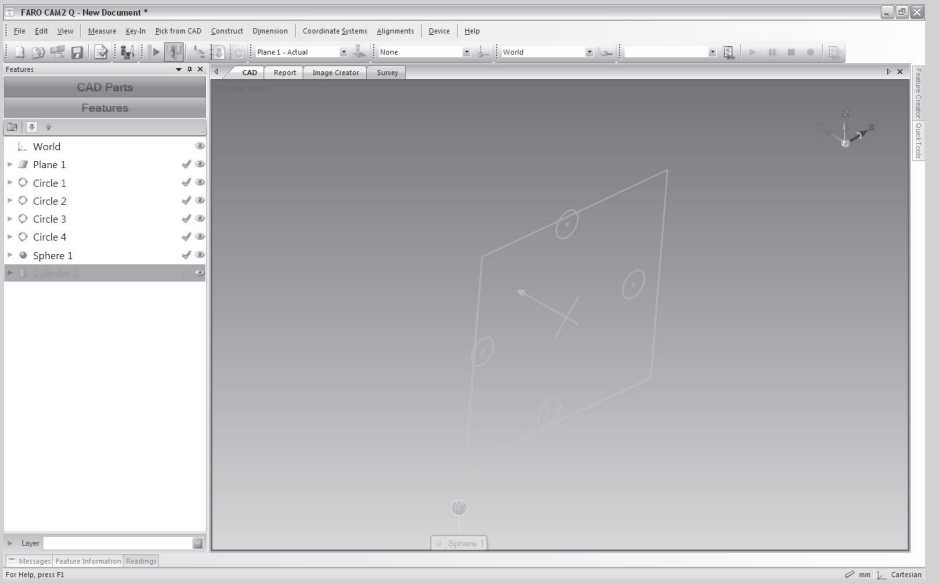
1. Select Workflow Mode	Click the ***Measure/Template Mode*** icon to switch to ***Measure*** mode. 

For the next steps, either click each feature on the ***Measure*** menu or click a feature from the ***Feature Creator*** panel.

Hot Keys
F10:
Measure Circle

F12:
Measure Sphere

2. Select ***Circle***	On the ***Measure*** menu, click ***Circle***.
3. Collect readings	Use the ***FRONT*** button to collect at least three readings on Circle 1.
4. Compensate Probe	Pull the probe to the center of the Circle 1 when you finish collecting readings.
5. Collect readings	Use the ***FRONT*** button to collect at least three readings on Circle 3.
6. Compensate Probe	Pull the probe to the center of the Circle 3 when you finish collecting readings.
7. Collect readings	Use the ***FRONT*** button to collect at least three readings on Circle 5.
8. Compensate Probe	Pull the probe to the center of the Circle 5 when you finish collecting readings.
9. Collect readings	Use the ***FRONT*** button to collect at least three readings on Circle 7.
10. Compensate Probe	Pull the probe to the center of the Circle 7 when you finish collecting readings.
11. Cancel Circle Command	Press the ***Back*** button to cancel the Measure Circle command.
12. Select ***Sphere***	On the ***Measure*** menu, click ***Sphere***.
13. Collect readings	Use the ***FRONT*** button to collect at least three readings on Sphere A.
14. Compensate Probe	Pull the probe away from the surface of Sphere A when you finish collecting readings.
15. Cancel Sphere Command	Press the ***Back*** button to cancel the Measure Sphere command.



ACTIVITY: Add Features To The Features Panel In The Template Mode

Hot Keys
F11:
Measure Cylinder

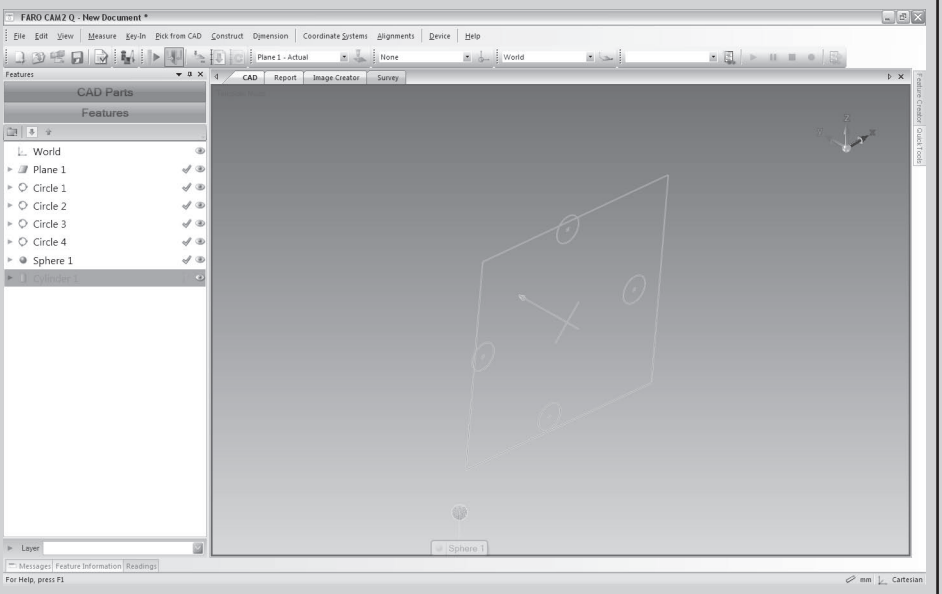
For this activity, you will add the following features to the ***Features*** panel to measure later:

- One cylinder

1. Select Workflow Mode	Click the ***Measure/Template Mode*** icon to switch to ***Template*** mode. 

For the next steps, either click each feature on the ***Measure*** menu or click a feature from the ***Feature Creator*** panel.

2. Add cylinder	Click ***Cylinder*** once to add this feature to the ***Features*** panel.



ACTIVITY: Measure Feature From The Features Panel

For this activity, you will now add measurements to the cylinder features you just added on the Features panel:

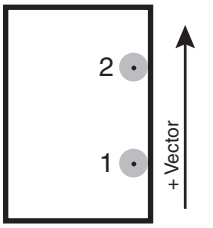
- Cylinder 1

1. Select feature names	On the ***Features*** panel, click ***Cylinder 1*** added in ***Template*** mode.
2. Select ***Add Readings***	Right-click on the highlighted features, and click ***Add Readings***.
3. ***Measure Mode***	Click the ***Yes*** button to change to ***Measure*** mode.

Measure the cylinder when prompted in the ***Prompt*** panel of the ***Measurement Window*** (i.e., "Collecting readings on Cylinder 1").

4. Measure Cylinder 9	For the cylinder, use the ***Front*** button to collect at least six readings.
5. Compensate and accept readings	Pull the probe to the center of the cylinder. Press the ***Back*** button to accept the readings.

When measuring a cylinder, the first two readings determine the direction of the cylinder.



Renaming a Feature

To rename a feature:

- Double-click a feature on the ***Features*** panel.
- In the ***Feature Properties*** panel, rename the feature by highlighting the name and typing in a new name. Click the ***Modify*** button to continue.

ACTIVITY: Renaming A Feature

For this activity, you will rename the features in the ***Feature Properties*** panel:

- Circles 1, 3, 5, 7, Cylinder 9, and Sphere A

1. Select ***Circle 4***	Double-click ***Circle 4*** in the ***Features*** panel.
2. Type new name	With the feature name highlighted, type `Circle 7` and click ***Modify***.
3. Select ***Circle 3***	Double-click ***Circle 3*** in the ***Features*** panel.
4. Type new name	With the feature name highlighted, type `Circle 5` and click ***Modify***.
5. Select ***Circle 2***	Double-click ***Circle 2*** in the ***Features*** panel.
6. Type new name	With the feature name highlighted, type `Circle 3` and click ***Modify***.

7. Select ***Sphere 1***	Double-click ***Sphere 1*** in the ***Features*** panel.
8. Type new name	With the feature name highlighted, type `Sphere A` and click ***Modify***.
9. Select ***Cylinder 1***	Double-click ***Cylinder 1*** in the ***Features*** panel
10. Type new name	With the feature name highlighted, type `Cylinder 9` and click ***Modify***. Click ***Close*** to complete the feature renaming.

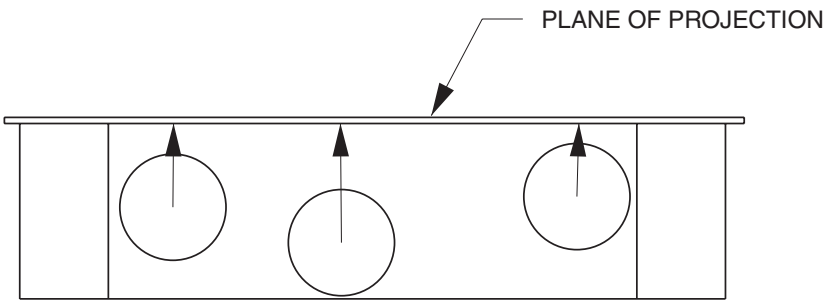
Projection Plane

Plane of Projection for 2D features

In order for the CAM2 Q to determine a 2D feature's relative position in space, these features must lay on a plane. This plane is called the plane of projection.

The following 2D features must lay on a plane of projection:

- Circle
- Line (2D line)
- Slot (Round and Rectangular)
- Ellipse
- 2D polyline



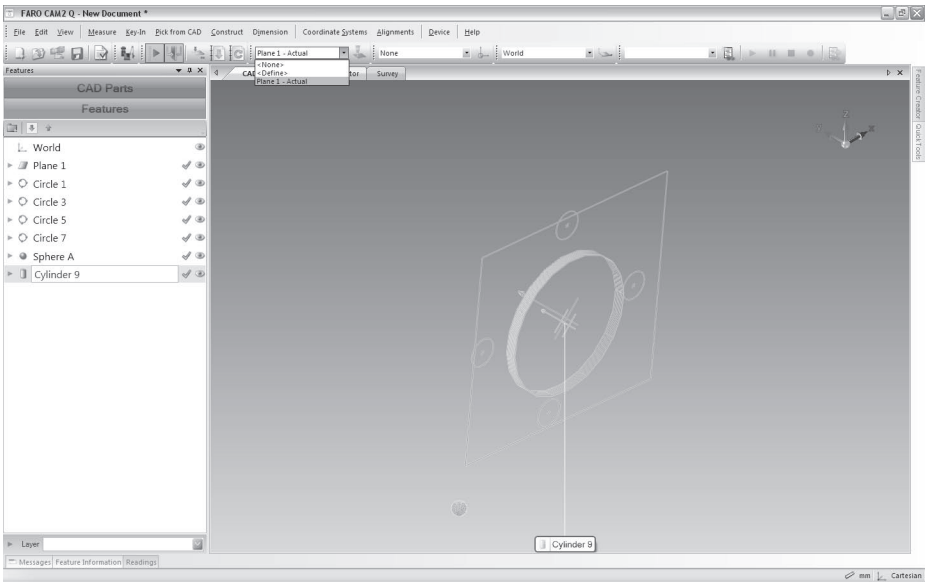
Defining a Plane of Projection

Before measuring a 2D feature, you must first measure a plane to define the plane of projection.

Setting the Plane of Projection

After measuring the plane, you measure the 2D feature, and then set the plane of projection. To set the correct plane of projection:

- On the ***Projection Plane*** toolbar, in the ***Active Projection Plane*** drop-down box, select the correct plane.



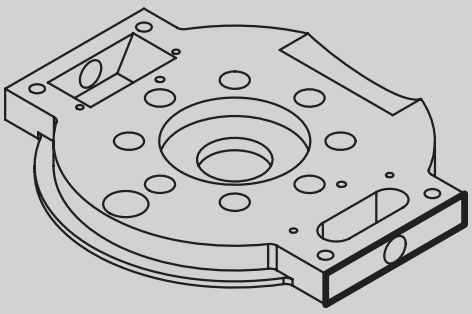
Plane of Projection: None

In some instances, the points collected for a feature will also define the plane of projection for that feature. In these instances, you want to select ***None*** for the plane of projection:

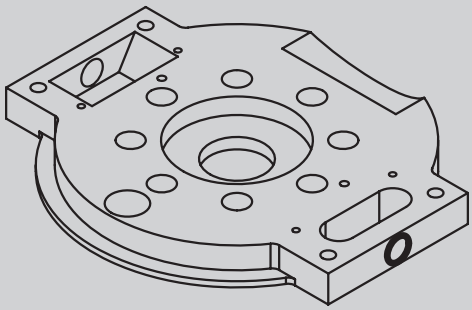
- On the ***Projection Plane*** toolbar, in the ***Active Projection Plane*** drop-down box, select ***None***.

ACTIVITY: Measure Features On A New Plane

For this activity, you will measure features, set the plane of projection, and rename the feature.

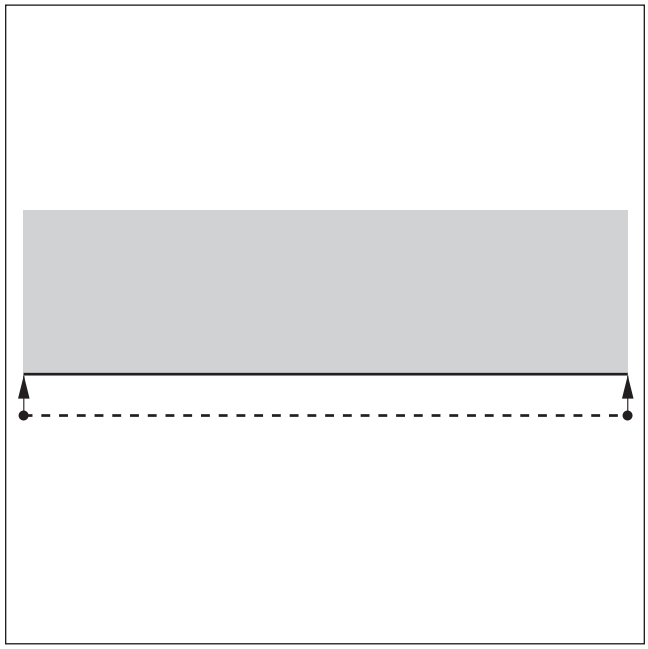
1. ***Measure Mode***	If necessary, click the ***Measure/Template Mode*** icon so that you are in ***Measure*** mode.
2. Measure Plane	Measure Plane on Circle 11 side. 

The plane of projection for Circle 11, in the Active Projection Plane drop-down box, now shows as Plane 2.

3. Measure Circle 11	Measure Circle 11. 
4. Select ***Circle 2***	Double-click ***Circle 2*** in the ***Features*** panel.
5. Type new name	With the feature name highlighted, type `Circle 11`. Click ***Modify*** and then ***Close*** to finish.

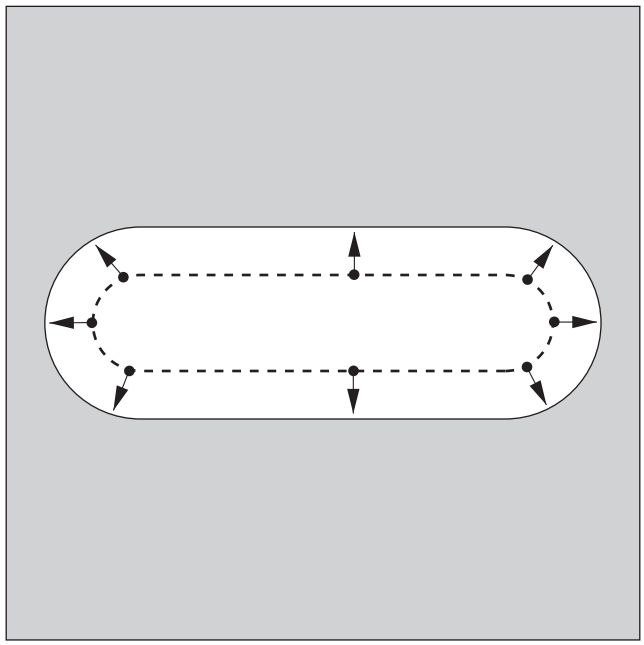
Measuring a 2D Line

When measuring a 2D line, probe location during the End Click is critical for correct placement of the line. You should correctly pull the probe away from the surface that represents the line.



Measuring a Round Slot

When measuring a round slot, probe location during the End Click is critical for calculating the correct diameter. You should correctly pull the probe to the center of the slot, or away from the outside of the slot in the case of an outside diameter round slot.

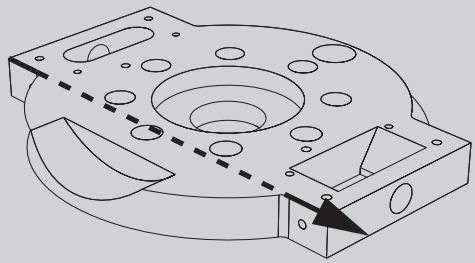
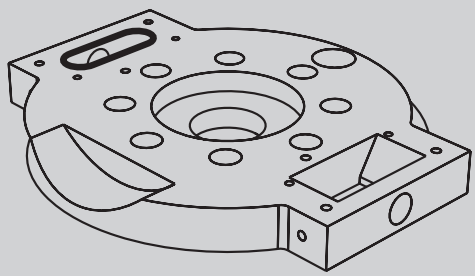
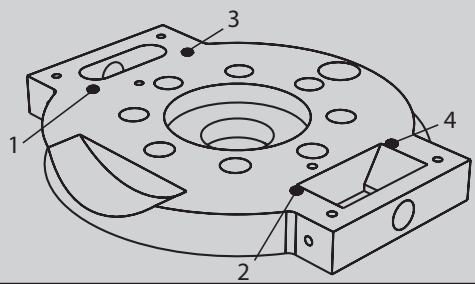


Measuring a Point with No Plane of Projection

When measuring a point, probe location during the End Click is not critical. The FaroArm collects the reading for the point at the center of its probe, and the End Click simply concludes the taking of readings.

ACTIVITY: Set The Plane Of Projection And Measure Basic Features

For this activity, you will set the plane of projection and measure more basic features in ***Measure*** mode.

1. Select ***Active Projection Plane***	On the ***Projection Plane*** toolbar, in the ***Active Projection Plane*** drop-down box, select ***Plane 1***.
2. Measure 2D Line	On the ***Measure*** menu, click ***2D Line*** and measure the line. 
3. Measure Round Slot	On the ***Measure*** menu, click ***Round Slot*** and measure the round slot. 
4. Measure Points	On the ***Measure*** menu, click ***Point*** and measure the four points by seating the 6mm probe firmly in each 5mm hole. 

Hot Keys
F4:
Measure 2D Line
F2:
Measure Point

ACTIVITY: Save a File

You will use the measurements you have taken to learn about the ***Construct*** and ***Dimension*** menus. To make this information easily accessible, you can save a file with your data.

1. ***Save***	On the ***File*** menu, click ***Save***. The ***Save As*** dialog box appears.
2. Browse to ***CAM2 Q***	In the ***Save In*** list, using the drop-down menu, select ***My Documents***, and click on the ***CAM2 Q*** folder.
3. Create new folder	In the ***CAM2 Q*** folder, click the ***Create New Folder*** icon. Name the folder [`your name`]. Then, double-click your folder to open it.
4. Type file name	In the File name box, type `Measure.fcd`.
5. ***Save***	Click ***Save***.

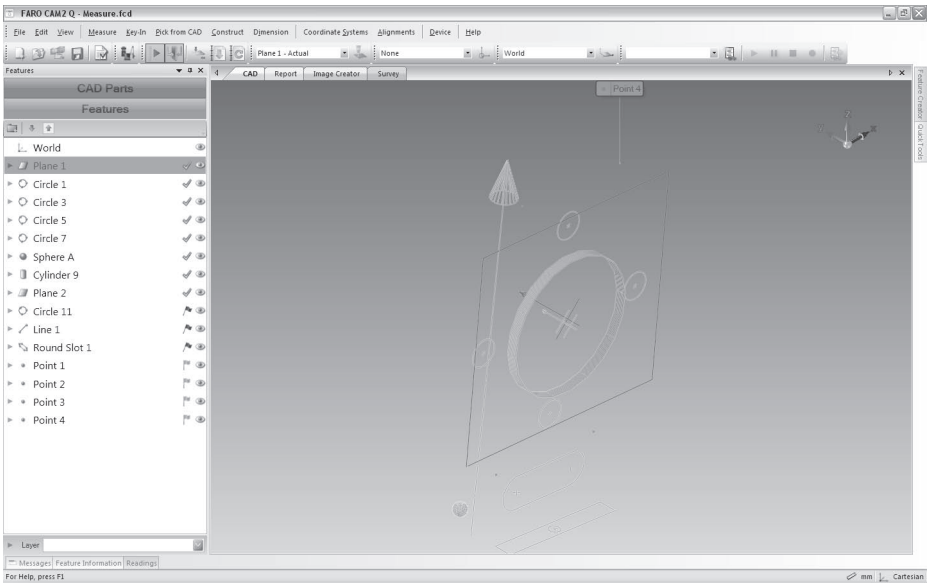
Feature Color

Each type of feature that you add to the measurement file has a unique color that matches the menu color from the ***Feature Creator*** panel. When you move the mouse pointer over a feature in the ***Main Window*** or the ***Features*** panel, the color changes to red. When you select feature from the ***Main Window*** or the ***Features*** panel, the color changes to a bright yellow.

Feature Status Icons

To the right of each feature in the Features panel is a status icon. The status icon changes color as you take measurements to indicate:

- You have not taken enough points to define a feature (gray flag icon)
- You have taken enough points to define the feature and it is in tolerance (green checkmark icon)
- You have taken enough points to define the feature and it is out of tolerance (red flag icon /blue flag icon)



Show and Hide Features

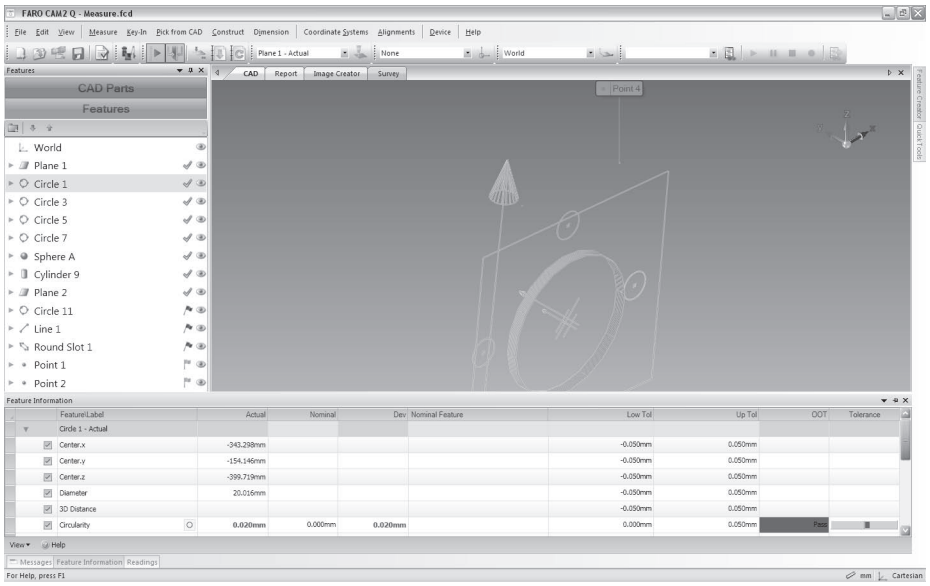
To the extreme right of all features is the Feature Visibility icon (eye icon). Click the Feature Visibility icon to hide a feature. Once hidden, the icon appears gray. Click the Feature Visibility icon again to show the feature.

When hidden, features remain in the feature tree but are not shown in the Main Window.

Evaluate Measurement Data

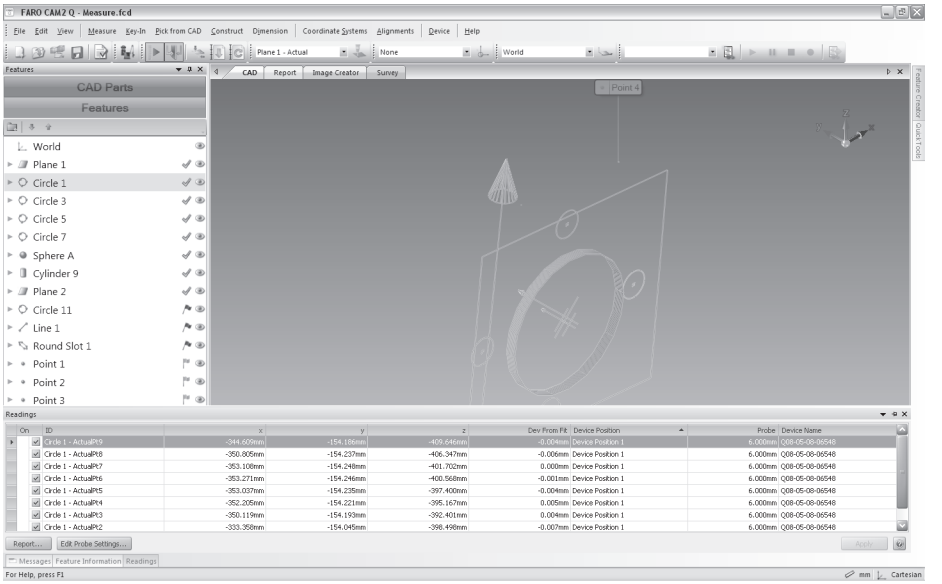
A number of screen elements assist you in assessing the quality of the part or feature you are measuring.

- ***Form*** is a measure of the geometric accuracy of the feature. Examples are the circularity of a circle, the straightness of a line, and the flatness of a plane. The measured feature's label shows the form value of the feature.
- ***In Tolerance*** is defined when you establish your nominals and tolerances. If a feature is in tolerance, the icon next to the feature in the ***Features*** panel will be a green check.
- ***Out of Tolerance*** is defined when you establish your nominals and tolerances. If a feature is out of tolerance, the icon next to the feature in the ***Features*** panel will be a red or blue flag.



Graphical tolerances use color gradients to show more detail about the actual number in a tolerance band. Blue and Red become lighter as the value approaches the-tolerance band edge. Green becomes darker as the actual number approaches the exact nominal value.

- ***Readings*** shows the deviation from fit for each point taken.



Adding/Removing Readings

Removing Readings from a Feature's Calculation

To remove a reading from a feature's calculation:

- On the ***Readings*** panel, clear the check box in the ***On*** column. Click the ***Apply*** button to re-calculate the feature with the remaining readings.

Make sure that you do not remove a reading need to solve the feature.

Adding Readings To An Existing Feature

To add readings to a feature:

- Right-click the feature on the ***Features*** panel or the on the image in the ***Main Window***, then click ***Add Readings***.
- Press the ***FRONT*** button and take more readings.

Permanently Removing Readings from a Feature

To permanently delete a reading from a feature:

- Press the ***Backspace*** key while measuring that feature. One reading is removed each time you press the ***Backspace*** key; or
- On the ***Readings*** panel, select the reading you want to remove, and press the ***Delete*** key.

After removing a reading, you may want to remeasure that specific area of the part to rule out user, device, or part error.

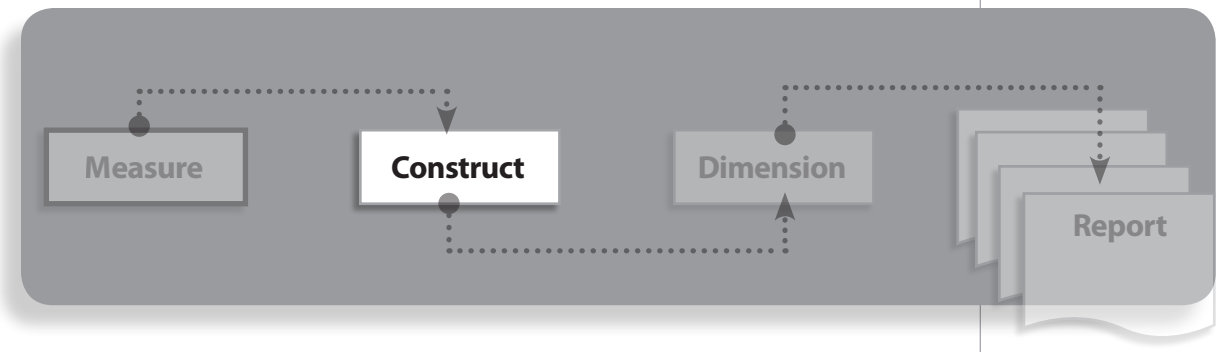
Measuring Mode: Scan Mode

So far, when measuring, you have taken single points. Another way to take points is by scanning, which takes points continuously while you press the ***FRONT*** button.

You can enable scanning by selecting a mode in the ***Measurement Window*** panel which appears when selecting a measurement command. The three modes are:

- **Single Point**: press the ***FRONT*** button to collect a single reading.
- **Time Interval**: press the ***FRONT*** button to start the time interval. Collects single readings over time.
- **Distance Interval**: press the ***FRONT*** button to start the distance interval. Collect a single reading when the probe moves a distance.

Lesson 2: Construct



Now that you know how to measure features, let's learn how to construct a feature. A construction allows you to create features that you cannot measure directly. Sometimes points or other features are specified on a drawing, but do not actually exist on the part, e.g. a bolt-hole circle.

Lesson Objective

With the assistance of the ***Help*** menu, you will be able to use all of the functions within the ***Construct*** menu. You will be able to:

- Identify point-, line-, and plane-reducibility for each feature type.
- Identify why constructions are needed.
- Locate, review, and evaluate feature construction quality.

Why are Constructions Needed?

Constructions allow you to create features that you cannot directly measure, such as a bolt-hole circle.

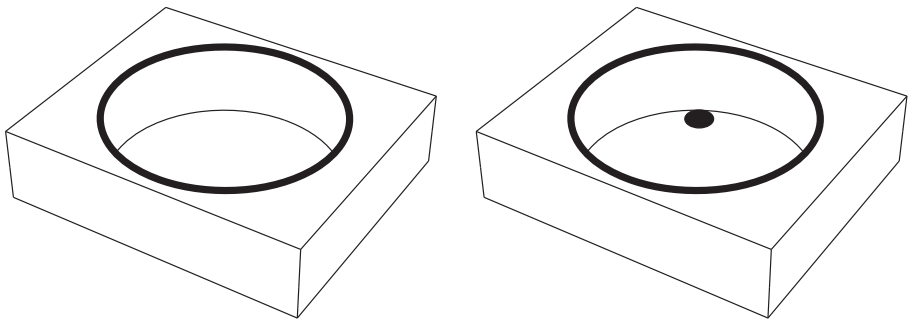
The activities in this Lesson allow you to practice some commonly used construction commands.

As we discuss and practice using the ***Construct*** menu, you will use features you saved in the Measure lesson.

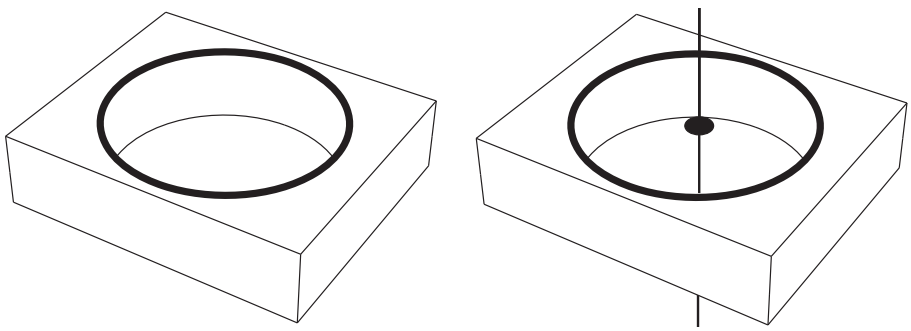
Feature Reducibility

Feature Reducibility means that a part of a feature can be used for constructions and dimensions. For example, a circle has three reducible parts.

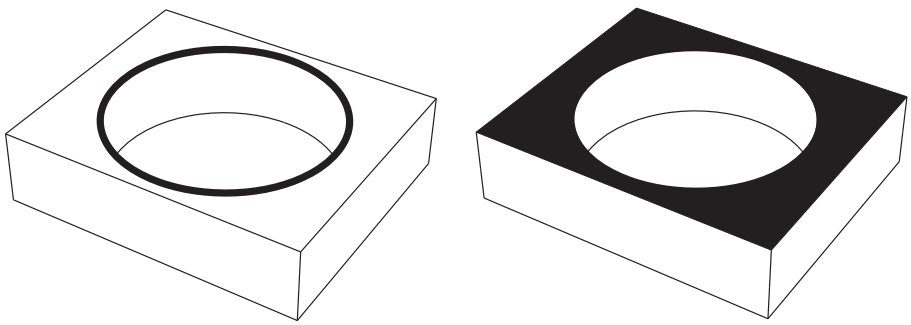
A circle is ***Point-Reducible***, which means it can be used like a point for constructions, or dimensions. The point of a circle is its center point.



A circle is ***Line-Reducible***. The line of a circle is perpendicular to the circle's plane through the center point.

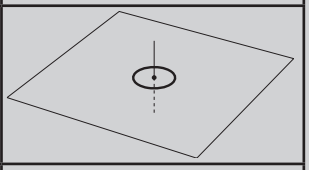
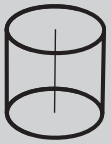
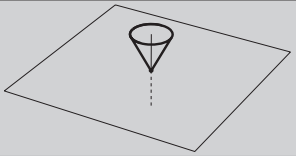
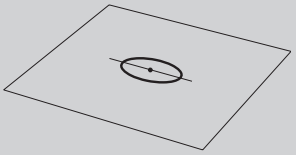
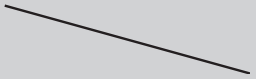
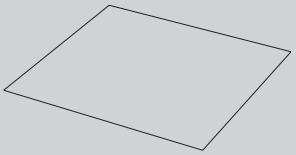
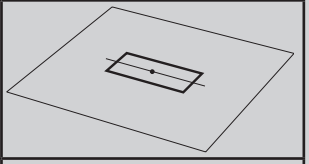
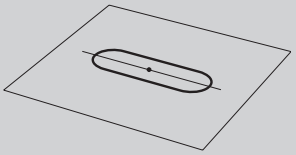
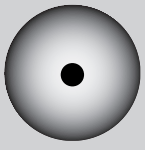


A circle is ***Plane-Reducible***. The plane of a circle is its plane of projection.



ACTIVITY: Feature Reducibility Exercise

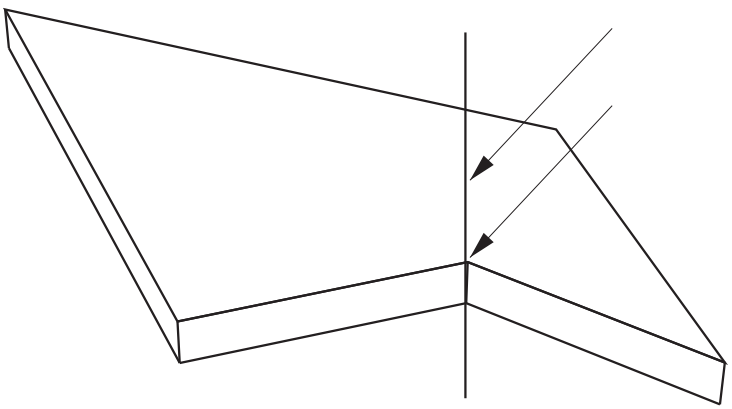
Look at each feature and determine if it is point-, line-, or plane-reducible. Some features may be reducible to more than one type of feature.

	Point	Line	Plane	
Circle				
Cylinder				
Cone				
Ellipse				
Line				
Plane				
Point				
Rectangular Slot				
Round Slot				
Sphere				

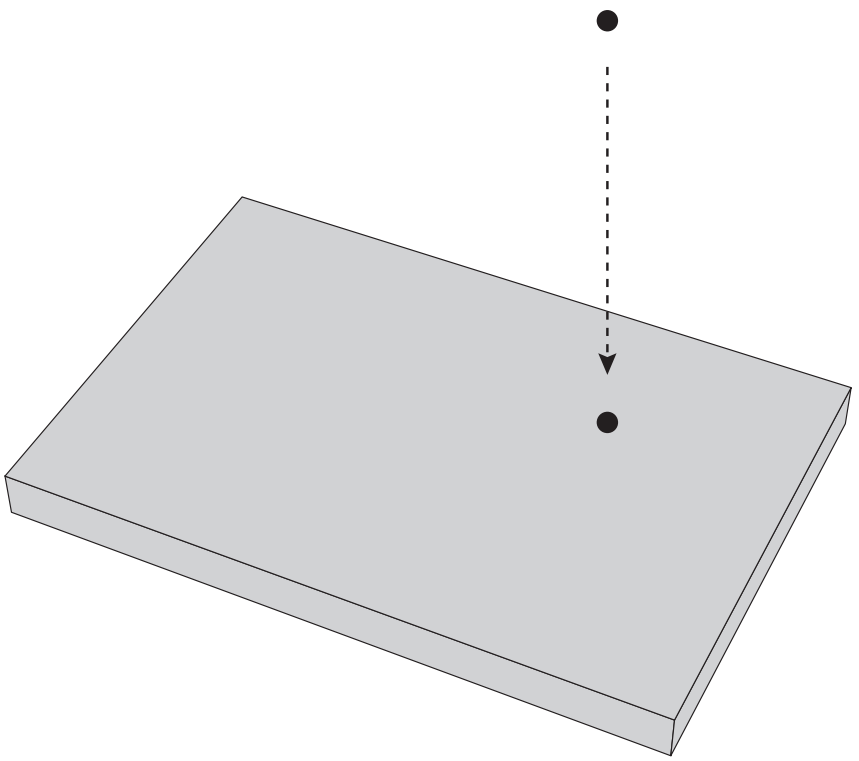
Point Constructions

Four of the most commonly used point constructions are ***Intersection***, ***Projection***, ***Bisection*** and ***Key-In***.

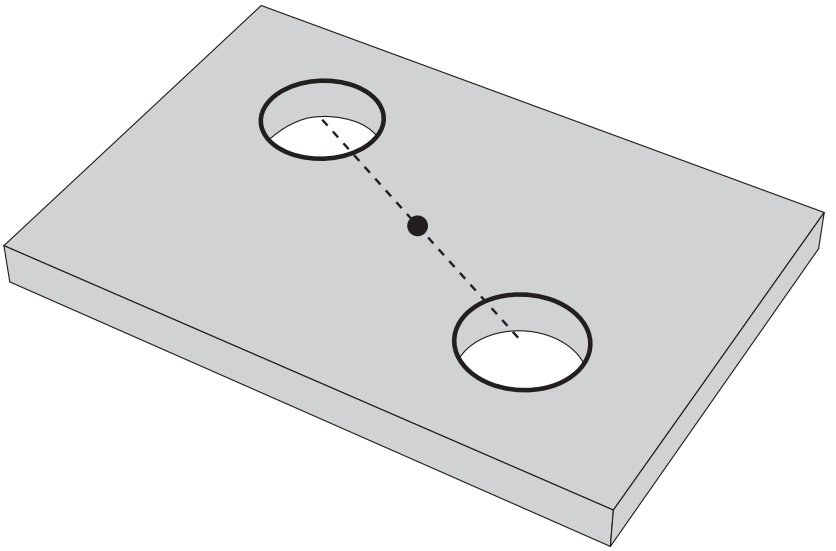
- ***Intersection*** allows you to construct a point from the intersection of two features or three planes.



- ***Projection*** allows you to construct a point and project it onto a feature.



- ***Bisection*** allows you to construct a point in the center of two point-reducible features.



- ***Key-In*** allows you to construct a point by entering the location of the point.

Key-in requires the input of nominals which we will discuss later in the workbook.

ACTIVITY: Open A Saved File

1. File ***Open***	On the ***File*** menu, click ***Open***.
2. ***My Documents***	In the ***Look In*** drop-down list, select ***My Documents***.
3. Select ***your folder***	Double-click on the ***CAM2 Q*** folder, and then double-click the your folder.
4. Select ***Measure.fcd***	Select ***Measure.fcd***
5. ***Open***	Click ***Open***.

ACTIVITY: Construct Point By Intersection Of Two Features

For this activity, you will construct a point from the intersection of Plane 1 and Cylinder 9.

1. ***By Intersection***	On the ***Construct*** menu, click ***Point***, then click ***By Intersection***. The ***Construct Point By Intersection*** dialog box appears.
2. Name the point	In the ***Name*** box, enter `Intersection Point`.
3. Click ***Two Features***	Click the ***Two Features*** option button.
4. Select first feature	In the first ***Feature*** list, select ***Plane 1***.

5.	Select second feature	In the second ***Feature*** list, select ***Cylinder 9***.
6.	Treat as ***Line***	In the ***treat as*** list, select ***Line***.
7.	***Create***	Click ***Create*** to view the result.
8.	***Close***	Click ***Close*** to close the dialog box.

ACTIVITY: Construct Point By Projection

For this activity, you will construct a point from the projection of Sphere A to Plane 1.

1.	***By Projection***	On the ***Construct*** menu, click ***Point***, then click ***By Projection***. The ***Construct Point By Projection*** dialog box appears.
2.	Name the point	In the ***Name*** box, enter `Projection Point`.
3.	Select point	In the ***Point*** list, select ***Sphere A***.
4.	Select feature to project to	In the ***Project To*** list, select ***Plane 1***.
5.	***Create***	Click ***Create*** to view the result.
6.	***Close***	Click ***Close*** to close the dialog box.

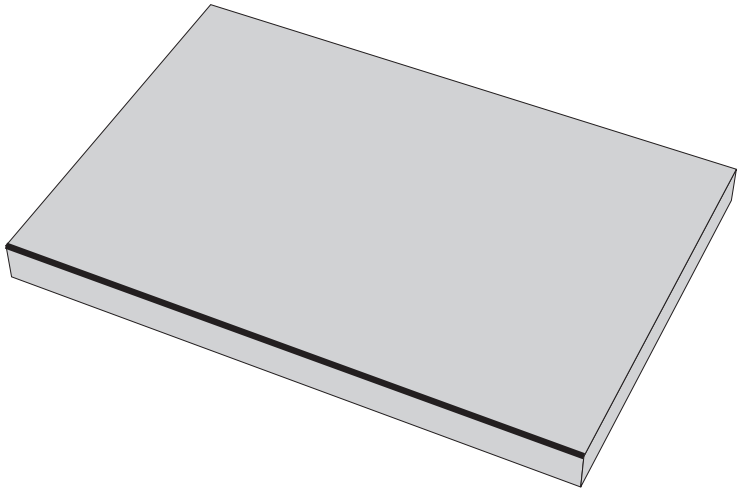
ACTIVITY: Construct Point By Bisection

For this activity, you will construct a point halfway between two points.

1.	***By Bisection***	On the ***Construct*** menu, click ***Point***, then click ***By Bisection***. The ***Construct Point By Bisection*** dialog box appears.
2.	Name the point	In the ***Name*** box, enter `Bisection Point`.
3.	Select first feature	In the ***Feature 1*** list, select ***Point 1***.
4.	Select second feature	In the ***Feature 2*** list, select ***Point 2***.
5.	***Create***	Click ***Create*** to view the result.
6.	***Close***	Click ***Close*** to close the dialog box.

Line Constructions

One of the most commonly used line constructions is ***Intersection of Two Planes.***



ACTIVITY: Construct Line By Intersection Of Two Planes

For this activity, you will construct a line using the intersection of Plane 1 and Plane 2.

1. ***By Intersection of Two Planes***	On the ***Construct*** menu, click ***Line***, then click ***By Intersection of 2 Planes***. The ***Construct Point By Intersection of Two Planes*** dialog box appears.
2. Name the line	In the ***Name*** box, enter `Constructed Line 1`.
3. Select first feature	In the ***Feature 1*** list, select ***Plane 1***.
4. Select second feature	In the ***Feature 2*** list, select ***Plane 2***.
5. ***Create***	Click ***Create*** to view the result.
6. ***Close***	Click ***Close*** to close the dialog box.

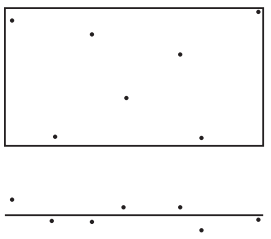
*To edit any of these features, select the feature in the Features panel, double-click, make changes in the **Feature Properties** panel, and click **Modify**.*

Plane Constructions

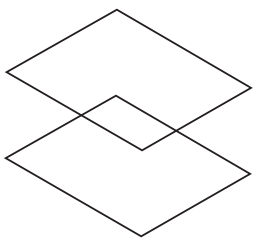
Three of the most commonly used plane constructions are ***Best Fit***, ***By Offset*** and ***Perpendicular***.

You can preview a "By best Fit" feature before you create it, using a graphic feature in the Main window, and feature data in all of the By Best Fit dialog boxes.

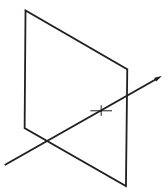
- ***Best Fit*** enables you to construct a plane by fitting it through point-reducible features.
- ***By Offset*** enables you to construct a plane at a known distance parallel to an existing plane. You can locate the plane by either:
  - ➡ Moving the plane by an absolute offset.
  - ➡ Placing it through another point.
  - ➡ Offsetting the plane along a normal (line).
- ***Perpendicular*** enables you to construct a plane perpendicular to a line.



Best Fit



By Offset



Perpendicular

ACTIVITY: Construct A Best Fit Plane

For this activity, you will construct a best fit plane using the previously measured four circles.

1. ***By Best Fit***	On the ***Construct*** menu, click ***Plane***, then click ***By Best Fit***. The ***Construct Plane By Best Fit*** dialog box appears.
2. Name the plane	In the ***Name*** box, enter `Best Fit Plane`.
3. Select ***Circles 1, 3, 5,*** and ***7***	In the ***Select Entities to Fit Through*** list, hold down the ***Ctrl*** key and select ***Circle 1***, ***Circle 3*** , ***Circle 5***, and ***Circle 7***.
4. ***Create***	Click ***Create*** to view the result.
5. ***Close***	Click ***Close*** to close the dialog box.

ACTIVITY: Construct Plane By Offset

For this activity, you will construct a plane parallel to the best fit plane.

1. ***By Offset***	On the Construct menu, click ***Plane***, then click ***By Offset***. The ***Construct Plane By Offset*** dialog box appears.
2. Name the plane	In the ***Name*** box, enter `Offset Plane`.
3. Select base feature	In the ***Base Feature*** list, select ***Best Fit Plane***.
4. Select ***Offset Along Normal*** and select feature.	Select the ***Offset Along Normal*** option button. Select ***Best Fit Plane*** from the drop-down list.
5. Enter distance	In the ***Distance*** box, enter either `20.00` ***(mm)*** or `0.7874` ***(in)***
6. ***Create***	Click ***Create*** to view the result.
7. ***Close***	Click ***Close*** to close the dialog box.

ACTIVITY: Construct Plane Perpendicular

For this activity, you will construct a plane perpendicular to the front edge.

1. ***Perpendicular***	On the ***Construct*** menu, click ***Plane***, then click ***Perpendicular***. The ***Construct Plane Perpendicular*** dialog box appears.
2. Name the plane	In the ***Name*** box, enter `Perpendicular Plane.`
3. Select ***Line*** and select feature	Select the ***Line*** option button. Select ***Line 1*** from the drop-down list.
4. Select point	In the ***Point*** box, select ***Circle 3***.
5. ***Create***	Click ***Create*** to view the result.
6. ***Close***	Click ***Close*** to close the dialog box.

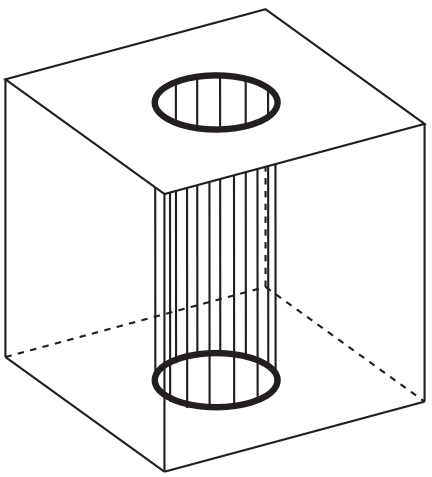
Circle Constructions

Three of the most commonly used circle constructions are ***Plane Intersect, Best Fit*** and ***Key In***.

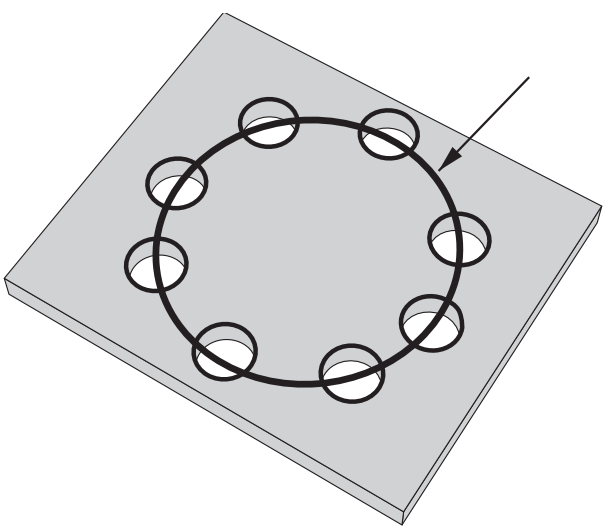
- ***Plane Intersect***—allows you to construct a circle from the intersection of a plane and a feature, such as a cone or cylinder.
- ***Best Fit***—allows you to construct a bolt circle pattern from previously measured circles.
- ***Key-In***—allows you to construct a circle by entering the name, location, diameter and normal vector.

Next, we will discuss ***Plane Intersect*** and ***Best Fit***. Since ***Key-In*** uses nominals, we will discuss that in the Nominals lesson.

Plane Intersect



Best Fit



ACTIVITY: Circle Intersect

For this activity, you will construct a circle using the intersection of a plane and a feature.

1. ***By Intersection of Plane and a Feature***	On the ***Construct*** menu, click ***Circle***, then click ***By Intersection of Plane and a Feature***. The ***Construct Circle By Intersection of Plane and a Feature*** dialog box appears.
2. Name the circle	In the ***Name*** box, enter `Circle by Intersection`.
3. Select first feature	In the ***Feature 1*** list, select ***Plane 1***.
4. Select second feature	In the ***Feature 2*** list, select ***Cylinder 9***.
5. Lock Plane of Circle	Select ***To Plane***.
6. Create	Click ***Create*** to view the result.
7. Close	Click ***Close*** to close the dialog box.

ACTIVITY: Best Fit

For this activity, you will construct a best fit bolt-hole pattern using three of the previously measured four circles.

1. ***By Best Fit***	On the ***Construct*** menu, click ***Circle***, then click ***By Best Fit***. The ***Construct Circle By Best Fit*** dialog box appears.
2. Name the circle	In the ***Name*** box, enter `Bolt Circle`.
3. Select entities to fit through	In the ***Select Entities to Fit Through*** list, hold down the ***Ctrl*** key and select ***Circle 1***, ***Circle 3***, and ***Circle 5***.
4. Click ***Use Plane*** and select ***Plane 1***	Make sure the ***Use Plane*** check box is selected and in the ***Use Plane*** list, select ***Plane 1***.
5. ***Create***	Click ***Create*** to view the result.
6. ***Close***	Click ***Close*** to close the dialog box.

ACTIVITY: Review And Evaluate Data

(Form And Diameter Of Constructed Features)

Now, let us review the data of the Bolt Circle.

1. Select Bolt Circle	Select the ***Bolt Circle*** in the ***Features*** panel.
2. ***Feature Information*** panel	At the bottom of the CAM2 Q window, click the ***Feature Information*** tab to open the ***Feature Information*** panel.
3. Review data	Note the data of the Bolt Circle in the ***Feature Label*** column. ☛ The ***Circularity*** (Form) will be "0" - because three points are all that is required to define a circle, the form of a circle defined by three points will always be perfect.
4. Note the diameter	Note the ***Diameter*** for reference when you add another circle in the next activity.

Form: _____

Diameter: _____

ACTIVITY: Edit The Data For Best Fit Constructed Circle

Edit the Bolt Circle.

1. Bolt-Circle ***Properties***	In the ***Features*** panel, double-click ***Bolt Circle***.
2. Select entities to fit through	In the ***Select Entities to Fit Through*** list, Ctrl + click ***Circle 7***.
3. ***Modify***	Click ***Modify*** to view the result.
4. ***Close***	Click ***Close*** to close the dialog box.

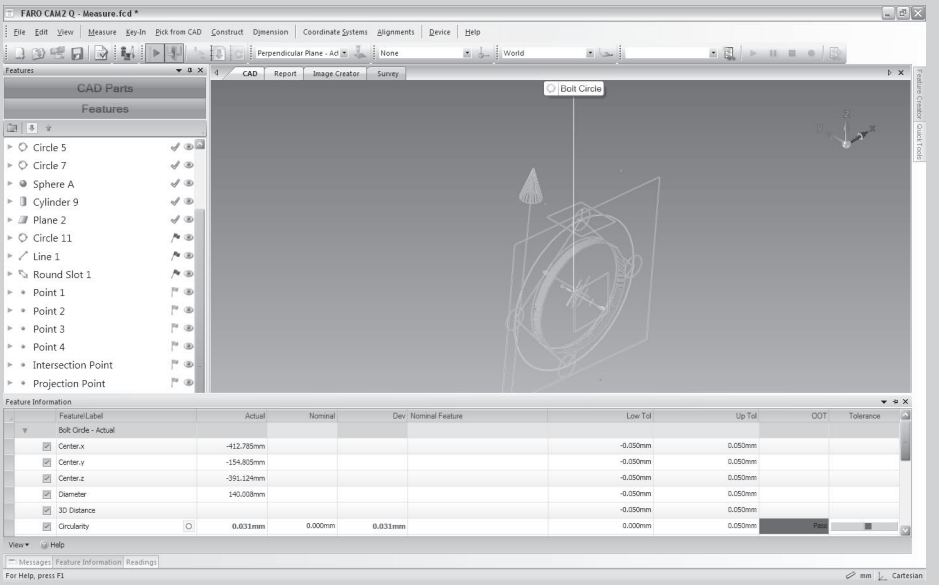
ACTIVITY: Review The Data For The Best Fit Constructed Circle

For this activity, you will compare the data of the edited Bolt Circle to the previous Bolt Circle data.

Form: _____

Diameter: _____

1. Select ***Bolt Circle***	On the ***Features*** panel, click ***Bolt Circle***.
2. ***Feature Information*** panel	Open the ***Feature Information*** panel.
3. View data	Note the updated data of the edited Bolt Circle. ☛ *Since the circle is defined by a best fit using more than three points, it will most likely show less than perfect form. Additionally, the diameter of the circle will be slightly different due to the change in form.*

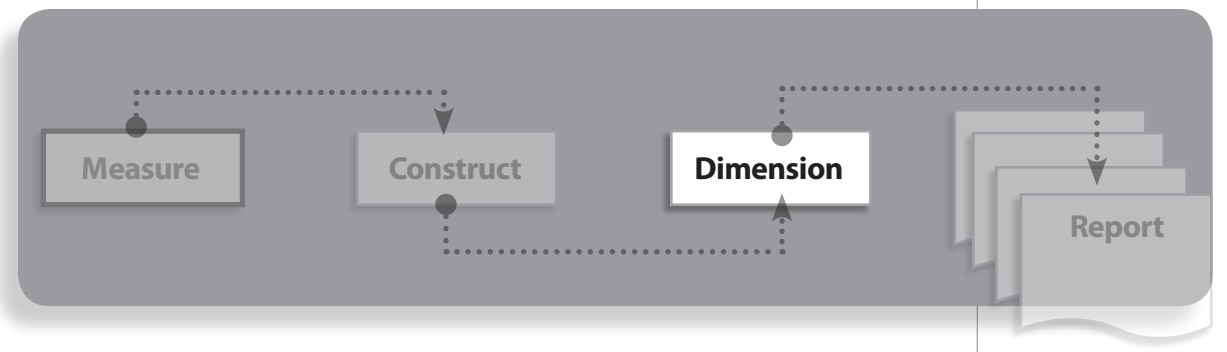


ACTIVITY: Save A File

You should save your work to this point.

1. ***Save As***	On the ***File*** menu, click ***Save As***. The ***Save As*** dialog box appears.
2. ***My Documents*** folder	In the ***Save In*** drop-down list, select ***My Documents***.
3. Your folder	Double-click on the ***CAM2 Q*** folder and browse to ***your folder***.
4. Enter `Construct.fcd`	In the File name box, type `Construct.fcd`.
5. ***Save***	Click ***Save***.

Lesson 3: Dimension



The third part of the ***Basic Measurement Strategy*** is ***Dimension***. A dimension describes the relationship between two or more features, e.g. the distance between or the angle between features.

Lesson Objective

With the assistance of the ***Help*** menu, you will be able to use all of the functions within the ***Dimension*** menu. You will be able to:

- Identify why dimensions are needed.
- Locate dimension data for evaluation.
- Edit features used to create a dimension.

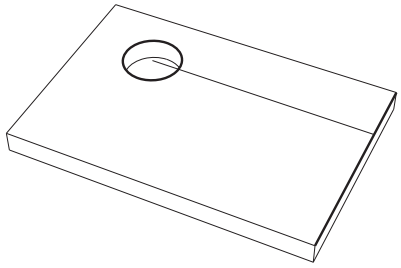
Types of Dimensions

There are two types of dimensions available in CAM2 Q: ***Length*** and ***Angle***.

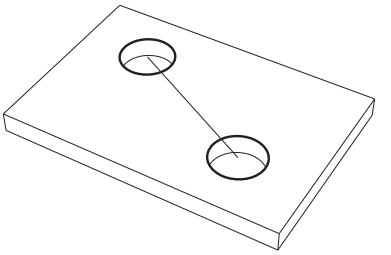
Length Dimensions

Two of the most commonly used length dimensions are ***Point to Line*** and ***Point to Point***.

- ***Point to Line*** calculates the relationship between a point and a line.



- ***Point to Point*** calculates the relationship between a point and a point.



ACTIVITY: Open A Saved File

1. File ***Open***	On the ***File*** menu, click ***Open***.
2. ***My Documents***	In the ***Look In*** drop-down list, select ***My Documents***.
3. Select ***your folder***	Double-click on the ***CAM2 Q*** folder, and then double-click the your folder.
4. Select ***Construct.fcd***	Select ***Construct.fcd***
5. ***Open***	Click ***Open***.

ACTIVITY: Dimensioning a Length from Features

For this activity, you will obtain the length between two features, a point (Sphere A) and a line (2D Line 1).

1. ***Length from Features***	On the ***Dimension*** menu, click ***Length From Features***. The ***Dimension Length From Features*** dialog box appears.
2. Open Help	Click the ***Help*** button. The CAM2 Q help file opens to the ***Length: from Features*** topic. Review the length examples and close the help file.
3. Enter name	In the ***Name*** box, enter `Point to Line`.
4. Select ***Sphere A*** and treat as ***Point***	In the first feature list, select ***Sphere A***. Then in the ***treat as*** list, select ***Point***.
5. Select ***2D Line 1*** and treat as ***Line***	In the second feature list, select ***Line 1***. Then in the ***treat as*** list, select ***Line***.
6. ***Create***	Click ***Create*** to view the result.
7. ***Close***	Click ***Close*** to close the dialog box.
8. View results	In the ***Features*** panel, select ***Point to Line*** and open the ***Feature Information*** panel. Note the length.

Length: _____

You are using the concept of feature reducibility to determine how to "treat" the feature.

ACTIVITY: Dimensioning Length from Features

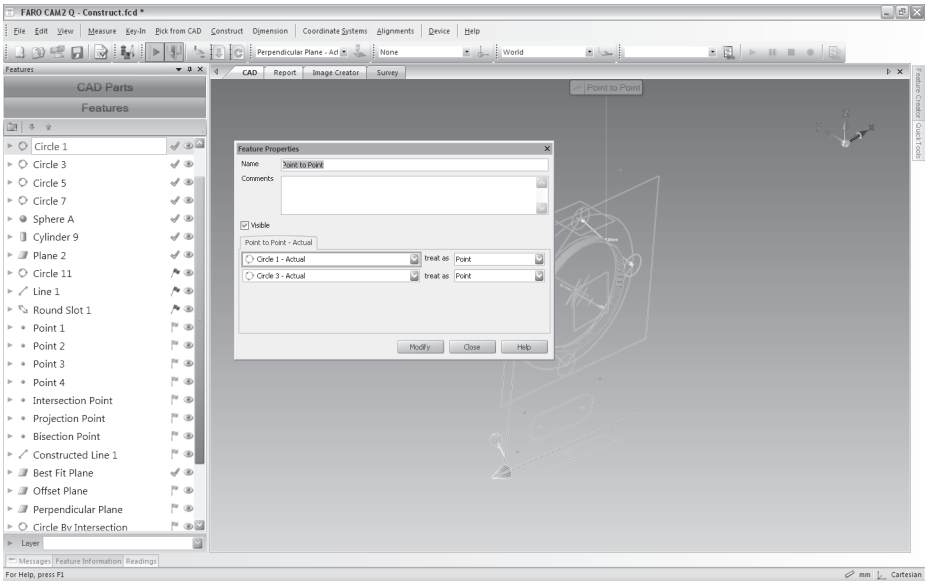
For this activity, you will obtain the length between two features, a point (Circle 1) and a point (Circle 3).

1.	***Length from Features***	On the ***Dimension*** menu, click ***Length From Features***. The ***Dimension Length From Features*** dialog box appears.
2.	Enter name	In the ***Name*** box, enter `Point to Point`.
3.	Select ***Circle 1*** and treat as ***Point***	In the first feature list, select ***Circle 1***. Then in the ***treat as*** list, select ***Point***.
4.	Select ***Circle 3*** and treat as ***Point***	In the second feature list, select ***Circle 3***. Then in the ***treat as*** list, select ***Point***.
5.	***Create***	Click ***Create*** to view the result.
6.	***Close***	Click ***Close*** to close the dialog box.
7.	View results	In the ***Features*** panel, select ***Point to Point*** and open the ***Feature Information*** panel. Note the length.

Length: _____

Editing Features

Sometimes when you will want to change the features you have selected to create a dimension. Rather than reconstruct all of the feature selections, you can do this using an editing dialog box.



ACTIVITY: Editing Features To Create A Line

For this activity, you will edit features to create a line.

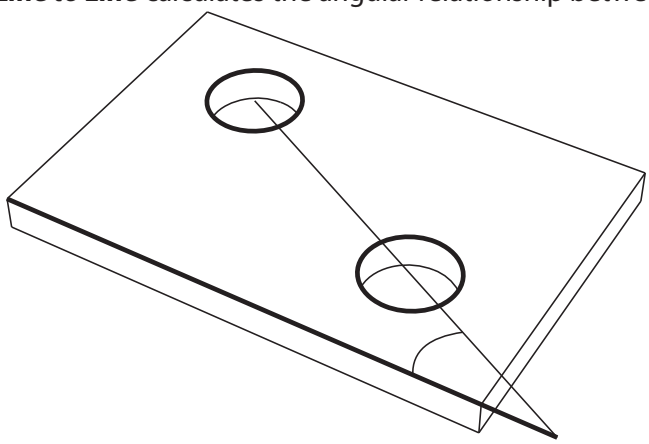
1. Select feature	In the ***Features*** panel, double-click ***Point to Point***.
2. Edit ***Properties***	The ***Feature Properties*** dialog box appears.
3. Select ***Circle 5***	Using the drop-down menu, replace ***Circle 3*** with ***Circle 5***.
4. Complete the change	Click ***Modify***.
5. ***Close***	Click ***Close*** to close the dialog box.
6. View results	In the ***Features*** panel, select ***Point to Point*** and open the ***Feature Information*** panel. Note the length changes as a reflection of the change in features selected.

Length: _____

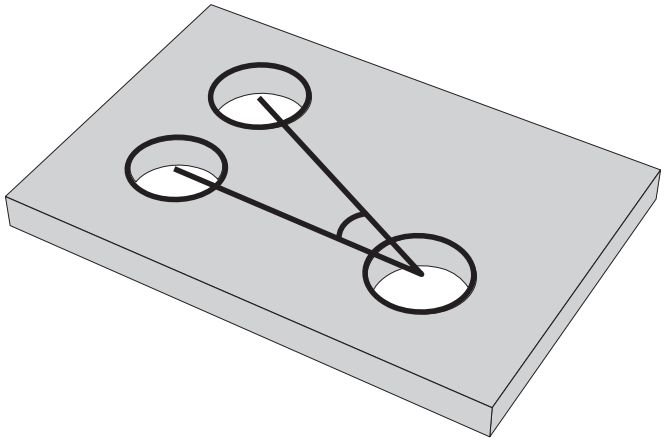
Angle Dimensions

Two of the most commonly used angle dimensions are ***Line to Line*** and ***Apex***.

- ***Line to Line*** calculates the angular relationship between two lines.



- ***Apex*** calculates the angular relationship between two points with a point-reducible feature as the apex.



ACTIVITY: Dimensioning Angles

For this activity, you will dimension an angle from features.

1. Dimension Angle From Features	In the ***Dimension*** menu, click ***Angle From Features***. The ***Dimension Angle From Features*** dialog box appears.
2. Open Help	Click the ***Help*** button. The CAM2 Q help file opens to the ***Angle: from Features*** topic. Review the differences between the two feature and three feature examples, and close the CAM2 Q help file.
3. Enter name	In the ***Name*** box, enter `Line to Line`.
4. Two Features option	Click the ***2 Features*** option button.
5. Select ***2D Line 1***	In the ***Feature 1*** list, select ***Line 1***.
6. Select ***Constructed Line 1***	In the ***Feature 2*** list, select ***Constructed Line 1***.
7. Create	Click ***Create*** to view the result.
8. Close	Click ***Close*** to close the dialog box.

ACTIVITY: Dimensioning Angles

For this activity, you will obtain the angle between Circle 1 and Circle 3 using Bolt Circle as the apex.

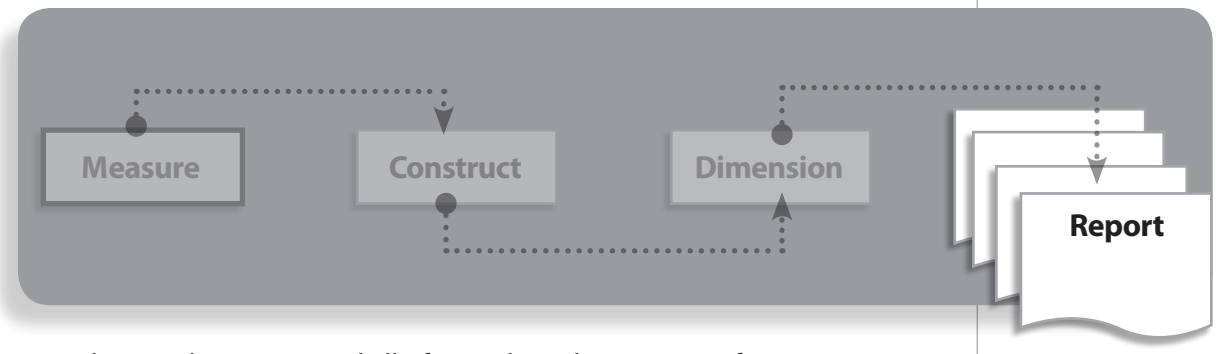
1. Dimension Angle From Features	In the ***Dimension*** menu, click ***Angle From Features***. The ***Dimension Angle From Features*** dialog box appears.
2. Enter name	In the ***Name*** box, enter `Apex`.
3. Click ***3 Points***	Click the ***3 Points*** option button.
4. Select first point	In the first ***Point*** list, select ***Circle 1***.
5. Select second point	In the second ***Point*** list, select ***Circle 3***.
6. Select apex	In the ***Apex*** list, select ***Bolt Circle***.
7. ***Create***	Click ***Create*** to view the result.
8. ***Close***	Click ***Close*** to close the dialog box.

ACTIVITY: Save A File

To make this information easily accessible, save a file with the data you have collected.

1. ***Save As***	On the ***File*** menu, click ***Save As***. The ***Save As*** dialog box appears.
2. ***My Documents*** Folder	In the ***Save In*** drop-down list, select ***My Documents***.
3. Your folder	Double-click on the ***CAM2 Q*** folder and browse to ***your folder***.
4. Enter name	In the File name box, type `Dimension.fcd`.
5. ***Save***	Click ***Save***.

Lesson 4: Report



Now that you have captured all of your data, the reporting function is an easy way to select, display, and save your data.

Lesson Objective

With the assistance of the ***Help*** menu, you will be able to create standard reports. You will be able to:

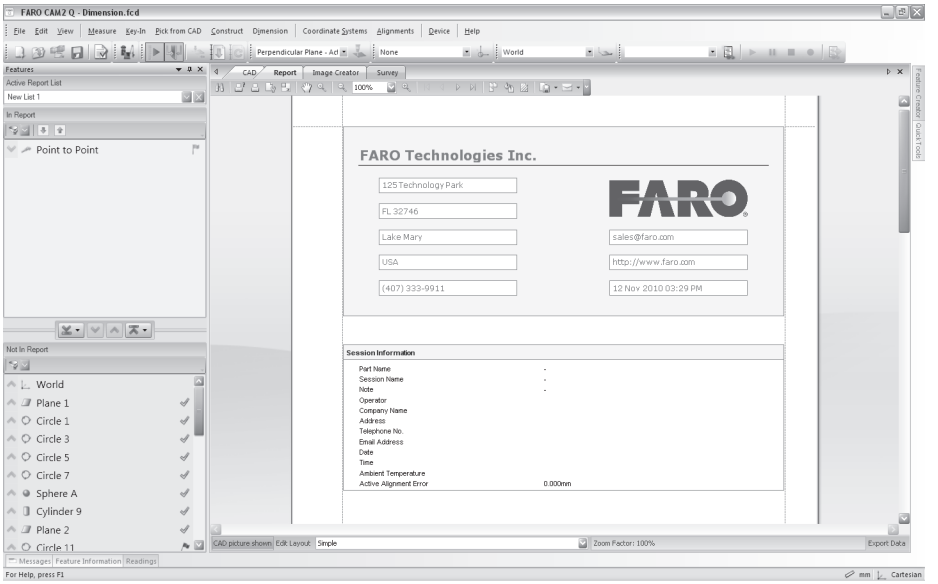
- Customize report content.
- Save a report.
- Locate and open saved reports.

Some of the common report customizations you will use are:

- Add or Remove Features
- Add or Remove a Picture
- Save a customized list of features
- Choose a report style

Reporting: Text Reports, Logos, Available Customizations

You access ***Reports*** using the ***Report*** tab located above the Main window. The ***Report Window*** appears and the ***Active Report*** List opens in the ***Feature*** panel. You can customize, export, or e-mail CAM2 Q reports in a variety of file types.



Automatically add features to the In Report list by selecting features in the Features panel before clicking the Report tab

Double-click any feature in the ***Not In Report*** list to move it to the ***In Report*** list. Double-click any feature in the ***In Report*** list to move it to the ***Not In Report*** list.

There are several actions which you can access using the ***Report Window*** icons. (from left to right)

1. Search: Searches the report for key words.

2. Printer Preferences: Sets printer preferences.

3. Print: Sends the report to the printer.

4. Page Setup: Changes the paper size, margins, and whether the printout will be in portrait or landscape.

5. Scale: Sets the size of the data on a printed page.

6. Hand Tool: Grabs the report and moves it in the window.

7. Magnifier: If the view is set to see all pages of a multiple page report, this selects one of the pages for a close up view.

8. Zoom Out: Zooms out each time you select the icon.

9. Zoom %: Selects a percentage to increase or decrease the screen size.

10. Zoom In: Zooms in each time you select the icon.

11. First Page: Scrolls to the first page of the report.

12. Previous Page: Scrolls to the previous page of the report.

13. Next Page: Scrolls to the next page of the report.

14. Last Page: Scrolls to the last page of the report.

1***5. Multiple Page:*** Sets the number of pages that display.

16. Color: Sets the background or page color of the report.

17. Watermark: Overlays an image, text, or both onto a report.

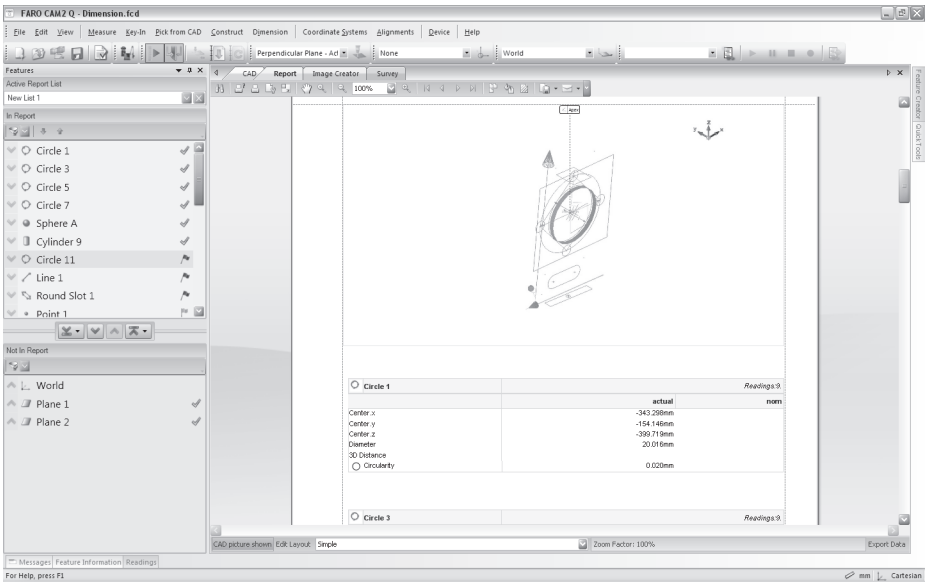
- You can add text to be watermarked. You can also choose the text direction, font, color, size, and bold or italic.
- You can add a picture or graphic to be watermarked. Use the ***Load Image*** button to browse for a picture file. Use the ***size mode***, ***horizontal*** and ***vertical alignment*** to set the position of the picture.
- Both ***Text Watermark*** and ***Picture Watermark*** have a slider bar that allows for the adjustment of the transparency of the watermark.
- Select or clear the ***Position*** radio buttons to position the watermark in front or behind the report text.
- ***Paper Range*** Selects the pages on which the watermark will appear.

18. Export Document: Saves the report in specific file formats.

19. Send via E-mail: Sends a report via e-mail in specific file formats.

Add or Remove Features on a Report

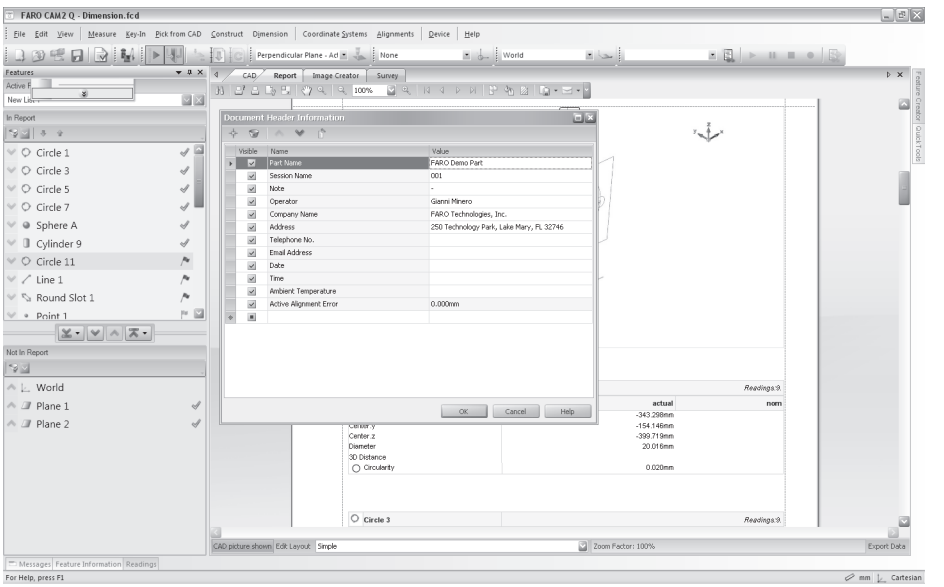
You can add any measured feature to the report. On the ***Features*** panel's ***Active Report List***, add or remove features in the report by using the Add or Remove arrows.



Document Header Information

To create and add header information, which appears as Session Information, to a report:

- On the ***Edit*** menu, click ***Document Header Information***.
- On the ***Document Header Information*** dialog box, type the desired information into the ***Value*** fields.
- Uncheck fields that you do not want to include on your report.



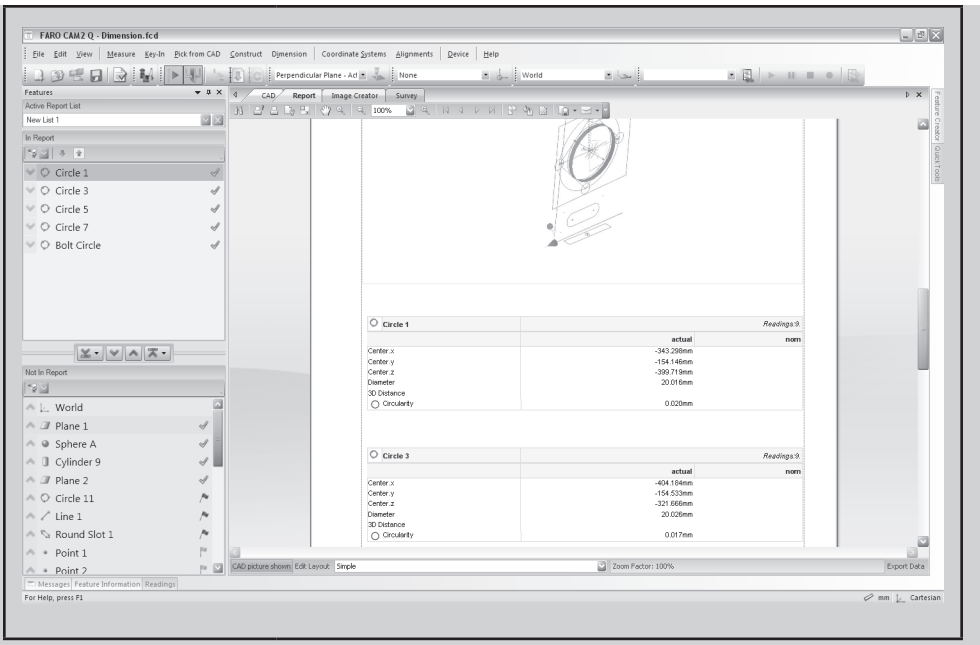
Add or Remove a Picture Preview from a Report

The ***Report*** function takes a picture of what is currently showing on the ***Main Window***. To add or remove this picture from the report, use the ***CAD Picture*** button on the bottom of the ***Report Window***.

ACTIVITY: Create A Report With Customized Content

For this activity, you will create a report including custom content.

1. Open file	On the ***File*** menu, click ***Open***.
2. Select ***Dimension.fcd***	Browse to your folder and double-click ***Dimension.fcd.*** ☛ *Before clicking the **Report** tab, make sure the view on the **Main Window** is how you would like it to appear on the report. Zoom in on the entire part or the area of the part you want to include in the report. A good way to zoom in on and orient the part is to use the hot keys. The **E** key zooms all and **6** orients the part to a top view.*
3. ***Report*** tab	Click the ***Report*** tab.
4. Add features to report	On the ***Features*** panel's ***Active Report List***, double-click on the features to add to the report in the ***Not In Report*** section. You can also use the Add arrows.
5. Select features to add to report	Select the following features to be added to the report: ***Bolt Circle***, ***Circle 1***, ***Circle 3***, ***Circle 5***, ***Circle 7.***
6. Remove features from report	On the ***In Report*** section at the top of the list, select the Down arrow to the left of ***World*** to remove it from the ***In Report*** section. You do not need to add ***World*** to this report because it represents the device's coordinate system.
7. Preview report	Scroll through the report using the slider bar on the right side of the report page.
8. View circle data	Notice how the data for the circles is reported. Once nominals have been entered, the report will display deviation and whether the feature is in or out of tolerance. For more information on nominals, see the Nominals and Tolerances module.



Saving a Customized List of Features

You can save the customized data list that you create, so you can recall the report you created without having to recreate the customized list.

To do this, use CAM2 Q's ***Active Report List*** feature.

ACTIVITY: Saving A Customized List Of Features

For this activity, you will save a customized list of features.

1. New report list	On the ***Features*** panel, on the ***Active Report List***, select ***New List*** from the drop-down list.
2. Enter name	Highlight ***New List*** and type in `My Report`.
3. Press ***Enter***	Press the ***Enter*** key.

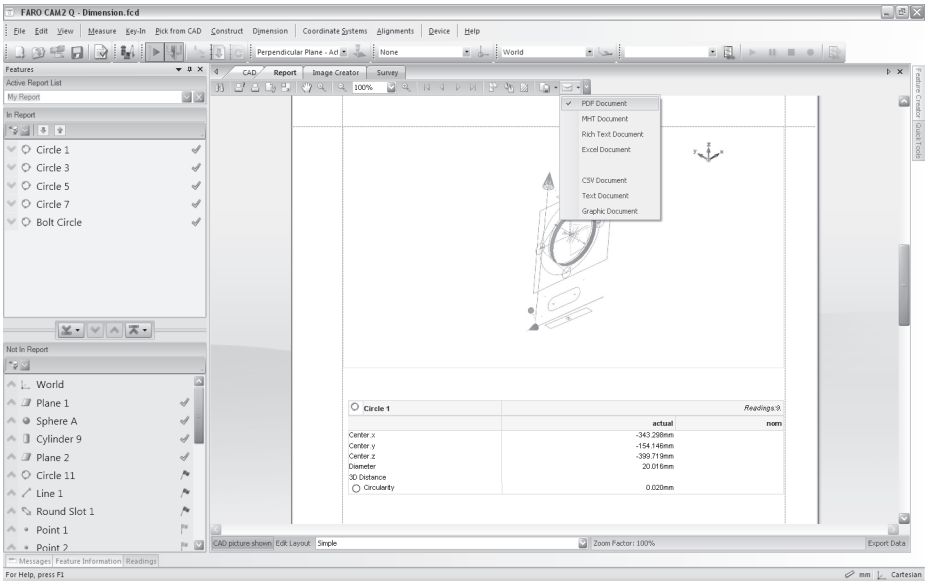
View Report Style Options

Click the ***Report*** tab to see a preview of your current report. There are several available styles of reports: ***Simple***, ***Tabular,*** and others.

When not using nominals, a ***Simple*** style report may be preferable. When nominals are present, a ***Tabular*** style report may be preferable. By auto-hiding, or pinning, the ***Feature Information*** panel you can scroll down and see the results of the inspection routine.

Save a Report

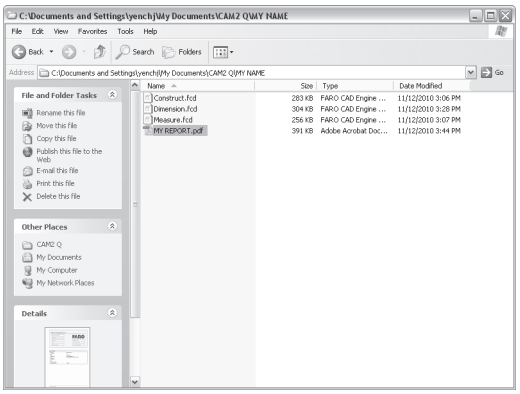
To save a report, click the ***Export Document*** drop-down icon.



From the ***Export Document*** icon drop-down list, you can save reports as PDF, HTML, MHT, Rich Text , Excel, CSV, Plain Text or a Graphics document (.jpeg, .bmp, .gif, .tiff, .png, .emf, or .wmf files).

Opening a Saved Report

To open a saved report, double-click the file name.



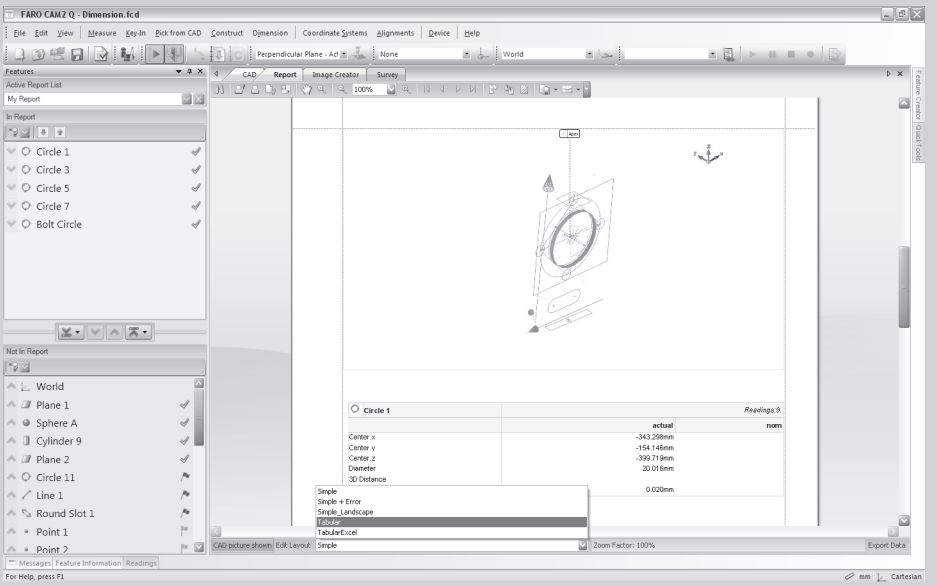
E-mail a Report

To e-mail a report, first export the report. From the ***Export Document*** icon drop-down list, you can pick the format of the report. You can e-mail reports in all of the available formats (PDF, HTML, MHT, Rich Text , Excel, CSV, Plain Text or a Graphics document).

ACTIVITY: Changing The Report View Style

For this activity, you will change the layout of the report.

1. ***Report*** tab	Click the ***Report*** tab.
2. Select report style	In the ***Edit Layout*** drop-down box, select ***Tabular***. The report now shows the data in tabular view.



☛ *Go to page 2 of the report and select **Simple** to view the differences between the two report styles.*

ACTIVITY: Saving The Report

For this activity, you will save the report.

1. Save report	From the ***Export Document*** icon drop-down list, select the ***HTML Document*** option.
2. Export options	In the ***HTML Export Option***s dialog box, click ***OK*** (We will use the default setting for this activity).
3. Your folder	Browse to your folder in ***/My Documents/CAM2 Q***.
4. Enter name	Save the report with the name `My Report`. If prompted to overwrite the existing file, click ***Yes***.
5. Automatically open file	When prompted "Do you want to open this file?" click ***Yes***. You can now view the file and close when done.

The file is now saved on the computer in the ***My Documents/CAM2 Q***/ [***your name***] folder.

ACTIVITY: Opening A Saved Report

For this activity, you will open a saved report.

1. Open ***My Computer***	Click the Windows ***Start*** button, and click ***My Computer***.
2. ***My Documents*** folder	On the left side, in the ***Other Places*** section, select ***My Documents***.
3. Your folder	In ***My Documents***, double-click the ***CAM2 Q*** folder, then double-click ***your folder.***
4. Open report	Double-click ***My Report*** to open it.

ACTIVITY: Saving The File

For this activity, you will save a CAM2 Q file.

1. ***Save As***	On the ***File*** menu, click ***Save As***.
2. Your folder	Using the drop-down menu, click ***My Documents***, click ***CAM2 Q***, then ***your folder***.
3. Enter name	In the ***File*** name box, type in `Report.fcd`
4. ***Save***	Click ***Save***.

Creating a Report Template

The default report template shows FARO's company information at the top of each report. You can create a customized report template with your company information.

- Select a report format from the combo box at the bottom of the ***Report*** panel and click the ***Edit Layout*** button. This starts the DevExpress template editor.
- Select ***File***, then ***Save As*** to save the template with a new name . This ensures that you do not replace the default CAM2 Q report template.
- Save the new template in the ***CAM2 Q, Templates, Reporting*** folder.
- Edit the fields in the ***Report*** section of the editor.
- ***Save*** the template.

ACTIVITY: Creating A Report Template

For this activity, you will create a report template with your company information.

1. ***Edit Layout***	Select a report format and click the ***Edit Layout*** button.
2. ***Save As***	On the ***File*** menu, click ***Save As***.
3. ***Reporting*** folder	Using the drop-down menu, click ***My Documents***, ***CAM2 Q***, ***Templates***, then click ***Reporting***.
4. Enter name	In the ***File*** name box, type `[your name]`.
5. ***Save***	Click ***Save***.
6. Edit the ***Report*** fields	Type your company information in the report fields.
7. ***Save***	Click ***Save***.
8. Close ***Report Designer***	Close the ***Report Designer*** by clicking the ***X*** in the upper right corner.

To view your template, select your report template in the ***Edit Layout*** drop-down box.

Feature Grouping

When you work with a large number of features, it can be helpful to organize them using a Group Folder.

You can use these folders to report features in multiple coordinate systems within the same report. You can assign a coordinate system, coordinate system type, and unit of measurement for the contents of the folder, and a unique screen view for the folder that appears in the report just above the features.

ACTIVITY: Creating A Feature Group

For this activity, you will create a feature group folder, edit its properties, capture a view, and review your current report template.

1. CAD tab	Click the CAD tab to return to the Main Window.
2. Create a New Group Folder	Click the ***Folder*** icon in the upper left corner of the ***Features*** panel.
3. Name the Group	Select the ***Group 1*** folder, double-click and type `Circles` for the name of the folder. Click ***Modify*** and then ***Close*** to continue.
4. Add Features	Select the four circles (1,3,5,7) and drag them onto the Circles folder.
5. Set Top View	Press the ***4***, ***5***, or ***6*** hot key to set the top view. ***Zoom In*** (***I*** hot key) and ***Pan*** (Arrow keys) so that the four circles fill the Main Window.
6. Set the Screen View	Select the ***Circles*** folder. Right-click and select ***Capture View Data*** from the menu.
7. Review the Report	Click the ***Report*** tab and look at the updated report. Notice the addition of the screen view of the four circles.
8. Save Report	From the ***Export Document*** icon drop-down list, select the ***HTML*** option.
9. Export options	In the ***HTML Export Option***s dialog box, click ***OK*** (We will use the default setting for this activity).
10. Your folder	Browse to ***your folder*** in ***My Documents/CAM2 Q***.
11. Enter name	Save the report with the name `My Report 2`. If prompted to overwrite the existing file, click ***Yes***.
12. Automatically open file	When prompted "Do you want to open this file?" click ***Yes***. You can now view the file and close when done.
13. ***Save As***	On the ***File*** menu, click ***Save As***.
14. Your folder	Using the drop-down menu, click ***My Documents***, click ***CAM2 Q***, then ***your folder***.
15. Enter name	In the ***File*** name box, type in `Report 2.fcd`.
16. ***Save***	Click ***Save***.

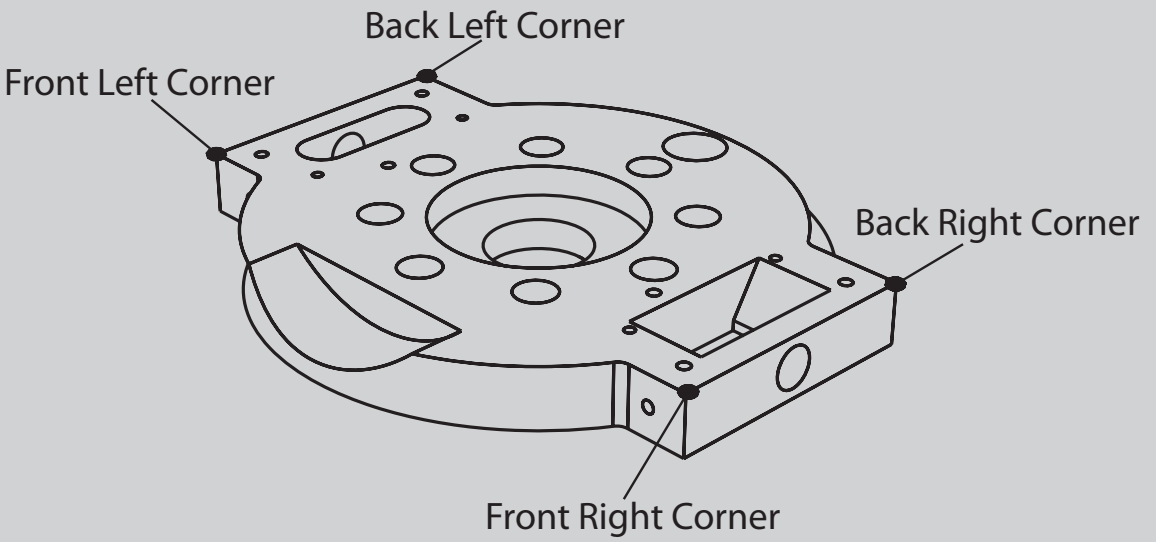
Lesson 5: Review Activities

Objective: Apply the Basic Measurement Strategy

Complete each of the Review Activities. When you are finished, have your instructor review and initial your work. If you have any questions, first refer to the previous modules, next use the ***Help*** menu. If neither of these provide the information you need, ask the instructor.

ACTIVITY: Construct Point

- Open a new file and measure the features you need to construct a point for each corner of the demo part appropriately naming each: ***Back Right Corner***, ***Back Left Corner***, ***Front Right Corner***, and ***Front Left Corner***.



ACTIVITY: Dimension Point To Point

- Determine the length from the ***Front Right Corner*** to the ***Back Left Corner*** of the demo part. Name it ***Length 1***.
- Determine the length from the ***Front Left Corner*** to the ***Back Right Corner*** of the demo part. Name it ***Length 2***.

ACTIVITY: Report

- Add ***Length 1*** and ***Length 2*** to the Report section.
- Export and save a PDF of the report. Name it ***My First Report***.

All Activities Complete

Instructor initials: _____

Date: _____

CAM2Q

Module 4:
Move Device

The CAM2 Q software and your measurement device give you many possibilities for measuring parts. The *Move Device* function was designed with complete portability in mind. You may not always have the same size part to measure; you may have to measure a small part one day and a large part the next; or your device or your part may move in between or during a measurement session. In this module, you will learn how to move your measurement device and continue measuring.

Prerequisite: Basic Measurement.

MODULE OBJECTIVES

You will be able to:

- You will be able to move the device and continue to measure a part.
- Determine how many device positions are needed.
- Determine appropriate locations for the device.
- Verify the hardware setup.
- Determine appropriate action to take when relocation results are not within the required accuracy of the measurement job.

CAM2Q

SCENARIO: MOVE DEVICE

Now we need to measure a bigger part. You are going to have to move the measurement device to get all of the measurements we need. If you need help, try looking for the information in the CAM2 Q ***Help*** menu or on FARO's website.

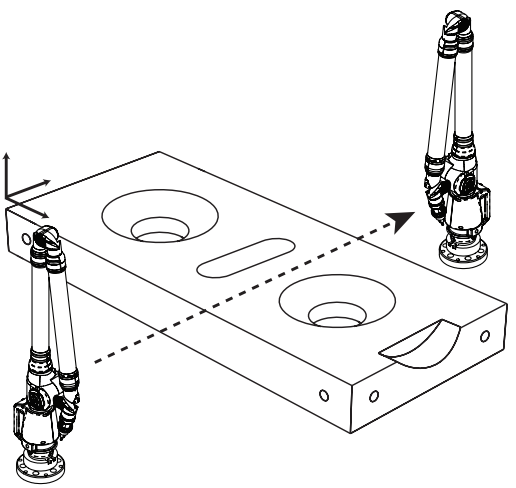
I have measured from only one device position so far , and I have to move the measurement device to continue measuring my part. What are the things I need to think about before I move the measurement device?

Strategy: Move Device

- Before measuring, how do you determine the best positions for the device?
- Can you ensure a stable mounting at every device position?
- How many device positions do you need?
- Will you be able to reach all the features to be measured?
- Which features will be re-measured from each device position?

Move Device

Move Device is a function that allows you to move the device around the part while maintaining the working coordinate system.



When To Use Move Device

- When portions of the part are beyond the FaroArm's reach and you cannot measure all features from one location, this function enables you to move the FaroArm and measure the remainder of the part.
- You want to move the device if the relationship between the part and the device changes. For example, the device or part may move (bumping etc.) or the part may experience change in size due to change in temperature (temperature drift). In these instances, you can use ***Move Device*** to quickly reposition the device into a known orientation relative to the part.

Placement of the device is important. If possible, it is best to measure from only one device position. Use the Move Device command only when necessary.

Feature Window

Monitoring the Relationship Between the Part and the Device

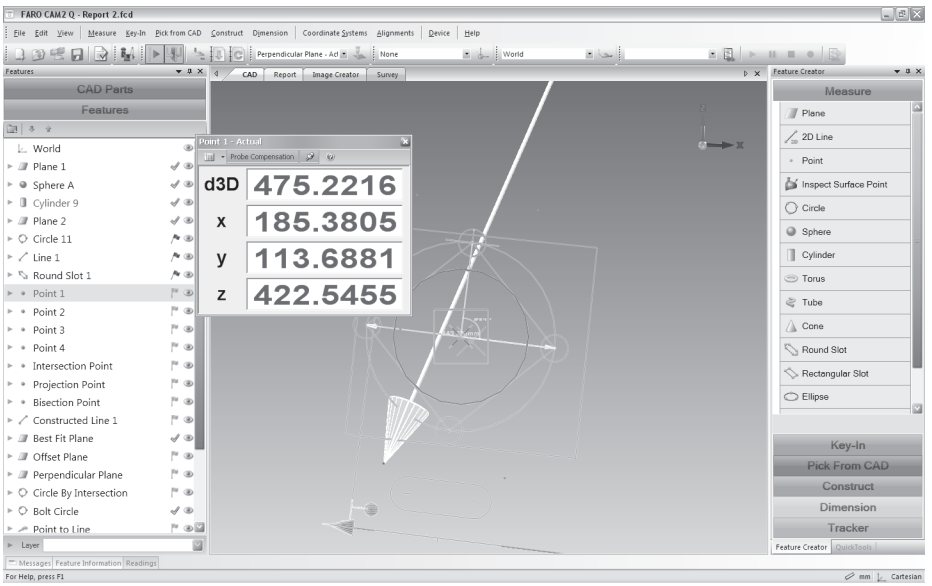
The ***Feature*** window shows a ***Digital ReadOut (DRO)*** of a specific feature you choose. The ***DRO*** in this window shows you where the feature is compared to where the probe is at that time. You can use the feature's ***DRO*** to detect changes in the position of the part relative to the measurement device.

This is accomplished as follows:

- On the ***Features*** panel, click a sphere or point you have measured.
- On the ***View*** menu, click ***Show Feature Window***. The ***Feature*** window appears.

When you place the probe at the point location or on the sphere's surface, the values in the window will be "0".

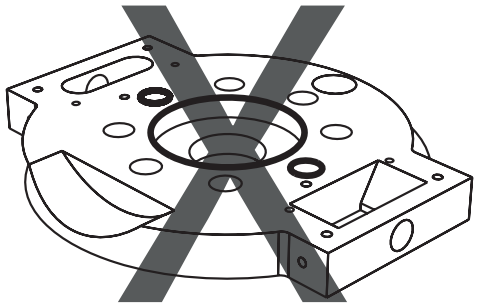
Use the *Feature window* frequently during a measurement session to check for movement, especially when working from a tripod.



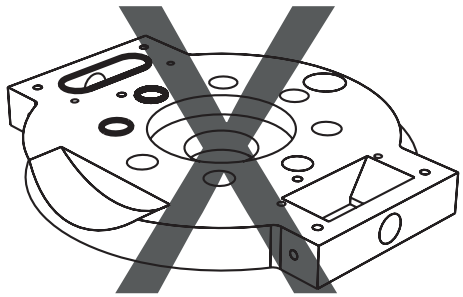
It is good practice to periodically place the probe at the point location or on the sphere's surface If the values are not "0", it means the relative position of the feature/part to the device has changed.

Move Device: Basic Concepts

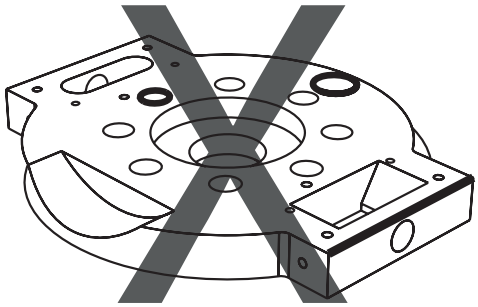
- You need a minimum of three points, or three "point-reducible" features, to complete the ***Move Device*** function. This means you can use points, circles, or spheres, or any combination.
- The three features you choose to measure cannot lay in a straight line. There must be some spacing between the features; think triangle.



- The features you choose should encompass at least 75% of the part. For example, if your part is 48 x 22 x 18 inches, features that encompass only 5 x 5 x 5 inches of the part are not an ideal choice.

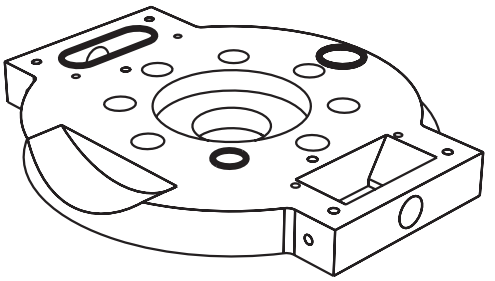


- The area local to those points will mathematically fit well, but the locations of any features you measure far away from the relocation points will have a greater uncertainty. The right edge of the part is also not a point-reducible feature.



While the minimum set for moving a device is three point-reducible features, for best results use four or more features that are not in the same plane.

- For the demonstration part, these features are one set that would be appropriate.

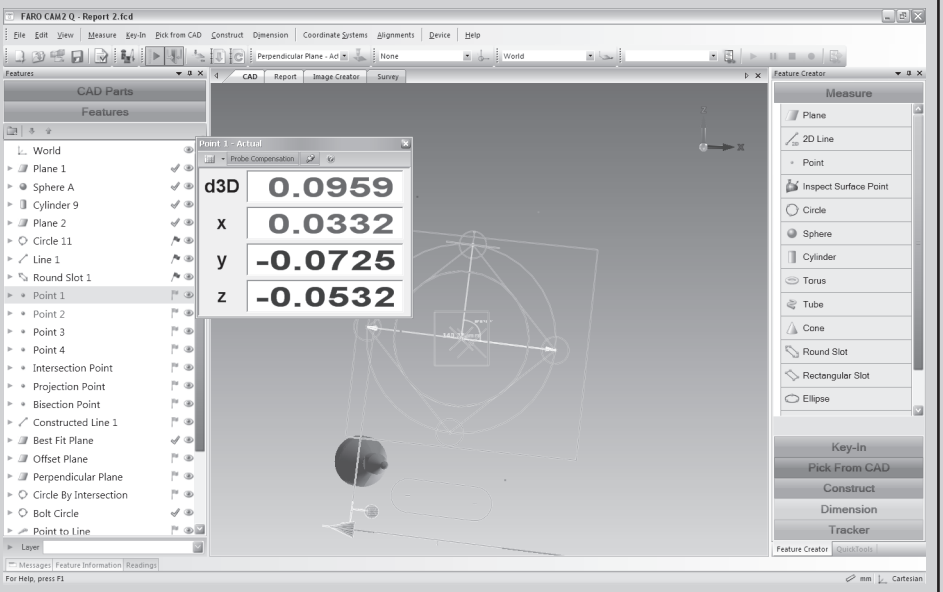


PRE-ACTIVITY:

Option 1 If training files are available	Option 2 If starting with a new file
Open Report.fcd	Open a new file
	Add and measure: • Point 1 • Point 2 • Point 3 • Point 4

ACTIVITY: Executing A Move Device Position

1. Open the ***Report.fcd*** file	On the ***File*** menu, click ***Open***. From your folder, open ***Report.fcd***.
2. Show the F***eature*** window for ***Point 1***	On the ***Features*** panel, click ***Point 1***. On the ***View*** menu, click ***Show Feature Window***. ☛ *or press the **D** hot key.*



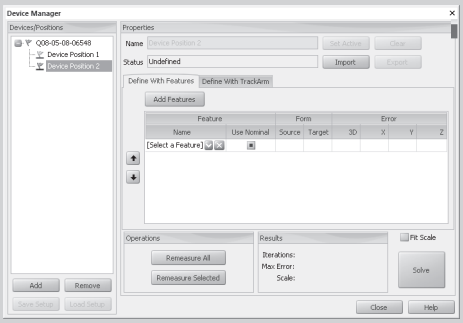
3. Verify the ***Point 1*** measurement	Look in the ***Feature*** window for ***Point 1*** by moving the probe to that location.

Has the relationship between the device and the part changed?

☛ *Keep in mind that Point 1 was measured earlier. At that time, the values were "0".)*

If the values have not changed, move either the part or device slightly and re-check the ***Feature*** window for Point 1 again. Notice that the values are no longer "0" , and it is now necessary to realign the device and part. Whether the relationship between the part and the device changes intentionally or unintentionally, the process for realigning the device to the part is the same.

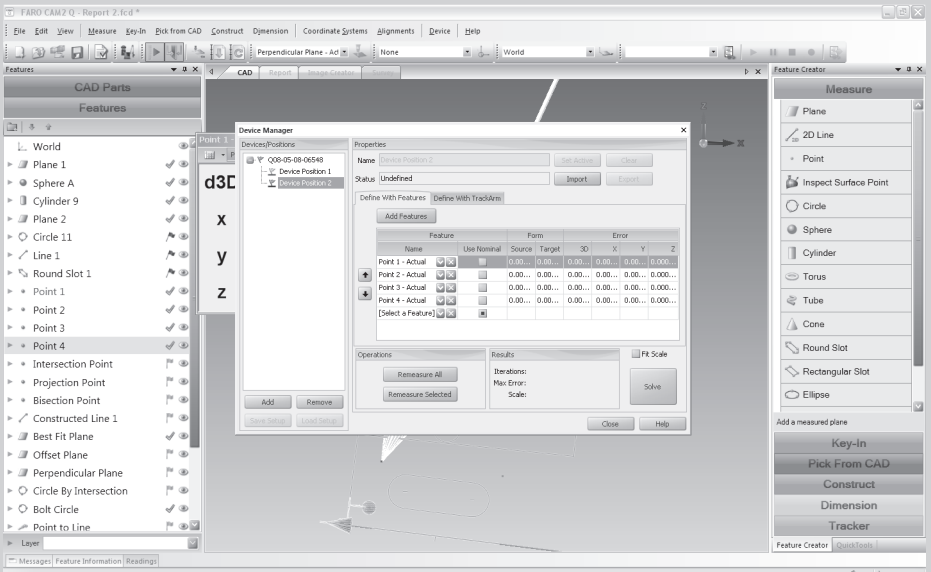
4. Execute the ***Move Device*** command	On the ***Device*** menu, click ***Move Device***. or press the ***M*** hotkey.



5. Switch to ***Measure Mode***	If necessary, click the ***Measure/Template Mode*** icon so that you are in ***Measure*** mode.
6. Select ***Points 1*** through ***4***	On the ***Features*** panel, select ***Points 1*** through ***4***. (***CTRL*** + click Point 1, Point 2, Point 3, Point 4)
7. Remeasure all features	On the ***Device Manager*** window's ***Properties*** section, click ***Add Features***, then click ***Remeasure All***.

You will then be prompted to remeasure ***Points 1*** through ***4***, which allows the software to re-position the device.

☛ *Make sure to re-measure the points in the correct order (i.e. when the software prompts you to measure Point 1, make sure it is Point 1 you are measuring). Measuring in the wrong order will result in inaccurate re-positioning of the device.*



8. Solve the new device position	Click ***Solve***. The software mathematically realigns the device to the part. Click ***Close*** to finish.
9. Verify the ***Point 1*** measurement	Look in the ***Feature*** window for ***Point 1*** by moving the probe to that location. Notice that the values are "0.0".

10. Verify other points	In the ***Feature*** window click the ***Lock to Feature*** (pin icon) button. Move the probe to the other three point locations. Notice that the values are "0.0".
11. Save the file as ***Move Device. fcd***	On the File menu, click ***Save As***. Browse to your folder. Name the file `Move Device.fcd` and click ***Save***.

Reviewing Device Positions

If you just need to review and edit existing device positions, use the ***Device*** > ***Device Manager*** command. Using the ***Device*** > ***Move Device*** command will open the ***Device Manager*** panel AND add a new device position.

CAM2Q

Module 5:
Coordinate Systems

So far, you have learned the basic strategy to follow when performing a measurement job: *Measure*, *Construct*, *Dimension*, and *Report*. You have also learned about some of the environmental and calibration considerations when you measure.

Now, we will cover creating coordinate systems. Using the features that you construct, a coordinate system gives your measurements a framework for defining location. Using the appropriate coordinate system as you measure parts plays a critical role in the interpretation of the data. Creating coordinate systems on a part requires analyzing the part and determining where the coordinate system or systems should originate. In this module, we will also determine the best features with which to create your coordinate system.

Prerequisite: Basic Measurement.

MODULE OBJECTIVES

You will be able to:

- List at least two reasons for creating coordinate systems
- Analyze requirements to determine effective coordinate system location
- Identify the effect on the data of the chosen primary axis for the coordinate system
- Identify the attributes of a repeatable coordinate system
- Recognize the impact of feature selection on the data
- Create a coordinate system using any of three techniques

CAM2Q

SCENARIO: ADDING COORDINATE VALUES

The boss says, " The report is great. Thanks for your hard work; the customer will be pleased with how quickly we provided the data. Can we add coordinate values?"

I need a coordinate system on my part to provide a framework for my measurements. Where should I position my X, Y, and Z axes? Do I need more than one coordinate system for reference?

Strategy: Coordinate Systems

- **What are the dimensions, lengths, and feature positions of the part that I need to report?**
- **Where is the most logical placement for the coordinate system?**
- **Which features do I need to measure and/or construct to create the coordinate system?**
- **Are multiple coordinate systems required?**

What is a Coordinate System

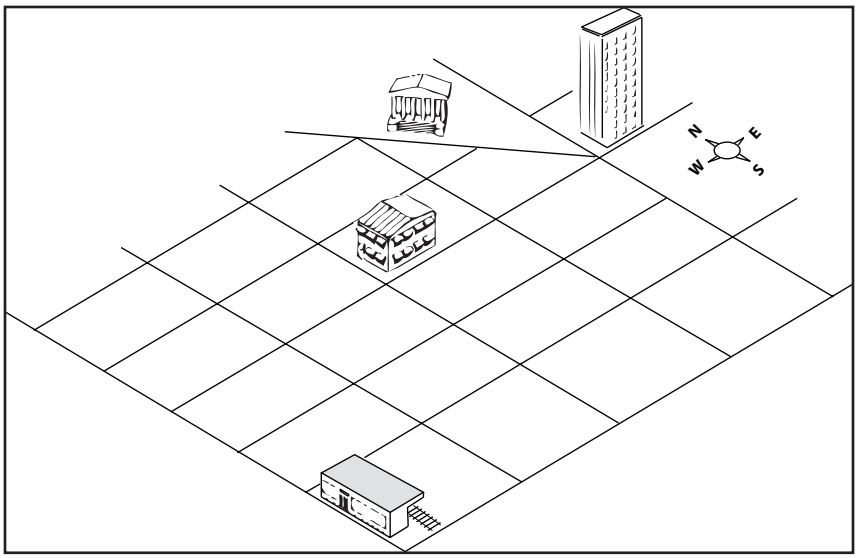
A coordinate system is a frame of reference that allows you to find the unique position of a point in a plane or in space, and to see the relationship between two points in space.

Why Use a Coordinate System

Placing a part into a coordinate system enables you to:

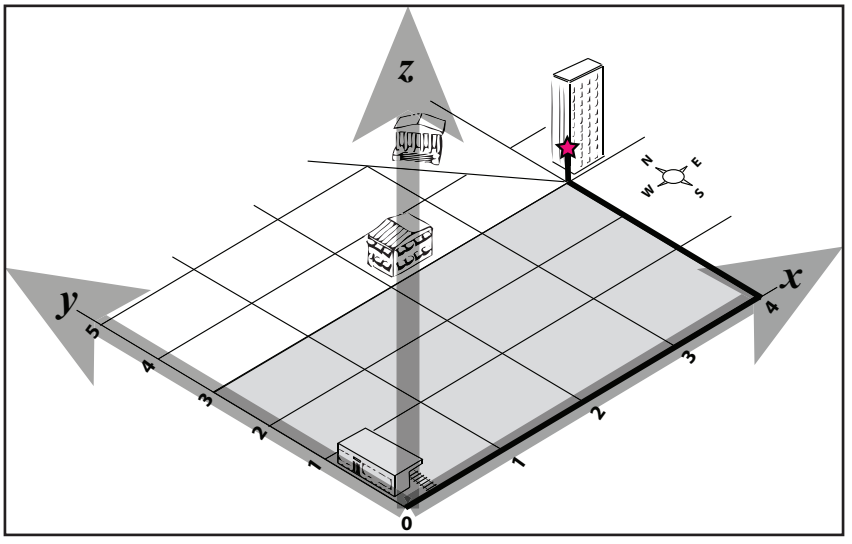
- Effectively report measured features – Since you have a unique position identified for each point and feature, your report can very clearly and specifically describe the location and dimensions of the features and part.
- Establish a reference for all part measurements so the data on the part ia aligned to the same reference points in a blueprint or CAD file. This enables straightforward comparisons between feature measurements and nominals.
- Accurately report the condition of a part to someone analyzing your measurement file who does not have access to the physical part.

Think of a three dimensional coordinate system as a three dimensional map:



Imagine that you want to give someone directions from the train station to the 3rd floor of the tall building in the map above.

If you add three dimensional coordinates to the map, you can use those coordinates to describe how many streets to go in the "X" direction (4); then how many streets to go in the "Y" direction (3); and finally and how many floors to go up in the "Z" direction (3).

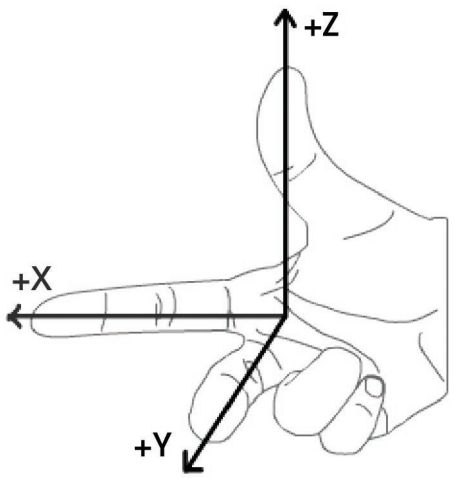


Right-Hand Rule

Cartesian coordinates describe the position of points in space relative to three mutually perpendicular axes X, Y, & Z. The intersection of these axes form the origin, or zero of the system.

Each axis emanates from the origin according to the right-hand rule. The right-hand rule defines the positive direction of each axis, such that when the hand is positioned according to the figure below, the + Z points along

the thumb, the + X axis points along the index finger, and the + Y axis points along the middle finger.



☛ *CAM2 Q does not support left-hand rule coordinate systems.*

Feature Selection

The accuracy of the measurements you take in your coordinate system relies heavily on the consistent accuracy of the machined features you use to define your coordinate system.

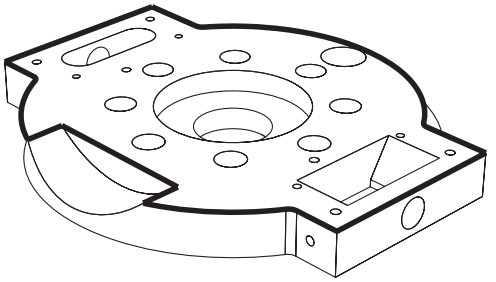
For repeatability, look for the following when selecting features to define your coordinate system:

- ***Surfaces that have a smooth finish:*** These provide the most consistent repeatable measurements.
- ***Features critical to part machining:*** These generally will be more carefully controlled than those not critical to machining and therefore will be consistent.
- ***Features that are relatively large compared to the part size.***
- ***Features that have a low measured Form value.***

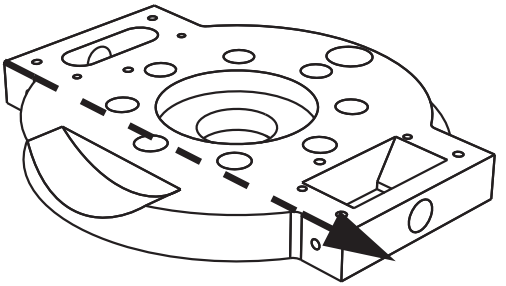
Defining Coordinate Systems Using 3-2-1

You can define coordinate systems in a variety of ways. The simplest is the 3-2-1 or Plane-Line-Point method. 3-2-1 describes the minimum number of points that you use to define the base features of a coordinate system – a ***Plane*** (3 points), a ***Line*** (2 points) and a ***Point*** (1 point).

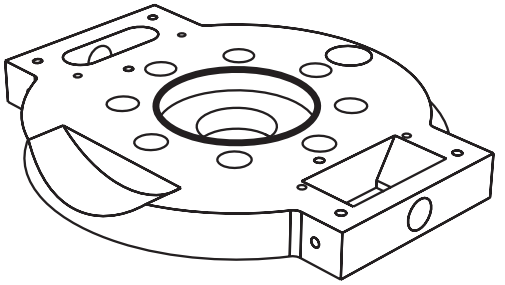
Standard construction diagrams of plane, line and point showing how they create a coordinate system:



Plane

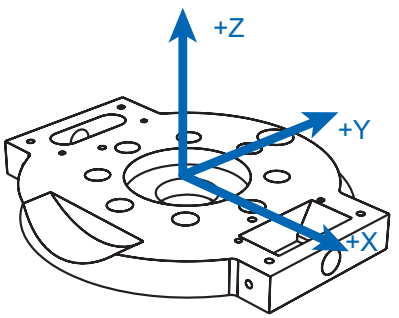


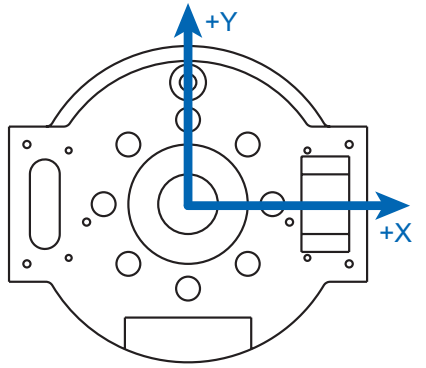
Line



Point

(Center point of the Circle)





PRE-ACTIVITY:

Option 1 If training files are available	Option 2 If starting with a new file
Open Advanced 1.fcd	Open a new file
Measure all features	Add and measure/construct: ● Plane 1 ● 2D Line 1 (across the front of the part on the handle side) ● Circle 1 ● Circle 3 ● Circle 5 ● Circle 7 ● Cylinder 9 ● Sphere A ● Bolt Circle ● Intersection Point (of Plane 1 and Cylinder 9)

ACTIVITY: Create A Coordinate System

For this activity, you will construct a coordinate system using 3-2-1 (plane, line, point).

1. Start ***Coordinate Systems*** command	On the ***Coordinate Systems*** menu, click ***Create Using the Wizard***. ☛ *or press the* ***C*** *hot key.*
2. Dialog box	The ***Construct Coordinate System Using The Wizard*** dialog box appears.
3. Enter feature name	In the ***Name*** box, enter `3-2-1`.
4. ***Basic*** tab	Click the ***Basic*** tab.
5. Choose a plane for ***+Z***	In the ***Plane Feature*** list, select ***Plane 1***. In the ***Axis*** list, select ***+Z***.
6. Choose a line for ***+X***	In the ***Line Feature*** list, select ***2D Line 1***. In the ***Axis*** list, select ***+X***.

You have now selected what will be the Z and X axes of your parts coordinate system.

7. Choose a circle for the origin	In the ***Point Feature*** list, select ***Bolt Circle***.

This identifies your origin, or the point where your X, Y, and Z will all be "0".

8. Align with ***World***	Select the ***Align with*** box. From the ***Align with*** list, select ***World***.

This aligns the coordinate system you are creating on the part with coordinate system of the device. From now on, any measurements you take will use the part coordinate system as a reference.

9. ***OK***	Click ***OK***.

Notice the ***Active Coordinate System*** toolbar at the top of the screen. Your coordinate system, 3-2-1, now shows in the drop-down list and replaces World as the active coordinate system. When you work with multiple coordinate systems, this is where you will control which coordinate system is active.

10. ***Save As***	On the File menu, click ***Save As***.
11. Your folder	Using the drop-down menu, click ***My Documents***, ***CAM2 Q***, and then double-click on ***your folder***.
12. Enter name	In the ***File*** name box, type in `CS1`.
13. ***Save***	Click ***Save***.

If necessary, set the SW Isometric View and Zoom All to see your complete file by pressing the ***E*** and ***8*** hot keys. These commands are also available in the ***View*** menu.

ACTIVITY: View Report

For this activity, you will view a report.

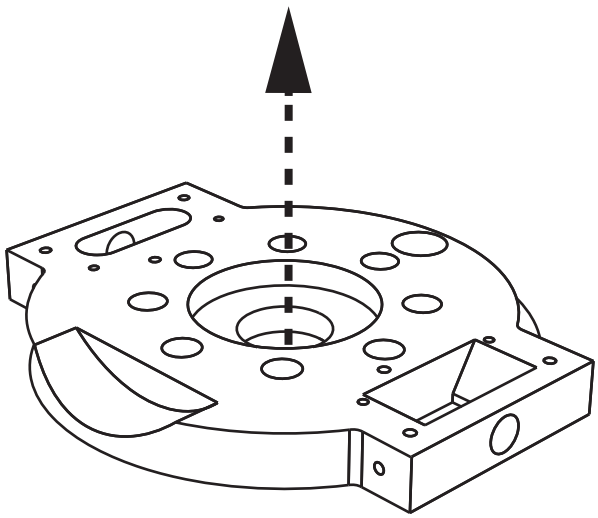
1. ***Report*** tab	Click the ***Report*** tab.
2. Choose report list	On the ***Features*** panel's, ***Active Report List***, select ***My Report.*** ☛ *Or, you may add the features to the In Report section.*
3. View report	Observe the feature data on the report. The X, Y, and Z values are now relative to the coordinate system origin you created for the part.
4. ***CAD*** tab	Click the ***CAD*** tab.

Create A Coordinate System

As you choose features for each of these coordinate systems, notice the difference between the Measured and Constructed features by the geometry icon to the left of the feature name.

Using a Line as Primary

If the longest, most important feature on your part is a line, you can use the ***Line as Primary*** option for creating your part coordinate system.



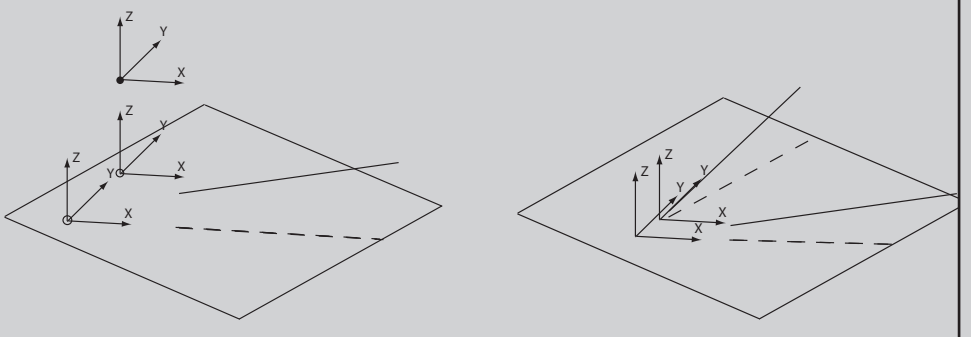
ACTIVITY: Create A Coordinate System Using A Line As Primary

For this activity, you will construct a coordinate system using a Line as Primary.

1. Start ***Coordinate Systems*** command	On the ***Coordinate Systems*** menu, click ***Create Using the Wizard***. ☛ *or press the* ***C*** *hot key.*
2. Dialog box	The ***Construct Coordinate System Using The Wizard*** dialog box appears.
3. Enter feature name	In the ***Name*** box, enter `Line as Primary`.
4. ***Advanced*** tab	Click the ***Advanced*** tab.
5. Choose a line for ***+Z***	In the ***Primary*** section, select the ***Line*** button. In the first feature list, select ***Cylinder 9***. In the ***Axis*** list, select ***+Z***.
6. Choose a line for ***+X***	In the ***Secondary*** section, select the ***Line*** button. In the first feature list, select ***2D Line 1***. In the ***Axis*** list, select ***+X***.
7. Choose a point for the origin	In the ***Origin*** section feature list, select ***Circle 3***.
8. Align with ***World***	Clear the ***Align with*** box.
9. ***OK***	Click ***OK***.

ACTIVITY: View Report

For this activity, you will view a report.

1. ***Report*** tab	Click the ***Report*** tab.
2. Choose report list	On the ***Features*** panel, in the ***Active Report List***, select ***My Report***.
3. View report	Observe the feature data on the report. The X, Y, Z values have changed because your coordinate system has changed.
	
4. ***CAD*** tab	Click the ***CAD*** tab.

Create A Coordinate System (Origin Options)

Origin Options

There are several origin options when creating a coordinate system.

- Hold Position: the coordinate system is at the center of the feature.
- Hold to Primary: the coordinate system is at the closest point on the primary feature from the origin feature.
- Hold to Secondary: the coordinate system is at the closest point on the secondary feature from the origin feature.
- Hold to Key-In: the coordinate system corresponds to the location (primary, secondary and origin) keyed-in.

ACTIVITY: Create A Coordinate System (Origin Options)

For this activity, you will construct a coordinate system using different origin options.

1. Start ***Coordinate Systems*** command	On the ***Coordinate Systems*** menu, click ***Create Using the Wizard***. ☛ *or press the **C** hot key.*
2. Dialog box	The ***Construct Coordinate System Using The Wizard*** dialog box appears.
3. Enter feature name	In the ***Name*** box, enter `Origins`.
4. ***Advanced*** tab	Click the ***Advanced*** tab.
5. Primary	In the ***Primary*** section, select the ***Plane*** button. Select ***Plane 1***. In the ***Axis*** list, select ***+Z***.
6. Secondary	In the ***Secondary*** section, select the ***Line*** button. Select ***2D Line 1***. In the ***Axis*** list, select ***+Y***.
7. Choose origin	In the ***Origin*** section feature list, select ***Sphere A***. Select ***Hold Position***. Clear the ***Align with*** box.
8. ***OK***	Click ***OK***.

You now have a coordinate system with the origin at the center of the sphere. Now, let's modify the origin.

9. ***Origins*** properties	On the ***Features Panel***, select ***Origins***, double-click to edit the feature properties.
10. Choose origin	Select ***Hold to Primary***.
11. ***Modify***	Click ***Modify*** and move the window so you can see the coordinate system location on the screen.

You now have a coordinate system with the origin at the closest point on the plane to the sphere. Press Hot Key ***6*** for a top view. Now, let's modify the origin again.

12. Choose origin	Select ***Hold to Secondary***.
13. ***Modify***	Click Modify and then click ***Close***.

You now have a coordinate system with the origin at the line.

ACTIVITY: View Report

For this activity, you will view a report.

1. ***Report*** tab	Click the ***Report*** tab.
2. Choose report list	On the ***Features*** panel's ***Active Report List***, add all features to the ***In Report*** section.
3. View report	Observe the feature data on the report.
4. ***CAD*** tab	Click the ***CAD*** tab.

ACTIVITY: Saving The File

For this activity, you will save a CAM2 Q file.

1. ***Save As***	On the ***File*** menu, click ***Save As***.
2. Your folder	Using the drop-down menu, click ***My Documents***, ***CAM2 Q***, then double-click ***your folder***.
3. Enter name	In the ***File*** name box, type in `CS2.fcd`.
4. ***Save***	Click ***Save***.

CAM2Q

Module 6:
Nominals and Tolerances

You have just created a coordinate system. Now, you will learn how to compare your part's actual measurements to the design specifications.

Prerequisite: Coordinate Systems.

MODULE OBJECTIVES

You will be able to:

- Incorporate nominals and tolerances into the measurement process and report any variances.
- Identify the appropriate XYZ values of the nominals.
- Identify the change in tolerance due to feature size or decimal places.
- Change and save tolerance options.
- Use alternative reporting options.
- Use alternative nominal association and automatic feature construction workflow.

CAM2Q

SCENARIO: NOMINALS AND TOLERANCES

You gave the report to your boss and he says, " This looks great, but how does this compare to the customer's blueprint? How can I tell if the part is manufactured according to the specifications on the blueprint?

I now have to add nominals and tolerances in order to compare actual measurements with design specifications. How can I incorporate nominals and tolerances into my measurement and reporting process?

Strategy: Nominals and Tolerances

- What are the best methods for entering nominals and tolerances for this part?
- How will nominals and tolerances help me report the data?

Definitions: Nominals & Tolerances

Nominals are the designed values of the part. The numbers specified on a blueprint are nominal values.

Tolerances are the acceptable difference between the actual measured value and the nominal value of a feature.

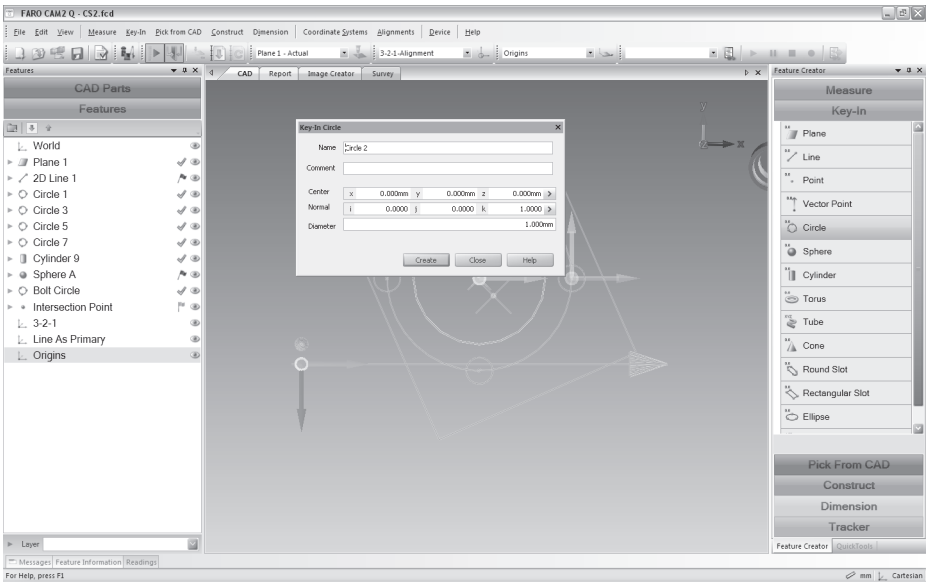
You determine the quality of a part by comparing your actual values to the nominal values within a tolerance range.

Entering Nominals: Circle Example

Prior to entering nominals, it is necessary to select the appropriate coordinate system for referencing the nominals.

There are three nominals that you can key-in for a circle:

Center:	A positional element defining the location of the circle relative to the coordinate system;
The Normal:	The orientation of the circle relative to the coordinate system; and
Diameter:	The size of the circle.



You can type a value in any unit, and CAM2 Q will convert the value to the default unit value. For example, the measurement file's unit of measurement is millimeters but the diameter is 10inches. Type 10in and CAM2 Q will automatically convert the value to 254 millimeters.

PRE-ACTIVITY:

Option 1 If continuing from the previous module with no device or part setup changes	Option 2 If your device or part setup has changed	Option 3 If starting with a new file
Open CS1.fcd	Open Advanced 1.fcd	Open a new file
	Measure: ● Plane 1 ● Circle 1 ● Circle 3 ● Circle 5 ● Circle 7 Create CS1 from ***ACTIVITY: Create Coordinate System in the*** Coordinate Systems module.	Add and measure/ construct: ● Plane 1 ● Circle 1 ● Circle 3 ● Circle 5 ● Circle 7 ● Bolt Circle Create CS1 from ***ACTIVITY: Create Coordinate System in the*** Coordinate Systems module.

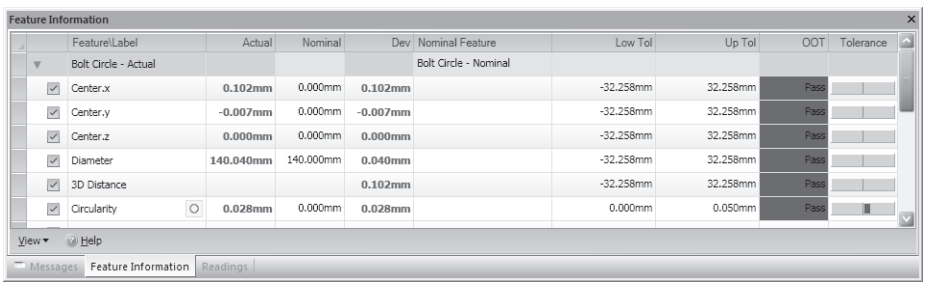
The Feature Information Panel

The ***Feature Information*** panel displays nominal (or target) values, actual values, and variations for a variety of positional and statistical elements. Key values are:

- X, Y, Z
- 3D Distance
- Diameter
- Form (GD&T tolerance)

The ***OOT*** (Out of Tolerance) column indicates whether the feature measured is within acceptable tolerance limits.

- ***Pass*** (1) and the color green indicate that the feature is within tolerance.
- A ***number*** (2) (indicating how much out of tolerance the value is) and a red or blue color indicates how much the value is out of tolerance. If the feature is out of tolerance on the high side (3) the color is red; if the feature is out of tolerance on the low side (4) the color is blue.



ACTIVITY: Entering Nominal Data

For this activity, you will select a coordinate system and key in Nominals.

1. Highlight ***Circle 1***	On the ***Features*** panel, click ***Circle 1***.
2. Expand Feature	To the left of the feature name, click the right arrow to expand ***Circle 1***. You now see the actual and nominal features of ***Circle 1***. Notice that the nominal feature is gray because it does not exist yet.
3. Open the ***Feature Information*** panel	Click the ***Feature Information*** tab.
4. Key In nominals for Circle 1	In the ***Center x*** Nominal cell, enter 0.00mm In the ***Center y*** Nominal cell, enter 70.00mm. In the ***Center z*** Nominal cell, enter 0.00mm In the ***Diameter*** Nominal cell, enter 20.00mm.
5. Highlight ***Circle 3***	On the ***Features*** panel, click ***Circle 3***.
6. Key In nominals for Circle 3	In the ***Center x*** Nominal cell, enter 70.00mm In the ***Center y*** Nominal cell, enter 0.00mm. In the ***Center z*** Nominal cell, enter 0.00mm In the ***Diameter*** Nominal cell, enter 20.00mm.
7. Highlight ***Circle 5***	On the ***Features*** panel, click ***Circle 5***.
8. Key In nominals for Circle 5	In the ***Center x*** Nominal cell, enter 0.00mm In the ***Center y*** Nominal cell, enter -70.00mm. In the ***Center z*** Nominal cell, enter 0.00mm In the ***Diameter*** Nominal cell, enter 20.00mm.
9. Highlight ***Circle 7***	On the ***Features*** panel, click ***Circle 7***.
10. Key In nominals for Circle 7	In the ***Center x*** Nominal cell, enter -70.00mm In the ***Center y*** Nominal cell, enter 0.00mm. In the ***Center z*** Nominal cell, enter 0.00mm In the ***Diameter*** Nominal cell, enter 20.00mm.

You have just keyed in the nominals for the four circles that you used to construct the bolt circle in an earlier exercise. Now, you will key in the nominals for the ***Bolt Circle*** you created in the earlier exercise.

11. Highlight ***Bolt Circle***	On the ***Features*** panel, click ***Bolt Circle***.

12. Key In nominals for Bolt Circle	In the ***Center x*** Nominal cell, enter `0.00mm` In the ***Center y*** Nominal cell, enter `0.00mm.` In the ***Center z*** Nominal cell, enter `0.00mm` In the ***Diameter*** Nominal cell, enter `140.00mm.`

Changing Tolerances Settings

The Out of Tolerance (***OOT)*** values you see are the result of default tolerances. You can manually change these tolerances or they can be set prior to taking any measurement.

- On the ***Edit*** menu, click ***Preferences***, and then click ***Tolerance*** and ***Geometries*** tab.

ACTIVITY: Changing Tolerance Settings

For this activity, you will change tolerance settings in the ***Feature Information*** panel.

1. Highlight the ***Bolt Circle***	On the ***Features*** panel, click ***Bolt Circle***.
2. Open the ***Feature Information*** panel	Click the ***Feature Information*** tab.
3. Enter an Up Tolerance of ***.00001***	On the ***Feature Information*** panel's ***Circularity*** row, enter `.00001` in the ***Up Tol*** column, and press Enter.

Notice that the cell in the ***OOT*** column is displaying a color and a number. Since you set the tolerance limit very low, the value is beyond the tolerance limit. The color red indicates that the form value is higher than the tolerance you specified and the number indicates how far the value is out of form tolerance.

4. Change the Up Tolerance to ***.5***	In the ***Circularity*** row, in the ***Up Tol*** column, enter `.5` and press Enter.

Notice that the cell in the ***OOT*** column is displaying the color green and the word ***Pass***. Since you set the tolerance limit very high, the value is well within the tolerance limits.

5. Save your file with a new name	On the ***File*** menu, click ***Save As***. Enter `Nominals 1.fcd` as the file name.

Setting Tolerances in a Measurement File

The default tolerances for each feature type are a preference in the measurement file. Use the ***Preferences*** command to change tolerance values for future features or apply a set of tolerances to *all* existing features. CAM2 Q also supports DIN/ISO tolerance schemes.

You can also use the ***Copy Tolerances*** and ***Paste Tolerances*** commands from the ***Edit*** menu to copy tolerances from a feature and paste to another similar feature. For example, modify the tolerances of a circle, copy, and paste to other circles in your drawing file.

Save time by setting the feature tolerances before adding any features to the measurement file.

ACTIVITY: Copying Tolerances

For this activity, you will change tolerances for a feature and copy them to other similar features.

1. Edit the tolerances a circle	Select ***Circle 1*** in the ***Features*** panel. In the ***Feature Information*** panel, clear the Z dimension and 3D Distance checkboxes. Edit the X dimension, Y dimension, and Diameter tolerances as follows: ❒ Lower Tolerance `-0.25 mm`. ❒ Upper Tolerance `+0.25 mm`.

Now you can copy the tolerances to the other three circles.

2. Copy the Tolerances	Select ***Circle 1*** in the ***Features*** panel. Right-click and select ***Copy Tolerances***.
3. Paste the Tolerances	Select ***Circle 3*** in the ***Features*** panel. Right-click and select ***Paste Tolerances***.
4. Paste to remaining circles	Repeat for circles 5 and 7.

An Alternative Workflow

In the previous activities, you added nominal values to existing measured features. However, you can also set up the measurement file with all of the nominal features before connecting the measurement device. Then use the ***Add Measurement*** function to create measured (actual) features from the existing nominal features.

PRE-ACTIVITY:

Option 2 If your device or part setup has changed	Option 3 If starting with a new file
Open Advanced 3.fcd	Open a new file

	In ***Template Mode***:
	● Add the following nominal features from the Key-In menu:
	  ● Circle 1 (0, 70, 0)
	  ● Circle 3 (70, 0, 0)
	  ● Circle 5 (0, -70, 0)
	  ● Circle 7 (-70, 0, 0)
	● Construct the nominal Bolt Circle using Circles 1,3,5,7.
	● Add the following actual features from the Measure menu:
	  ● Plane 1
	  ● Line 1
	  ● Circle 9
	● Create a Coordinate System using Plane 1, Line 1 and Circle 9.

ACTIVITY: Adding Measurements To Nominals

For this activity, you will add a measurement to the nominal bolt circle which automatically creates and associates a measured feature to each nominal circle.

1. Add readings to coordinate system features	Select Plane 1, Line 1 and Circle 9 in the Features panel. From the ***Measure*** menu, select ***Add Readings***. ☛ *You can also right-click the selected features and select the **Add Readings** command from the menu.*
2. Measure the features	Watch the prompt area of the Measurement Window for instructions to measure the plane, line and circle.
3. Add a measurement to the Bolt Circle	Select Bolt Circle. Right-click and select ***Add Measurement*** from the menu.

Add Measurement automatically adds actual features to all of the nominal features that construct the Bolt Circle. Each of the four circles (1,3,5,7) now have an actual feature associated to its nominal, and the ***Add Readings*** command automatically starts.

4. Measure each circle	Watch the prompt area of the Measurement Window for instructions to measure each of the four circles.

Now that each of the circles has solved, the Bolt Circle solves.

5. Save your file with a new name	On the ***File*** menu, click ***Save As***. Enter `Nominals 2.fcd` as the file name.
6. Review results	Select the Bolt Circle and look at the results in the ***Feature Information*** panel. Select other circles and review the results of the measurement. Change tolerance settings for any of the circles as you did earlier in this module.

CAM2Q

Module 7:
Review Activities

Complete each of the Review Activities. When you are finished, have your instructor review and initial your work. If you have any questions, first refer to the previous modules, next use the *Help* menu. If neither of these provide the information you need, ask the instructor.

Prerequisite: Basic Measurement, Coordinate Systems, Nominals and Tolerances.

MODULE OBJECTIVES

You will be able to:

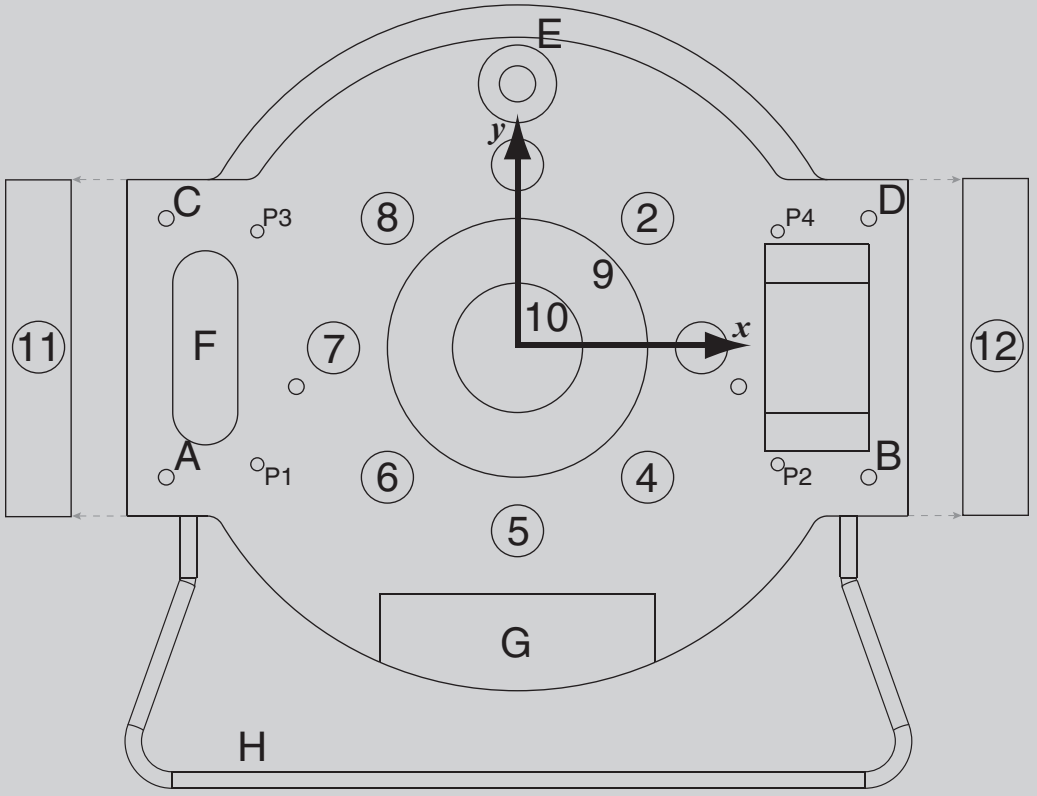
- Demonstrate ability to use Measure, Construct, Dimension, and Report software menus, add nominals, set tolerances, identify variations between nominals and actual measurements, create and configure a final report.

CAM2Q

Final Review Activities

ACTIVITY: Coordinate System

- Open a new file (make sure mm is the current measurement unit).
- Create the coordinate system in the figure below. Measure the features needed and align to the part. *(Hint: Pay attention to the origin location and direction of the 'x' axis)*



ACTIVITY: Nominals, Tolerances, And Constructions

- Measure and then enter the following nominals for circles 1, 3, 5, and 7:

  ☛ *You can also add the nominals first, and then add measurements to the nominals.*

  1 = "***0,70,0***"; Diameter = "***20mm***"

  3 = "***70,0,0***"; Diameter = "***20mm***"

  5 = "***0,-70,0***"; Diameter = "***20mm***"

  7 = "***-70,0,0***"; Diameter = "***20mm***"

- Enter a +/- tolerance of ***.25 mm***.
- Create a bolt hole circle using circles 1, 3, 5, and 7.
- The Bolt Hole Circle diameter nominal is ***140 mm***.
- Bolt Hole Circle Diameter = _____mm.
- Bolt Hole Form = _____mm.
- Are the form and diameter within tolerance? How do you know?

ACTIVITY: Constructions And Tolerances

- Construct a circle at the intersection of ***Cone E*** and the top plane of the part.
- The circle diameter nominal is ***29.99 mm.***
- Enter a +/- tolerance of ***.25 mm***.
- Circle Diameter = _____mm.
- Is it in tolerance? How do you know?

ACTIVITY: Dimensions

- Dimension the angle between circles 1, 5, and 7 using 5 as the apex.
- Angle = _____degrees.

ACTIVITY: Reports

- View the report.
- Report all elements measured and constructed above.
- Save the report as a PDF titled "***Final Review Activities***" in your folder.

All Activities Complete

Instructor Initials: _____

Date: _____

CAM2Q

Module 8:
Working with CAD

In this module, you will become familiar with using a CAD file as nominal data for your part inspection.

Prerequisite: none

MODULE OBJECTIVES

You will be able to:

- Import a CAD model
- Identify the translatable CAD model file formats
- Prepare a CAD model for easy use in inspection
- Define nominal features and surfaces from the CAD

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SCENARIO: WORKING WITH CAD

The boss says, "These results are great, but it has taken a lot of time to key-in all of the nominal information for the features. We need to measure a bigger part and we need the results quickly. You are going to have to import and prepare a CAD model to use in the inspection process. This is going to reduce the inspection time and get the results to the customer more quickly. If you need help, try looking for the information in the CAM2 Q Help menu or on FARO's web site."

"I have the CAD model. What are the things I need to think about before beginning the inspection process? Is the model ready for use in the inspection process? If not, what steps do I have to perform in order to use the CAD model?"

Strategy: Working with CAD

- How do you decide if you should use CAD?
- What is the file format of the CAD?
- Can I get the CAD into a translatable file format?
- Have I prepared the CAD to be inspected?

The use of a CAD model can help decrease inspection time and reduce error, making the inspection process easier and more accurate.

The workflow improves and inspection time is reduced by having the nominal information for each feature already calculated in the CAD file. The nominal information is extracted from the CAD model and automatically associated to the measured feature. Eliminating the need to manually key-in information increases operator productivity and reduces the risk of error.

Types of CAD Data

There are three types of CAD data: wireframe, surface, and solid.

Wireframe: These entities define the outline of a part. They can include points, circles, arcs, lines, polylines and splines. A polyline is a group of line segments stitched together to approximate a curve. A spline is a simple curve.

Surface: A surface is used to define the outer boundary of a part. Surfaces are typically used for complex curved parts such as automotive and aerospace sheet metal parts. Surfaces are very popular for creating cutting paths for CNC cutting tools.

Solid: A solid model is also used to define the outer boundary of a part, but solids have a thickness or mass. Solid modeling is popular for the design of a variety of parts. Many CAD systems start with a solid model then create the 2D drawings and 3D surfaces from the solid.

Standard CAD Data File Formats

CAM2 Q is built with the Parasolid file format (*.x_t). You should ***always*** try to export from your CAD system to this file format.

If the Parasolid file format is not available, the following CAD data file formats are included with CAM2 Q. Translators for other CAD data file formats are available and must be purchased from FARO.

IGES: IGES (Initial Graphics Exchange Standard) was created to standardize data transfer between different CAD systems. IGES works very well for surface and wireframe entities, but not for text and dimensions.

VDA: VDA is similar to IGES in that it is a standard used to translate data between different CAD systems. VDA is particularly prevalent in Europe.

When To Use CAD

- When inspecting only simple features, use a wireframe CAD model.
- When inspecting surfaces, use a solid or surface CAD model.
- When the part is complex and/or has many surfaces and features that need to be inspected, use a solid or surface CAD model.

Using CAD with CAM2

CAM2 Q translates a CAD file into a Parasolid (.x_t) file format that can be imported to a CAM2 Q measurement file using the following methods:

1. CAM2 Q automatically translates and adds most CAD files using the ***Open*** command from the ***File*** menu.

2. Translating multiple CAD files with CAM2 Q is a two-step process:

   - First, translate the CAD file using the ***FARO CAD Translator*** command from the ***File*** menu.

   - Once translated, add the file using the ***Import CAD*** command from the ***File*** menu.

   Currently, this is the only method to translate SolidWorks files.

☛ *If the features on the CAD are not drawn appropriately for inspection, they need to be constructed and relabeled before importing into* CAM2 Q.

☛ *Another time saver is preparing nominals before measurement. This results in a smoother and easier workflow for the end operator by saving time (use of a prepared CAD model) and eliminating the opportunity to enter an incorrect nominal value (use of Pick from CAD nominal vs. Key-In nominal).*

ACTIVITY: Import CAD Files

In this activity, you will import a translated CAD file:

1. New file	From the ***File*** menu, select ***New***.
2. Import CAD	From the ***File*** menu, select ***Import***, then select ***CAD***.
3. Select file	From the ***\CAM2 Q v1.5 Training Files*** folder, select ***Demo Part Q.x_t.*** Click ***Open***.

Features on the CAD model may not be drawn appropriately for inspection. For example, some drafters and CAD programs will use two arcs and two lines to define a round slot. Because the nominal geometry is not a slot, the nominal will not automatically associate to the measured round slot. In order to auto associate the nominal slot to the measured slot, we need to first create a nominal round slot on the CAD geometry.

Defining nominals prior to inspection can decrease inspection time.

ACTIVITY: Define Nominal Features And Surfaces From The CAD

In this activity you will pick features from CAD model and rename Circle 1, Circle 3, Circle 5, Circle 7 and Circle 9 from the Demo Part Q.x_t CAD file. Use the map of the training part to determine the locations of the circles.

1. ***Pick from CAD***	On the ***Feature Creator***, click ***Pick From CAD***.
2. ***Circle***	Click ***Circle***.
3. Add ***Circle 1***	On the ***Pick Circle from CAD*** window, check the name for Circle 1 and click on the circle's edge on the CAD model. Circle 1 appears in the ***Features*** panel. ☛ *If clicking the edge does not return a circle, continue clicking at least two more points on the edge to define a circle.*
4. Add other circles	With the ***Pick Circle from CAD*** window remaining open, first rename and then click on the edge for Circle 3, Circle 5, Circle 7 and Circle 9.
5. ***Close***	Click ***Close***.

Pick Round Slot 1 and rename it as Round Slot F from the .x_t CAD file. Use the map of the training part to determine the location of the slot.

1. ***Pick from CAD***	On the ***Feature Creator***, click ***Pick From CAD***.
2. ***Round Slot***	Click ***Round Slot***.

3. Add ***Round Slot***	From the ***Pick Round Slot from CAD*** window, click on the round slot's edge. Round Slot 1 appears in the ***Features*** panel. ☛ *If clicking the edge does not return a round slot, continue clicking three points that define the arc edge on one end of the slot and on three points that define the arc edge on the other end of the slot.*
4. ***Close***	Click ***Close***.
5. Rename feature	From the ***Features*** panel, right click on Round Slot 1 and select ***Rename***. Backspace and rename the feature as Round Slot F.

Save the CAD file that you have prepared for inspection:

1. ***Save As***	From the ***File*** menu, select ***Save As***.
2. Your folder	Browse to your folder and type `WWCAD.fcd`.
3. ***Save***	Click ***Save***.

CAM2Q

Module 9:
QuickTools Programming

In this module, you will become familiar with recording, editing, and playing a QuickTools program. You will create this program with and without the measurement device connected to your computer.

Prerequisite: Basic Measurement, Coordinate Systems, Nominals and Tolerances

MODULE OBJECTIVES

You will be able to:

- Build a repeatable measurement routine using measurements, constructions, dimensions, and reporting
- Save, open, edit, and run a repeatable measurement routine
- Design a program that performs an accurate metrological inspection of a part

CAM2Q

CAM2Q

SCENARIO: QUICKTOOLS PROGRAMMING

A typical QuickTools program takes an operator through the steps of a part program with digital pictures of the part and color-coded targets to make the program operator-friendly. QuickTools programming while recording allows you to create a program by automatically capturing measurement commands using a measurement device. QuickTools programming while not recording allows you to create a program by selecting features from the available menus without using a measurement device.

"I can create a QuickTools program which contains a set of measurement commands for repetitive part measurement. What are the things I need to think about to make the QuickTools program as effective and user-friendly as possible?

Strategy: Working with QuickTools

QuickTools allow you to save a set of CAM2 Q commands in a list to run anytime the drawing file is open, facilitating repeatable part measurement. You can create multiple QuickTools programs in a single measurement file and play them whenever you choose.

QuickTools Program Commands

QuickTools programs may contain many CAM2 Q commands; however, Construction, Dimension, Coordinate System, Alignment, and Device Position commands are not added to programs since these features are dependent on data to solve.

QuickTools Program Context and Usability

You can increase usability of the QuickTools program for all operators by including pictures of the part to be measured (including depictions of the precise locations at which to take measurements), and important textual information for any command in the program. The more detail you provide, the easier your QuickTools program will be to use.

QuickTools Panel Commands

At the bottom of the QuickTools panel is the QuickTools toolbar.

1 The ***New Program*** button creates a new QuickTools program.

2 The ***Delete*** button erases a specific command in the list of commands (or erases a program entirely).

3 The ***Play*** button executes the sequence of commands in the current program.

4 The ***Pause*** button pauses the current program.

5 The ***Stop*** button stops the current program.

6 The ***Record*** button allows you capture all future commands into the QuickTools program (until you click the ***Stop*** button).

7 The ***Lock QuickTools*** button allows you to secure the QuickTool from being edited, overwritten, or deleted from the measurement file. This lock uses a unique password.

> **CAUTION**: If you forget a password there is no way to recover it. Consider keeping a backup copy of the measurement file with a QuickTool that is not password-protected.

8 The ***Help*** button launches the QuickTools section of the CAM2 Q help file.

9 The ***Tools*** drop-down menu has special commands to add to a program.

- ***View Report*** - Displays an existing report template and report list.
- ***Add Comment*** - Adds a text message.
- ***Change Adapter*** - Changes an SMR adapter on the FARO Laser Tracker.

Programming in Measure and Template Modes

You may add commands to QuickTools programs while in either ***Measure*** mode or ***Template*** mode. This gives you the flexibility of creating a QuickTools program while simultaneously performing live measurements (***Measure*** mode), or creating a QuickTools program without using a measurement device (***Template*** mode).

This chart highlights the differences between recording QuickTools programs in ***Measure*** or ***Template*** mode.

	Recording a QuickTool in Measure Mode	Recording a QuickTool in Template Mode
Insert Measurement Feature	• Adds feature to measurement file • Adds measurement command to the QuickTool • Prompts operator to measure feature	• Adds feature to measurement file
Manually Invoke "Add Readings"	• Adds measurement command to the QuickTool • Prompts operator to measure feature	• Adds a measurement command to the QuickTool

PRE-ACTIVITY:

Option 1 If using CAD	Option 2 If not using CAD
Open a new file.	Open a new file.
Import the CAD file: ***\CAM2 Q*** ***CAM2 Q v1.4 Training Files*** ***Demo Part Q.x_t***	
Pick From CAD: • Circle 1 • Circle 3 • Circle 5 • Circle 7	Key-In nominal features in ***Template Mode***: • Circle 1 • Circle 3 • Circle 5 • Circle 7

ACTIVITY: Creating A New QuickTools Program

In this activity, you will create a new QuickTools program using the QuickTools panel.

1. Open the ***QuickTools*** panel	Click the ***QuickTools*** tab in the lower right side of the screen to open the ***QuickTools*** panel.
2. Create a QuickTools program	Click ***New Program***. A new (blank) QuickTools program will appear as ***New Program 1.***
3. Rename the QuickTools program	Rename the QuickTools program by selecting ***New Program 1*** and typing `Practice Program`.

ACTIVITY: Adding Features To A QuickTools Program In Measure Mode

In this activity, you will add measurement commands to the "Practice Program" QuickTools program in ***Measure*** mode.

1. Begin recording	Click the ***Record*** button.
2. Measure a Plane	On the ***Measure*** menu, select ***Plane.*** Measure Plane 1**.** Cancel the ***Plane*** command.
3. Rename the plane	Select the feature name and type `Datum A`.
4. Expand QuickTool	In the QuickTools panel, click the Plus Sign to expand the QuickTool program and see the commands.
5. Measure a 2D line	Press the ***F4*** hot key. Measure the edge across the front of the part. Cancel the ***2D Line*** command.
6. Rename the line	Select the feature name and type `Datum B`.
7. Measure a circle	Press the ***F10*** hot key. Measure Circle 9. Cancel the ***Circle*** command.
8. Rename the circle	Select the feature name and type `Datum C`.

An alternative method to add one or more commands to a QuickTools program is to select measured features in the ***Features*** panel and drag them onto the program.

9. Create using wizard	On the ***Coordinate Systems*** menu, click ***Create Using the Wizard***.
10. Add features	Choose ***Datum A*** as the Plane Feature, ***Datum B*** as the Line Feature, and ***Datum C*** as the Point Feature.
11. Align	Select the ***Align With*** checkbox and choose World.
12. Create	Click ***OK***.

13. Stop recording	In the ***QuickTools*** panel, click the ***Stop*** button.

Save the measurement file:

14. ***Save As***	From the ***File*** menu, select ***Save As***.
15. Your folder	Browse to your folder and type `Quicktools.fcd`.
16. ***Save***	Click ***Save***.

ACTIVITY: Playing The QuickTools Program

In this activity, you will complete the QuickTools program and play it.

1. Play the QuickTools program	Click ***Play*** to run the program.
2. Skip Document Header Information	Enter a part name. Click ***OK***.
3. Measure the part	Follow the ***Measurement Window*** panel and measure Datum A, Datum B, Datum C. Notice that the current feature highlights red in the ***Main Window***.

ACTIVITY: Adding Features To A QuickTools Program In Template Mode

In this activity, you will add measurement commands to the "Practice Program" QuickTools program in ***Template*** mode.

1. ***Template Mode***	If necessary, click the ***Measure/Template Mode*** icon so that you are in ***Template*** mode.
2. Begin recording	Click the ***Record*** button.
3. Add Measurement	On the Features panel, right-click the four circles (1,3,5,7) and select ***Add Measurement***.

Note that the measurements were added to the ***Features*** panel, but nothing has been added to the ***QuickTools*** panel. The "***Add Measurement***" is not a QuickTools program command. You will use the actual circles later in this module.

4. Add readings to the circle	On the ***Features*** panel, right-click ***Circle 1*** and select ***Add Readings.***

Note that the Add Readings command has added the "Add Readings to Circle 1" command to the QuickTools program.

5. Add readings to the circles	On the ***Features*** panel, select ***Circle 3, Circle 5,*** and ***Circle 7.*** Right-click and select ***Add Readings.***
6. Stop recording	Click the ***Stop*** button.

Since Constructions, Dimensions, Tolerancing, and Reporting are not recorded in a QuickTools program, all of this can be done before or after creating the Quick Tools program. Remember, these save as a part of the measurement file.

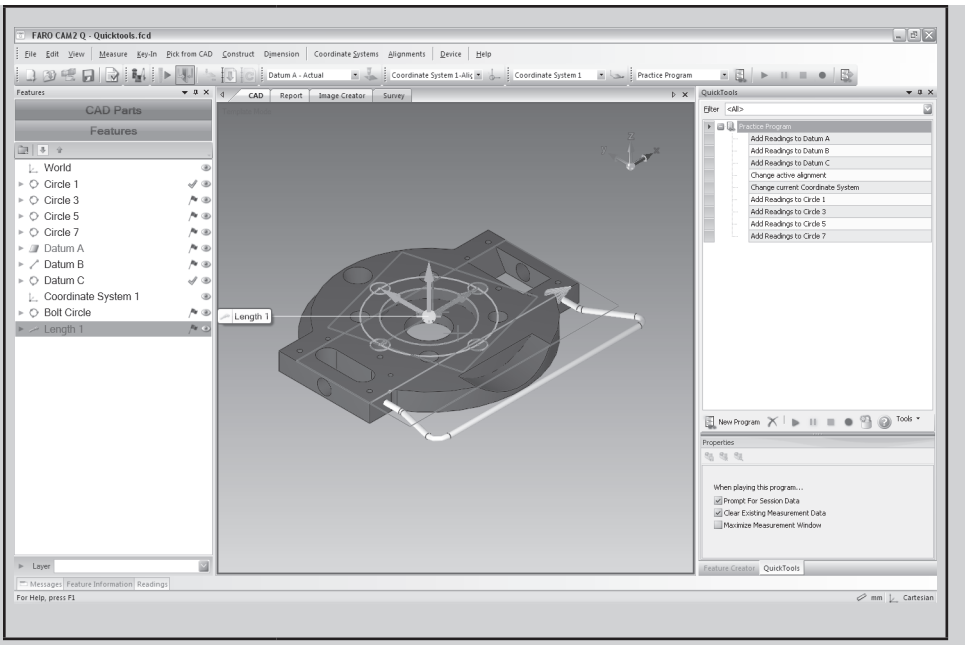
ACTIVITY: Constructions And Dimensions

In this activity, you will create a bolt circle and a length dimension.

1. Construct a Circle by Best Fit	On the ***Construct*** menu, select ***Circle***, and then select ***By Best Fit.***
2. Rename the Best Fit circle	Select the feature name and type `Bolt Circle.`
3. Select Datum A as the plane for the measured circles	If not already selected, select Circle 1 - Actual, Circle 3 - Actual, Circle 5 - Actual, and Circle 7 - Actual. Check the ***Use Plane*** checkbox and select Datum A - Actual from the drop-down list. Click ***Create*** and then click ***Close***.

Because ***Bolt Circle Actual*** uses measured (actual) circles (1,3,5,7) in the By Best Fit construction, the nominals of the measured circles are automatically used to construct the nominal by best fit circle (***Bolt Circle - Nominal***).

4. Create a dimension	On the ***Dimension*** menu, click ***Length From Features.*** Select Circle 3 - Actual treat as a point, and Circle 7 - Actual treat as a point. Click ***Create*** and then click ***Close***.



A nominal length is automatically created and associated.

ACTIVITY: Playing The QuickTools Program

In this activity, you will complete the QuickTools program and play it.

1. Play the QuickTools program	Click ***Play*** to run the program.
2. Document Header Information	Enter a part name. Click ***OK***.
3. Measure the part	Follow the ***Measurement Window*** panel and measure Datum A, Datum B, Datum C and the four circle features.

ACTIVITY: Editing Tolerances

In this activity, you will edit tolerances for some features that you want to view on the report.

1. Edit the tolerances for each circle	Select ***Circle 1*** in the ***Features*** panel. In the ***Feature Information*** panel, clear the Z dimension and 3D Distance checkboxes. Edit the X dimension, Y dimension, and Diameter tolerances as follows: ❒ Lower Tolerance `-0.25 mm`. ❒ Upper Tolerance `+0.25 mm`.

Now you can copy the tolerances to the other three circles.

2. Copy the Tolerances	Select ***Circle 1*** in the ***Features*** panel. Right-click and select ***Copy Tolerances***.
3. Paste the Tolerances	Select ***Circle 3*** in the ***Features*** panel. Right-click and select ***Paste Tolerances***.
4. Paste to remaining circles	Repeat for the other two circles.

ACTIVITY: Reporting The Results

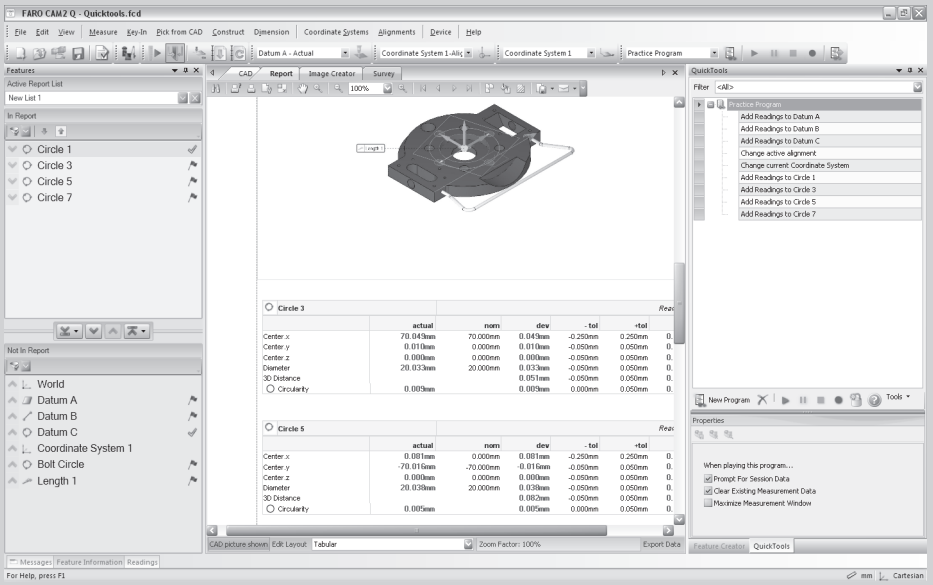
In this activity, you will make a report of the measurement results.

☛ *Before clicking the Report tab, make sure the view on the Main Window is what you want to appear on the report. Zoom in on the entire part or the area of the part you want to include in the report. A good way to zoom in on and orient the part is to use the hot keys. The, **6** hotkey orients the part to a top view, **E** hotkey zooms all, **I** zooms in, **O** zooms out and the arrow keys pan around the window.*

1. Switch to the Report view	Click the ***Report*** tab.

You can automatically switch to Report view when running the Quicktools program by adding the View Report command into the program. On the ***QuickTools*** toolbar, select ***Tools*** and select ***View Report*** while recording your QuickTools program.

2. Add each circle to the report	Add the four circles to the ***In Report*** list.



3. CAD tab	Click the CAD tab to return to the Main Window.

QuickTool Program Properties

There are three properties that may be enabled for each QuickTools program. When enabled, these properties activate special commands when the QuickTools program is started.

- ***Prompt for Session Data*** prompts the operator to update the contents of the *Document Header Information* dialog box before starting the first command.

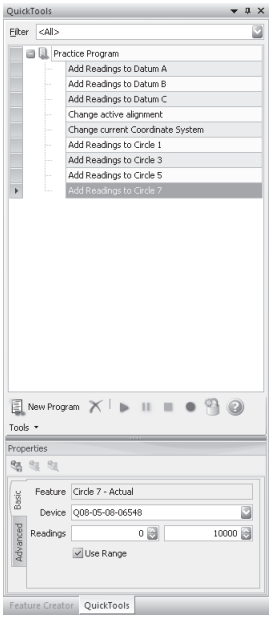
  ☛ *The Document Header Information dialog contains information such as Part Name, Session Name, and Operator, which may be used to provide unique information for facilitating repeatable part measurement.*

- ***Clear Existing Measurement Data*** removes ***all*** readings from the measurement file before starting the first command, providing for a "fresh" set of measurements each time the operator begins the QuickTools program.

- ***Maximize Measurement Window*** maximizes the ***Measurement Window*** panel to fill the entire computer screen when the program plays.

Command Properties

Each measurement command is listed as a single line item in the QuickTools panel. You can modify certain parameters of each measurement command using the Properties panel below the list of commands. Click a measurement command to access the command's parameters in the Properties panel.



CAD View Commands

Use the CAD View buttons to assign, preview, or delete a specific view of the features in the ***Main Window*** for any QuickTools command.

- ***Save CAD View for Step***: Manipulate the view of the ***Main Window***, and click this button to save this view to the QuickTools command.
- ***Clear Saved CAD View***: Click this button to delete an associated view of ***Main Window*** from the QuickTools command. If no CAD View has been saved to this command, this button is grayed out.
- ***Preview Saved CAD View***: Click this button to preview an associated view of the ***Main Window*** from the QuickTools command. If no CAD View has been saved to this command, this button is grayed out.

Command Parameters

For each QuickTools command, the ***Properties*** panel is divided into two tabs, Basic and Advanced.

Basic Tab

The Basic tab has 3 main parameters: Feature, Device, and Readings.

- ***Feature*** contains the name of the selected feature.
- ***Device*** Chooses which device will measure the feature (assuming there's more than one measurement device to be used).
- ***Readings*** Sets a low and high limit to the number of readings the user will add to each feature. Uncheck the ***Use Range*** check box to establish a high limit only.

Advanced Tab

The Advanced tab controls scanning functionality.

- ***Sample Mode*** Sets an interval parameter, a distance parameter, or both. An interval parameter takes a point over a specified time; a distance parameter takes a point over a specified distance.
- ***Replace Existing Readings*** removes any existing reading from the feature.

ACTIVITY: Modifying QuickTools Program Command Properties

In this activity, you will modify the properties of some of the measurement commands we've added to the QuickTools program.

1. Modify the Add Readings to Circle 1 command to take 7 readings	in the ***QuickTools*** panel, select the ***Add Readings to Circle 1*** command. On the ***Basic*** tab, clear the ***Use Range*** checkbox, and change the number of Readings to `7`.
2. Modify the Add Readings to Circle 5 command to take readings every .10 mm	Select the ***Add Readings to Circle 5*** command and click the ***Advanced*** tab. Select ***Distance Interval*** from the ***Sample Mode*** drop-down list, and enter `0.10` in the ***Distance*** field.
3. Modify the Add Readings to Circle 7 command to take readings every .5 mm	Select the ***Add Readings to Circle 7*** command and click the Advanced tab. Select ***Distance Interval*** from the ***Sample Mode*** drop-down list, and enter `0.5` in the ***Distance*** field.

Comments

A comment is a textual message that an operator will see when playing the QuickTools program, must acknowledge by clicking OK, and can thus be used to help guide the operator through the program by providing important information at key points.

You can add a comment to any QuickTools program command.

ACTIVITY: Adding Comments To A QuickTools Program Command

In this activity, you will add a comment, and a View Report command in the QuickTools program.

1. Start Recording	On the ***QuickTools*** panel, click ***Record***.
2. Select command	Select the ***Add Readings to Circle 1*** command
3. ***Add Comment***	On the ***QuickTools*** panel, click the ***Tools*** menu, and select ***Add Comment***.
4. Input text	In the ***Add Comment*** dialog box, enter: `Begin measuring the 20mm circles on the top face.` Click ***OK***.

When the QuickTools program is run, the operator will be presented with a text box containing your comment before continuing with the command list.

5. View Report command	On the ***QuickTools*** panel, click the ***Tools*** menu, and select ***View Report***. On the ***Report View Settings*** panel, Click ***OK***. This command always adds to the end of the QuickTools program.

The ***Tools*** menu also has the ***Change Adapter*** command for the FARO Laser Tracker. When executing this step of a QuickTools program, the operator is guided through the process of changing the adapter. When the adapter change is successful, the operator can continue running the QuickTools program.

ACTIVITY: Finishing And Playing The QuickTools Program

In this activity, you will complete the QuickTools program and play it.

1. Click ***Stop***	On the ***QuickTools*** panel, click ***Stop***. CAM2 Q stops recording commands and the QuickTools program is complete.
2. Save the file	From the ***File*** menu, Click ***Save*** to update your `Quicktools.fcd` file.
3. Play the QuickTools program	On the ***QuickTools*** panel, click ***Play*** to run the program.

Reordering Program Commands

After you've finished recording commands to the QuickTools program, you can easily change the order of the commands by dragging the command to another location in the list (clicking a command and holding the left mouse button, dragging the command to a new location within the QuickTools program, and releasing the left mouse button).

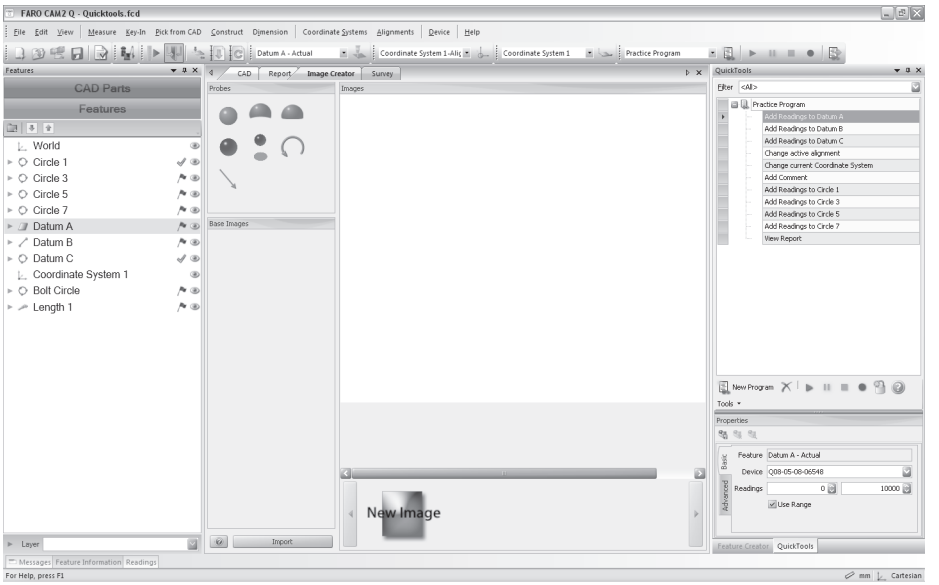
Reordering Program Commands

Deleting QuickTool commands from a program is simple, just select the command and click the ***Delete*** button (X icon) on the ***QuickTools*** panel.

Image Creator

Now that you have created your QuickTools program, you may add pictures to the measurement commands with Image Creator. Including pictures of the part with each command will show an operator exactly where to place the probe for recording measurements and where to record end-clicks.

Click the Image Creator tab to access Image Creator.



Picture Options

Picture options include:

- Photographs taken with a digital camera (*.jpg, *.png, *.bmp format)
- *.jpg, *.png, *.bmp files of your part from a CAD program
- Screen captures of your CAD model made with the CAM2 Q ***File*** < ***Export*** < ***Picture to File*** command (this may be the easiest option if you don't have access to a digital camera.)

When taking pictures of your part, include as many features in a shot as possible without sacrificing space, picture clarity, and quality. Capturing a big cluster of features from a long distance will not provide enough detailed information for the operator.

The ideal image size is 640x480 pixels; most cameras have an adjustable setting for the size of a picture. However, you should first add one image to a QuickTools program command, then play the command and look at the picture in the ***Measurement Window*** panel. One image can be used as many times as needed.

ACTIVITY: Adding Pictures With Image Creator

In this activity, you will add pictures to your QuickTools program using Image Creator.

1. Access Image Creator	Click the ***Image Creator*** tab to access Image Creator.
2. Import a base image into Image Creator	Click ***Import***. Browse to My Documents\ CAM2 Q\ v1.4 Training Files\Part Photographs\. In the Open dialog box, select all the images for the QuickTools program and click ***Open***. The images now appear in the ***Base Images*** box.
3. Assign the base image to the first measurement command	In the ***QuickTools*** panel, select the first measurement command, ***Add Readings to Datum A***. In the ***Image Creator*** panel, click ***New Image***. Drag an appropriate image for the first command from the ***Base Images*** box onto the ***Images*** screen.

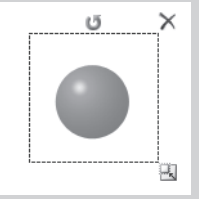
Green and Red Target Points (Probes)

After you've added pictures of the part into Image Creator, you can overlay green target points to identify the places to record readings and red target points to specify end clicks.

During the execution of a program, each reading and end click requires its own picture and target point (where the user should place the Probe/SMR).

ACTIVITY: Adding Green And Red Target Points To The Images

In this activity, you will add green and red target points (signifying Probe/SMR positions) to the pictures you've uploaded.

1. Add a green target point to the base image	Drag a green target point from the ***Probes*** box onto the image in the ***Images*** screen.
2. Rotate, resize, and position the green target point	Hover the mouse over the target point to view the target point's positioning menu. ❒ The arrow icon at the top rotates position of the target point. ❒ The X icon at the upper right deletes the target point form the Images screen. ❒ The extend icon in the lower right increases/decreases the size of the target point. 
3. Add and move green target points for each reading	Once the first target point is in location, click ***New Image***. Repeat steps 1 - 3 until all the readings needed have images and target points for each feature type. For this plane you should add seven green target points.

After you indicate the location of the readings, you will indicate the location to record an end click.

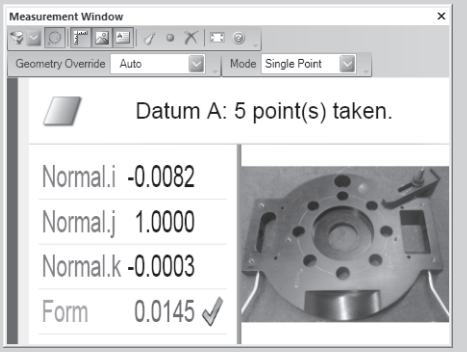
4. Add red target point	Drag a red target point from the ***Probes*** box onto the image in the ***Images*** screen.
5. Rotate, resize, and position the red target point	Rotate, resize, and position the red target point

Number of readings

for optimal results:
Plane - seven
Line - five
Circle - seven

Continue to add the appropriate images and target points for all of the remaining measurement commands.

☛ *When you run the QuickTools program, the Measurement Window panel will include the images you've created.*



Save the measurement file, which now includes all of the modifications to the QuickTools program.

6. Save the file	From the ***File*** menu, Click ***Save*** to update your `Quicktools.fcd` file. The QuickTools program, base images, and target point locations are saved with the *.fcd measurement file.

Editing a QuickTools Program

- Click any measurement command in the ***QuickTools*** panel to view and edit the properties in the ***Properties*** panel.
- You may add new commands by clicking ***Record*** in Measure or Template mode. Click ***Stop*** to cease recording new commands.
- You may also change the order of the commands by dragging the commands to another location within the program. If you have added a Comment command before a measurement command, make sure to also move the Comment.
- You may also delete commands from the program by selecting the command and clicking ***Delete*** (trashcan icon).

  ☛ *You may also delete the entire QuickTools program by selecting the program name and pressing the **Delete** key.*

CAM2Q

Module 10:
Iterative Alignments

In this module, you will become familiar with iterative (best-fit) alignments to align and compare measured features to nominal features.

Prerequisite: Basic Measurements, Nominals and Tolerances.

MODULE OBJECTIVES

You will be able to:

- Create Iterative alignments using the default settings, using weights, and using material temperature considerations
- Report the alignment results

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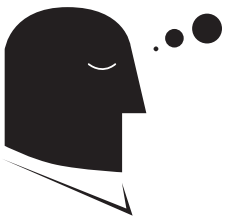
SCENARIO: ITERATIVE ALIGNMENTS

You are familiar with measuring features and viewing the results using a Coordinate System alignment, but now we will use a "best fit" alignment between the measured and the nominal features.



I need to make sure the features I measure on my part are comparable to their nominals. How do I perform an Iterative alignment to best-fit measured features to their corresponding nominals?

Iterative Alignments

An Iterative alignment uses point reducible features to best-fit them to their corresponding nominal values. You can create an Iterative alignment using three or more point-reducible features with associated nominals. You may add nominals to features that have already been measured, or you may create nominals and add measurements to the nominals.

Preferences for Iterative Alignment

Before you create your Iterative alignment, access the Measurement preference (***Edit > Preferences > Measurement > General*** tab) and ensure that the ***Use Last Plane*** check box is selected.

☛ *If the Use Last Plane check box is clear, each measured circle will have a separate projection plane.*

ACTIVITY: Creating Nominals For An Iterative Alignment

In this activity, you will create nominal circles from a CAD file for use in creating an Iterative alignment.

1. Open a new file	On the ***File*** Menu, click ***New***.
2. Import CAD	Click the ***Import CAD*** icon on the toolbar. Browse to the ***\My Documents\CAM2 Q v1.5 Training Files*** folder and select ***Demo Part Q.x_t***
3. Pick from CAD menu	On the ***Feature Creator***, select ***Pick From CAD***.
4. Circle	Select ***Circle***.
5. Pick from CAD	Click the Circle 1 edge. ***Circle 1*** appears in the ***Features*** panel.

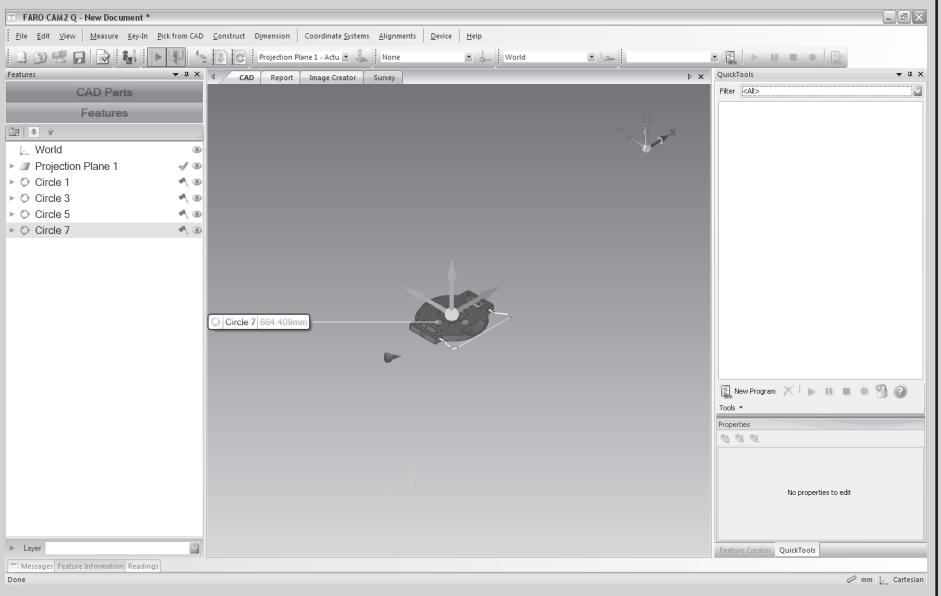
6. Repeat for other features	With the ***Pick Circle from CAD*** window remaining open, check the name and click the edge for Circle 3, Circle 5, and Circle 7.
7. Close	Click ***Close***.

Now that you've created the nominal features, you can add measurements to them.

ACTIVITY: Adding Measurements To The Nominals

In this activity, you will add measurements to the nominal circles you created in the last activity.

1. Add Measurement	Select ***Circle 1, Circle 3, Circle 5,*** and ***Circle 7***. Right-click select ***Add Measurement***. Since there is no active projection plane, CAM2 Q automatically adds a plane feature.
2. Measure Plane	Add readings to ***Projection Plane 1***.
3. Measure Circles	Add readings to ***Circle 1***, ***Circle 3***, ***Circle 5,*** and ***Circle 7***.



Zoom Out (use the ***O*** hot key) so that you can see your measurements and your CAD file. Notice that they are not aligned yet.

Measurement and Nominal Association

You have just created nominal features and added measurements to those features. By doing so, your measurements and nominals are automatically associated and a feature group is created. To verify this association, click each circle feature group in the ***Features*** panel and view the Nominal Feature column in the ***Feature Information*** panel.

You now have the measurements, nominals, and feature associations you need to create an Iterative alignment.

ACTIVITY: Creating An Iterative Alignment

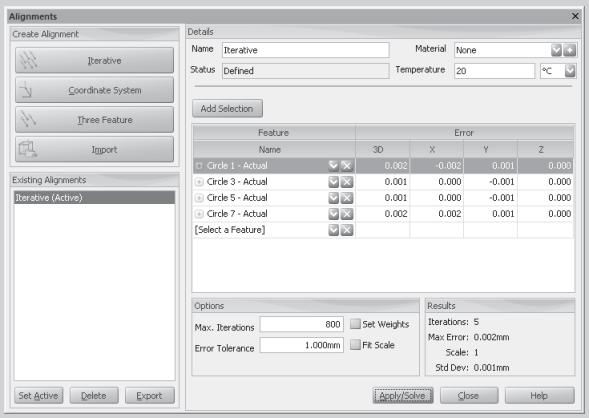
In this activity, you will create an Iterative alignment using the nominals and measurements you've created.

You can also press the *A* hot key to start the Alignments command.

1. Alignment command	From the ***Alignments*** menu, select ***Iterative Alignment***.
2. Select Circle 1 - Actual	Click the ***[Select a Feature]*** drop-down and select ***Circle 1 - Actual***.
3. Select other features	Repeat step 2 for ***Circle 3 - Actual***, ***Circle 5 - Actual***, and ***Circle 7 - Actual***.

You can also add these features to the Alignments dialog box by selecting them in the ***Features*** panel and clicking the ***Add Selection*** button.

4. Enter name	Enter `Iterative` in the Name box.
5. Apply/Solve	Click ***Apply/Solve*** to create the Iterative alignment.



Note that the "Iterative" alignment is now active. The X, Y, Z, and 3D columns show the calculated deviation of each feature in the best fit.

6. Close	Click ***Close***.

You should now be able to see that your measurements are aligned to the CAD file. Also notice that your alignment now shows active in the ***Alignments*** toolbar.

Weights

You can apply weights to the X, Y, and Z coordinates of each feature in the alignment. A weight ensures a higher priority for that feature in the calculation.

In the Alignments dialog box, select the ***Set Weights*** checkbox to add the Weights columns to the Alignments dialog box, then enter a value for the X, Y, and Z coordinate for each feature.

The default value is 1; input a number greater than 1 to emphasize a weight.

ACTIVITY: Adding Weights To An Iterative Alignment

In this activity, you will add weights to ***Iterative***.

1. Alignment command	Press the ***A*** hot key.
2. Set weights	Select the ***Set Weights*** checkbox to view the Weights columns.
3. Enter values	In the ***Weights*** column, double-click the X coordinate of Circle 1 - Actual and enter 5.
4. Apply/Solve	Click ***Apply/Solve*** and note the differences in the ***Results*** area (lower right).
5. Close	Click ***Close***.

Fit Scale

You can scale the measured features during the best fit calculations. The scale is the ratio of the measured features to their nominal features.

Select the ***Fit Scale*** check box to scale measured features during best fit calculations.

If the ***Fit Scale*** check box is not selected, no scaling is used in the calculations and the scale of the part is:1.

Material Temperature

Scaling for temperature allows you to measure parts in areas that are either warmer or cooler than the nominal temperature of 20°C (68°F). The measurement results are adjusted for thermal expansion or contraction.

You can adjust calculations for both the material and the current temperature of the part. To scale for material temperature changes:

- After creating an Alignment, in the ***Details*** section of the ***Alignment*** dialog box, click the ***Material*** drop-down arrow and select a material

from the library. You will see the coefficient of thermal expansion (CTE) next to the material name. You cannot change this value.

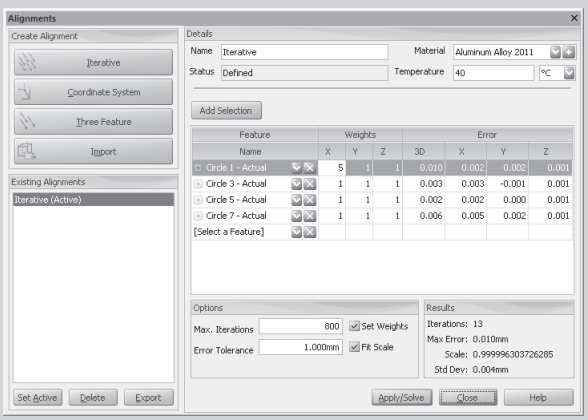
- Enter the temperature of the part, and select Celsius (°C) or Fahrenheit (°F) from the drop-down list.

To create a new material, click the ***Plus*** (+) button next to the ***Materials*** drop-down arrow. In the ***Create New Material*** dialog box, enter a name and coefficient of thermal expansion value for the new material. Click ***OK*** to add this material to the library.

ACTIVITY: Material Temperature - Iterative Alignment

In this activity, you will add material and temperature considerations to My Iterative Alignment.

1. Alignment command	From the ***Alignments*** menu, select ***Create/Edit*** to view Iterative.
2. Fit Scale	Select the ***Fit Scale*** checkbox.
3. Select Material	Click the ***Material*** drop-down (upper right) and choose ***Aluminum Alloy 2011***.
4. Enter Temperature	Enter `40` in the ***Temperature*** box.
5. Apply/Solve	Click ***Apply/Solve*** and note the differences in the ***Results*** area (lower right).
6. Close	Click ***Close***.



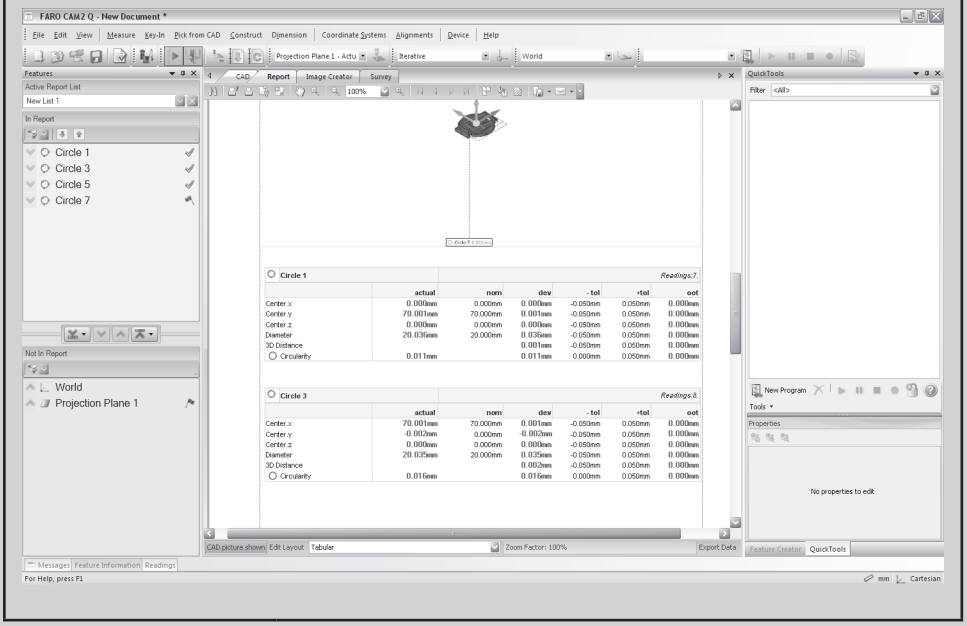
Now that you've created your Iterative alignment, you can create a report of the results.

ACTIVITY: Reporting The Results Of The Iterative Alignment

In this activity, you will view the results of the Iterative alignment using CAM2 Q's reports.

☛ *Before clicking the Report tab, make sure the view on the Main Window is what you want to appear on the report. Zoom in on the entire part or the area of the part you want to include in the report. A good way to zoom in on and orient the part is to use the hot keys. The **6** hotkey orients the part to a top view, **E** zooms all, **I** zooms in, **O** zooms out and the arrow keys pan around the window.*

1. Select the Circles	In the ***Features*** panel, select Circle 1, Circle 3, Circle 5, Circle 7.
2. Report tab	Click the ***Report*** tab.
3. Add features to the report	The four circles automatically add to the ***Active Report List***.



Review the measurement results of the four circles.

ACTIVITY: Save The Measurement File

In this activity, you will save your work in the measurement file.

1. Save As	From the ***File*** menu, select ***Save As***.
2. Your folder	Browse to your folder and type `Iterative.fcd`.
3. Save	Click ***Save***.

Auto Nominal Association

This function is not active until an alignment has solved.

Auto Nominal Association is a CAM2 Q preference which controls the automatic association of the nearest nominal to a measured feature - providing it falls within a set of tolerance zones. This preference is enabled by default. To enable or disable auto nominal association:

- On the ***Edit*** menu, click ***Preferences***. In the ***Preferences*** dialog box, select ***Measurement*** and the ***Auto Nominal*** tab.
- Select the ***Automatically Associate Nominal*** check box:

ACTIVITY: Measure Additional Features

In this activity, you will measure the remaining circle on the part. The ***Auto Nominal*** preference will automatically find and associate the nominal circles after the measured circle has solved.

1. Main Window	Click the ***CAD*** tab to return to the CAD screen.
2. Circle command	From the ***Measure*** menu, select ***Circle***.
3. Measure circles	Measure circles 2, 4, 6, 8. You can rename the features if you wish.
4. Review the report	Click the ***Report*** tab. Add these new circles to the report and review the results.

CAM2Q

Module 11:
Wireframe CAD Inspection

In this module, you will become familiar with using the wireframe data of a CAD model to facilitate inspection of your part.

Prerequisite: Working with CAD module.

MODULE OBJECTIVES

You will be able to:

- Identify proper alignment using the blueprint design
- Employ a wireframe CAD model with a Coordinate System Alignment

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SCENARIO: COMPARING MEASUREMENTS TO CAD

The core principle of wireframe CAD inspection is the comparison of physical part measurements to nominal CAD features. When you align the measurements and nominals, the differences between them can be identified and analyzed.

I have to inspect the part and compare the measurements to the "perfect world" features in my CAD file. Should I create a coordinate system based on my measurements and align with the CAD's coordinate system? What's the best way to compare my measurements to my CAD nominals?

The coordinate system on the physical part relates to the coordinate system of the part's CAD model. in CAM2 Q, you can align these coordinate systems and associate the measured features to the nominal features contained in the CAD model.

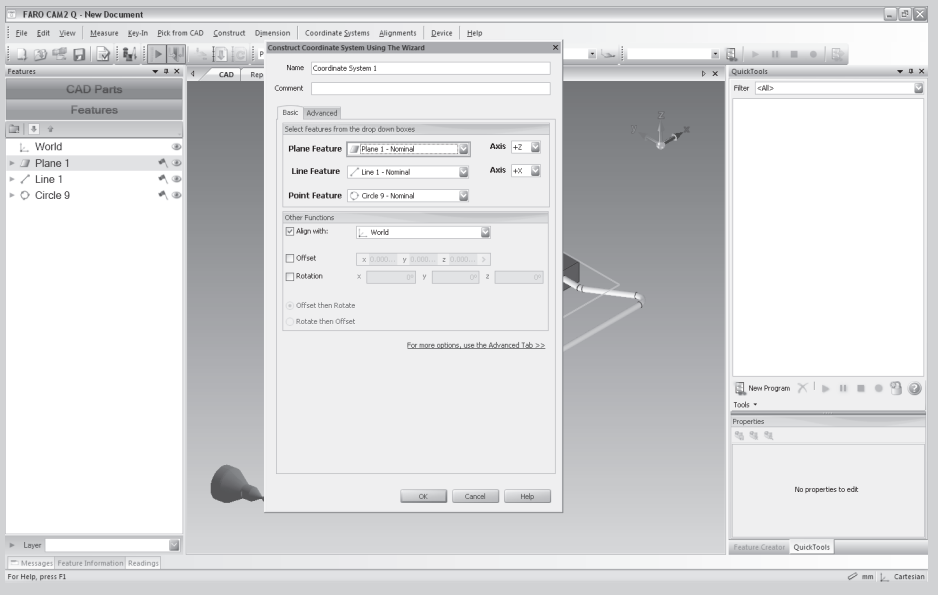
Wireframe CAD Inspection Workflow

1. Import the CAD file.
2. Measure the part and create a coordinate system.
3. Align the measurement coordinate system to the wireframe CAD model.
4. Measure other features for comparison to the CAD model.
5. Associate measured features to nominal features, or add measurement commands to nominal features.
6. Adjust positional and form tolerances for the holes, round slots, etc.
7. Create and view a report.

ACTIVITY: Measured Coordinate System And Aligning To CAD

In this activity, you will measure features, create a coordinate system with those features, and align the coordinate system with the CAD coordinate system for proper alignment.

1. Open a new file	Hold down the ***Ctrl*** key on the keyboard and press the ***N*** key.
2. Import CAD file	Import the Demo Part Q.x_t file. Then, press the ***S*** hot key to view the different shaded and wireframe views.

3. Measure Mode	If necessary, click the ***Measure/Template Mode*** icon so that you are in ***Measure*** mode.
4. Measure top plane	Measure the plane on the top of the demo plate. On the ***Measure*** menu, click ***Plane***.
5. Measure front line	Measure the front edge of the part from left to right. On the ***Measure*** menu, click ***2D Line***.
6. Measure Circle 9	Measure Circle 9. On the ***Measure*** menu, click ***Circle***. (You will need to rename the circle as `Circle 9`).
7. Coordinate Systems	Press the ***C*** hot key.
8. Select Plane 1	In the ***Plane Feature*** list, select ***Plane 1***. In the ***Axis*** list, select ***+Z***.
9. Select 2D Line 1	In the ***Line Feature*** list, select ***Line 1***. In the ***Axis*** list, select ***+X***. ☛ *You have just selected the Z and X axes of the new coordinate system.*
10. Select Circle 9	In the ***Point Feature*** list, select ***Circle 9***. ☛ *You have just established the center of Circle 9 as the origin.*
11. Align to World	Check the ***Align with*** box and select World ☛ *This aligns your measured coordinate system to the nominal (or World) coordinate system.*
	
12. Create and Close	Click ***OK.***

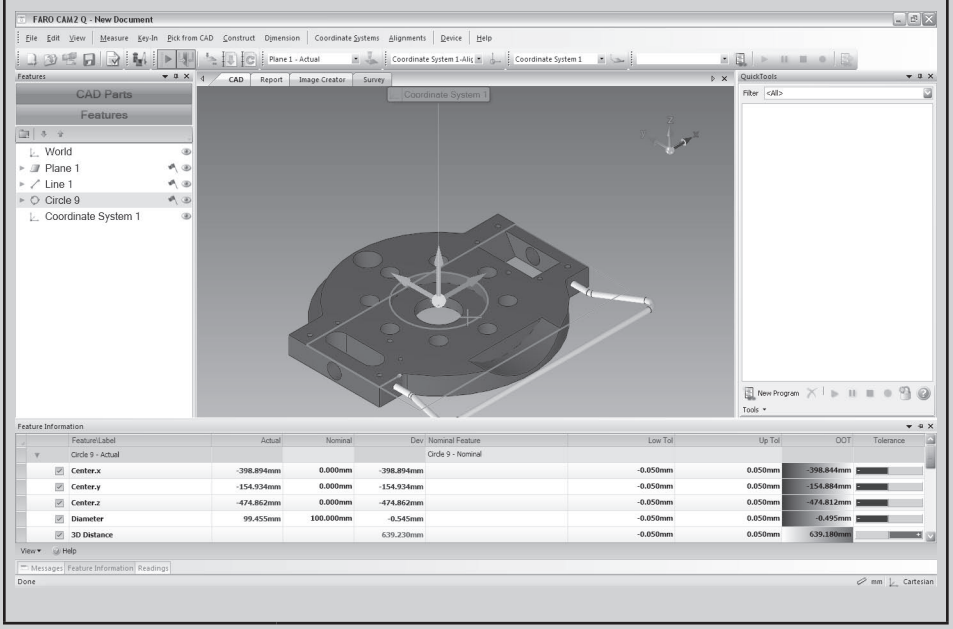
If you create nominal features and add measurements to them, your measurements and nominals are auto[illegible] associated.

Now that you've aligned your measured coordinate system with the nominal (World) coordinate system, you can compare your measured features to the nominal features of the CAD model.

ACTIVITY: Nominal Features And Comparing To Measurements

In this activity, you will check the automatic nominal of Circle 9 and review the measurement results.

1. ***Feature Information*** tab	Click the ***Feature Information*** tab. Check to see if a nominal was automatically created and associated.
2. View results	Use the ***Feature Information*** panel to compare the measured circle's measurements, positioning, and other GD&T statistics with its nominal.



Circle 9 results

Center.x Actual: _____

Center.y Actual: _____

Center.z Actual: _____

Diameter Actual: _____

3D Distance Actual: _____

Circularity Actual: _____

SCENARIO: SUB-ASSEMBLIES WITH LOCAL COORDINATE SYSTEMS, ASSEMBLIES WITH GLOBAL COORDINATE SYSTEMS

Parts that are sub-assemblies (or "piece parts") may have a part (local) coordinate system that relates to an assembly (global) coordinate system on the final assembly. For example, one automobile part may report its features based on a coordinate system established from the fully assembled automobile.

The part I am inspecting is a sub-component of a larger assembly. The CAD file of the sub-component has a coordinate system, and the larger assembly has a global coordinate system. How do I inspect using multiple coordinate systems?

If you know the relationship of the part coordinate system to the assembly coordinate system, you can apply an offset and rotation to align the part coordinate system with the assembly coordinate system. If you need smaller, more manageable dimensions, you can inspect the entire part from the part coordinate system. Then, apply an offset and rotation to the part coordinate system to move into the larger assembly coordinate system.

ACTIVITY: Offsetting A Coordinate System

In this activity, you will apply an offset to translate to the coordinate system you've created to a known position based on the final assembly.

For this example, let's assume the origin of the assembly coordinate system is located at X 500, Y 220, Z 0.

1. Edit Coordinate System 1	Select Coordinate System 1. Right-click and select ***Properties***.
2. Offset	Select the ***Offset*** check box and enter the following translation for X, Y, and Z: (500, 220, 0).
3. Modify	Click ***Modify*** and ***Close***.
4. View results	Click ***Circle 9*** and check the ***Feature Information*** panel to compare the differences in the X position, Y position, and the 3D Distance to the previous results.

CAM2Q

SCENARIO: THE CAD FILE'S COORDINATE SYSTEM IS IN THE INCORRECT LOCATION

When inspecting with wireframe CAD files, you may occasionally find that a CAD file's coordinate system is in the incorrect (or not optimal) location.

The CAD file that I'm to use for inspection has the coordinate system in the wrong place! What if I can't contact the supplier of the CAD file and convince them to send me an updated CAD file? How can I create the correct coordinate system using CAM2 Q?

The best option is to ask the CAD file's drafter to move the file's coordinate system to the appropriate position and orientation for your inspection. If this is not feasible, you may import the model into a CAD program and perform changes; however, problems may arise with the CAD model type when importing the file into a different CAD program.

CAM2 Q gives you the ability to move the CAD file's coordinate system using nominal features defined from the CAD model. In other words, you can make a new *nominal* coordinate system from the features of the CAD model, which you can then use to align your measurement device.

ACTIVITY: Creating A Nominal Coordinate System

In this activity, you will use the Pick from CAD commands to create a new coordinate system for your part.

1. New file	Open a new file.
2. Measure Mode	If necessary, click the ***Measure/Template Mode*** icon so that you are in ***Measure*** mode.
3. Import CAD file	Click the ***Import CAD*** icon.
4. Select file	In the ***Add CAD Model*** window, browse to ***CAM2 Q\ v1.5 Training Files*** folder, select ***Demo Part Q.x_t*** and click ***Open***.

5. Pick from CAD	On the ***Feature Creator***, select ***Pick From CAD***.
6. Plane	Select ***Plane***.
7. Pick points on plane	The ***Pick Plane from CAD*** window appears. Click on the part's top surface.
8. Create	Plane 1 appears in the ***Features*** panel.
9. Close	Click ***Close***.

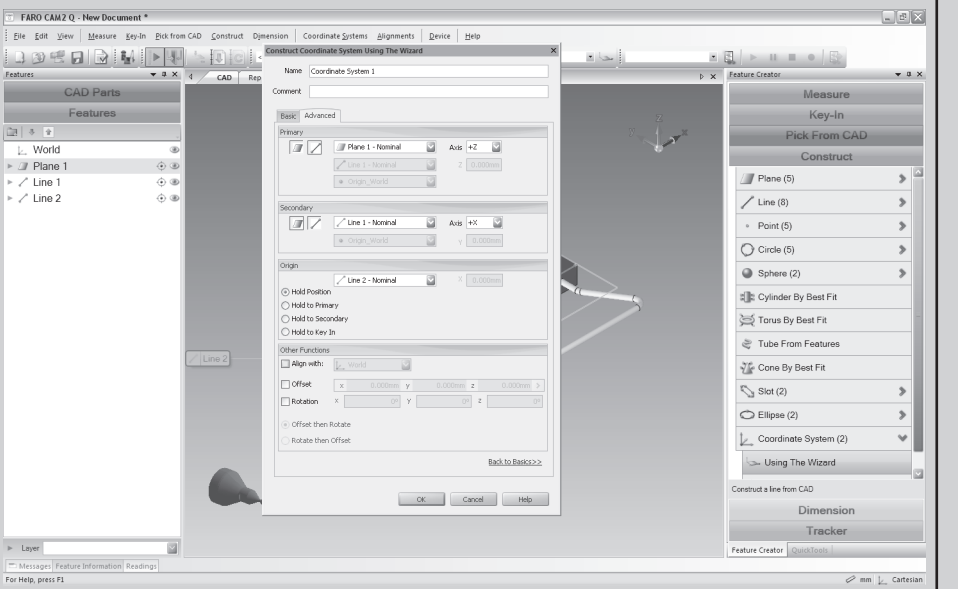
10. Pick from CAD	On the ***Feature Creator***, select ***Pick From CAD***.
11. Line	Select ***Line***.
12. Pick points on line	On the ***Pick Line from CAD*** window, click on the top front edge of the part.
13. Create	Line 1 appears in the ***Features*** panel.
14. Close	Click ***Close***.

15. Pick from CAD	On the ***Feature Creator***, select ***Pick From CAD***.
16. Line	Select ***Line***.
17. Pick points on circle	On the ***Pick Circle from CAD*** window and click on top edge of the part above Circle 11.
18. Create	Line 2 appears in the ***Features*** panel.
19. Close	Click ***Close***.

Note that with some CAD files you may have to click the minimum number of points on the CAD to create the feature.

20. Create coordinate system	On the ***Coordinate Systems*** menu, click ***Create Using the Wizard***.
21. Advanced tab	Click on the ***Advanced*** tab.

22. Select primary feature	In the ***Primary Feature*** list, select ***Plane 1 - Nominal***. In the ***Axis*** list, select ***+Z***.
23. Select secondary feature	In the ***Secondary Feature*** list, select ***Line 1 - Nominal***. In the ***Axis*** list, select ***+X***.
24. Select origin	In the ***Origin Feature*** list, select ***Line 2 - Nominal***.
25. Do not align	Clear the ***Align with*** check box.
26. Create and close	Click ***OK.***



You have just created a new nominal coordinate system for the CAD model using CAM2 Q's ***Construct from CAD*** and ***Coordinate Systems*** commands. In the next activity, you will measure the physical part, construct a part coordinate system, and align that coordinate system to the new nominal coordinate system.

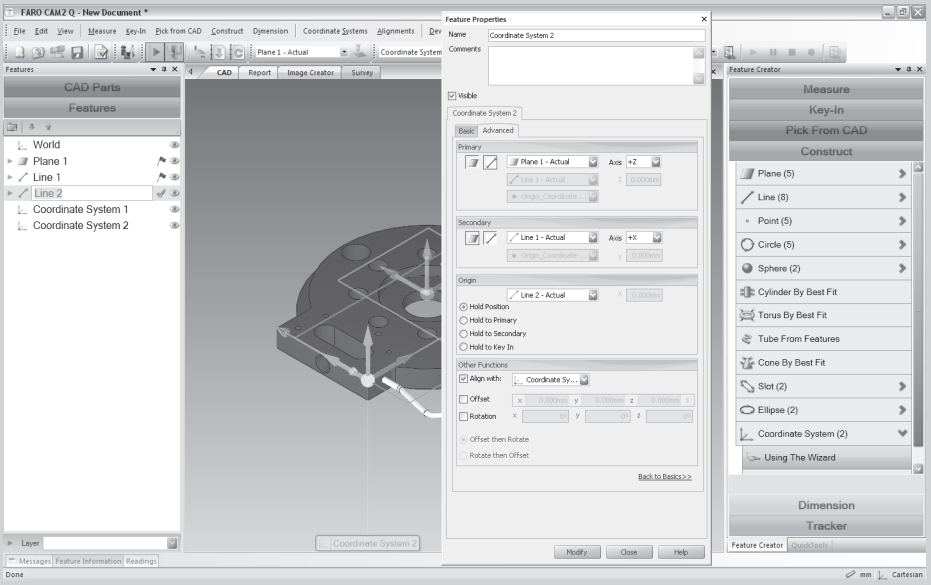
ACTIVITY: Aligning To The Nominal Coordinate System

In this activity, you will measure the part, create a coordinate system from the part measurements, and align the part's coordinate system to the nominal coordinate system (Coordinate System 1).

☛ *Make sure you are in **Measure** mode.*

1. Measure top plane	On the ***Measure*** menu, click ***Plane***. Measure the plane on the top of the demo plate. Cancel the Plane command.
2. Measure front line	On the ***Measure*** menu, click ***2D Line***. Measure the front line of the part from left to right.
3. Measure side line	Measure the side line of the part from front to back. Cancel the Line command.

4. Create coordinate system	Press the ***C*** hot key.
5. Advanced tab	Click on the ***Advanced*** tab.
6. Select primary feature	In the ***Primary Feature*** list, select ***Plane 2 - Actual***. In the ***Axis*** list, select ***+Z***.
7. Select secondary feature	In the ***Secondary Feature*** list, select ***Line 3 - Actual***. In the ***Axis*** list, select ***+X***.
8. Select origin	In the ***Origin Feature*** list, select ***Line 4 - Actual***
9. Align with Coordinate System 1	Select the ***Align with*** check box and select ***Coordinate System 1.***
10. OK	Click ***OK.***



The CAD coordinate system has moved to the correct orientation and the part is aligned. Notice that the measured features are now in a feature group with the nominal features. The feature group always uses the name of the nominals by default.

☛ *A quicker method to completing this measurement activity is to select the nominal features in the **Features** panel, right-click, and use the **Add Measurement** command. This also creates measurement features (-Actual) with the same name as the nominal features.*

ACTIVITY: Reporting The Results Of The CAD Inspection

In this activity, you will view the results of the wireframe CAD inspection using CAM2 Q's reports.

☛ *Before clicking the Report tab, make sure the view on the Main Window is what you want to appear on the report. Zoom in on the entire part or the area of the part you want to include in the report. A good way to zoom in on and orient the part is to use the hot keys. The **6** key orients the part to a top view, **E** key zooms all, **I** zooms in, **O** zooms out and the arrow keys pan around the window.*

1. Report tab	Click the ***Report*** tab.
2. Add features	On the ***Features*** panel, on the ***Active Report List***, click the ***Add all features to the list***. You can also select the arrow to the left of the feature to move it. ☛ *A quicker method is to select the features in the **Features** panel and then click the **Report** tab.*
3. View results	Compare the actual data to the nominal data.

CAM2Q

Module 12:
Surface Measurement for Inspection or Reverse Engineering

In this module, you will become familiar with measuring surface features for both surface inspection and reverse engineering. You will use surface inspection commands in CAM2 Q to inspect a part with free-form surfaces, edit complex measurement data, and create a report of the surface measurements. You will use the surface information in a CAD file as nominal data for the inspection of your part, and also create a CAD model of your part for export.

Prerequisite: Basic Measurement, Coordinate Systems, and Working with CAD modules.

MODULE OBJECTIVES

You will be able to:

- Inspect surfaces at any location (Inspect Surface Point, Inspect Surface)
- Inspect surfaces at a specific location (Home In Point, Vector Point)
- Inspect surfaces at any location with the data filtered to display at specific cross-sectional planes (Polyline, Polyline Group)
- Edit the surface data
- Report the surface data
- Measure to reverse engineer a part
- Output the data to an appropriate file format

LESSONS

CAM2Q

Measurement Mode Options

Measurement Mode options let you specify whether to add single or multiple readings. This is beneficial whether you are measuring a large feature or a free form surface. The addition of multiple readings over a specified time or distance is often called "scanning".

In the ***Measurement Window*** panel, use the Mode drop-down list to switch the reading collection mode.

- ***Single Point***: press the ***Front*** button, or the ***G*** key, to collect a single reading.
- ***Time Interval***: press the ***Front*** button, or the ***G*** key, to start the time interval. Collect a single reading over time.
- ***Distance Interval***: press the ***Front*** button, or the ***G*** key, to start the distance interval. Collect a single reading when the probe moves a distance.

To configure any Measurement Mode, click the ***Measurement Mode Setting***, or ***Settings,*** button in the ***FARO Device Control*** panel.

See the CAM2 Q chapter in the user manual of your current measurement device for more information.

Lesson 1: Surface Inspection with CAD

Up to this point, you have compared and reported the condition of simple features at their center point. With surface measurement, you can compare and report the condition of the surface at multiple locations. These measured to nominal results are more complex than simple feature results, and determine the exact location of out of tolerance conditions on the surface.

Lesson Objectives

Using the specialized surface measurement commands, you will be able to:

- Inspect surfaces at any location (Inspect Surface Point, Inspect Surface)
- Inspect surfaces at a specific location (Home In Point, Vector Point)
- Inspect surfaces at any location with the data filtered to display at specific cross-sectional planes (Polyline, Polyline Group)
- Edit the surface data
- Report the surface data

PRE-ACTIVITY:

Option 1 If training files are available	Option 2 If starting with a new file
Open Advanced 2.fcd	Open a new file
Select all of the features, then select ***Add Readings.*** Measure the features as prompted.	Add ***Demo Part Q.x_t*** to the measurement file. Add and measure: ● Plane 1 ● Line 1 (across the front of the part on the handle side) ● Circle 9

ACTIVITY: Align To A Surface CAD Model Using A Coordinate System Alignment

In this activity, we will align using a coordinate system alignment in a measurement file containing a CAD model.

1. Create coordinate system from measured features	On the ***Coordinate System*** menu, select ***Coordinate System Wizard***. Enter the name `Measured CS` for the measured coordinate system. Select ***Plane 1*** as the Plane Feature. Select ***Line 1*** as the Line Feature. Select ***Circle 9*** as the Point Feature. Click ***OK***.

ACTIVITY: Adjust Tolerances For Surface Points

In this activity, we will adjust tolerances for surface points using the Preferences command.

1. Adjust tolerances for surface points	On the ***Edit*** menu, click ***Preferences***, click ***Tolerance***, ***Geometries*** tab, then select ***Surface Point***. Expand the ***Surface Point*** group.
2. 3D Distance	In the ***3D Distance*** parameters, change the ***Lower*** tolerance to `-0.25mm` and the ***Upper*** tolerance to `0.25mm`.

Auto Nominal Association

Auto Nominal Association is a CAM2 Q preference which controls the automatic association of the nearest nominal to a measured feature - providing it falls within a set of tolerance zones. This preference is enabled by default. To enable or disable auto nominal association:

- On the ***Edit*** menu, click ***Preferences***. In the ***Preferences*** dialog box, select ***Measurement***, then select ***Auto Nominal*** tab.
- Select the Automatically Associate Nominal check box.

Auto Nominal Association is not active until an alignment has solved.

Inspect Surface Commands

There are two commands to measure a surface at random locations.

- ***Inspect Surface Point***: measure a single point on a surface at any location.
- ***Inspect Surface***: measure a single point or multiple points (point cloud) on a surface at any location.

Each reading for these features automatically creates a nominal point on the closest surface of your CAD model. The end click determines the compensation vector of the digitized point(s).

A disk represents each reading. The disk color is predetermined by the tolerance condition. The size of the disk is a ***Display*** preference.

ACTIVITY: Inspect a Surface Point

In this activity, we will measure and a inspect surface point.

1. Inspect Surface	On the ***Measure*** menu, select ***Inspect Surface Point***.
2. Measure the surface	Take one point on the top of the part. Cancel the command.
3. Review feature	On the ***Features*** panel, expand the ***Point 1*** feature. Note that this feature has a nominal feature that was automatically created and associated.

ACTIVITY: Inspect A Surface

In this activity, we will measure and inspect a surface.

1. Inspect Surface	On the ***Measure*** menu, select ***Inspect Surface***.
2. Measure the surface	Take multiple points on the top of the part in different locations. Cancel the command.
3. View the surface points	Open the ***Readings*** panel and look at the individual surface measurements.

ACTIVITY: Customize Labels

In this activity, we will review and customize the label options.

1. Turn labels on	Select the all the features, except World, in the ***Features*** panel. Right-click and select ***Label*** until all of the labels are visible on the ***Main*** Window.

2.	Customize labels	On the ***Edit*** menu, click ***Preferences.*** Click ***Display*** and the ***CAD View*** tab.
3.	Customize labels	Change some of the label settings and then click ***OK*** to view the changes.

Home In Points

Home In Points differ from Inspect Surface Points, in that you can measure a surface at a specific nominal location. This command will guide you, with graphics and sound, to the location of the nominal point, and will not accept a reading until the Probe/SMR is in the home in zone. The Probe/SMR must be inside the home in zone to record a reading - pressing the ***Front*** button (or the ***G*** key) outside the zone has no effect. Before starting this command, you must create a nominal point or nominal vector point to use as the Home In target.

ACTIVITY: Home In Points and Nominal Vector Points

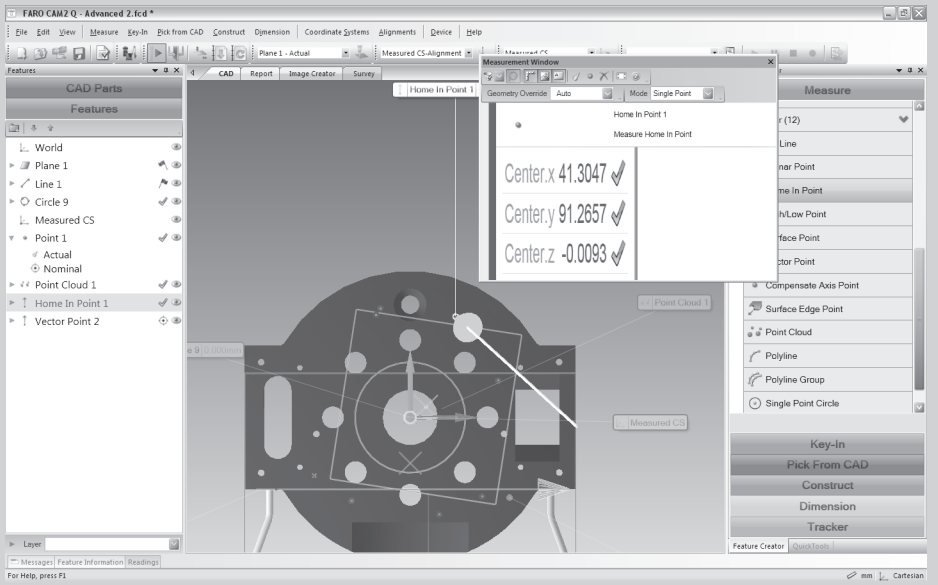
In this activity, we will define nominal locations for surface measurement.

1.	Vector Point	From the ***Pick From CAD*** menu, select ***Vector Point***.
2.	Select points	Select two points on top of part. Click ***Close***.

Now that you have the two nominal vector points, you can use each (as target points) in the ***Home In Point*** command, and measure the surface.

3.	Measure Home In Point	On the ***Measure*** menu, select ***Other***, then ***Home In Point***.
4.	Home In Point settings	For ***Approach Type***, select ***Key In***. For ***Target Point***, select ***Vector Point 1***. Change the ***Zone Diameter*** to `5.00mm`. Change the ***Approach Vector*** to ***I*** `0.00`***, J*** `0.00`***, K*** `1.00`.
5.	Create and Close.	Click ***Create***, then click ***Close***.

6. Inspect vector point	You now see the nominal point, the Home In Zone, and the current position of your probe. Move the probe to find the nominal point on the top surface of the part. Watch the graphic in the ***Main*** window and listen to the sound to help determine if you are in the Home In Zone. When you are in the Home In Zone, click the ***Front*** button (or the ***G*** key) to take a point, then pull away from the surface and click the ***Back*** button (or the ***H*** key).



7. Measure Home In Point	On the ***Measure*** menu, select ***Other***, then ***Home In Point***.
8. Home In Point settings	For ***Approach Type***, select ***Select Surface***. For ***Target Point***, select ***Vector Point 2***. For ***Surface***, select ***Demo Part Q.x_t***. Change the ***Zone Diameter*** to `5.00mm`.
9. Create and Close.	Click ***Create***, then click ***Close***.
10. Inspect vector point	You now see the nominal point, the Home In Zone, and the current position of your probe. Move the probe to find the point as with Home In Point 1.

ACTIVITY: Reporting The Surface Measurements

In this activity, we will create a report with the two measured surface (Home In) points we inspected in the previous activity.

1. Select Report Features	In the ***Features*** panel, select all the surface measurement features (created after the alignment).
2. Access the Report view	Click the ***Report*** tab.

Review the measurement results. Notice the 3D deviation between the measured point and the surface.

ACTIVITY: Save The Measurement File

In this activity, you will save your work in the measurement file.

1. ***Save As***	From the ***File*** menu, select ***Save As***.
2. Your folder	Browse to your folder and type `Surface Inspect.fcd`.
3. ***Save***	Click ***Save***.

Polylines

Polylines differ from an Inspect Surface point cloud, in that they display the measured points in the order they were taken, along a "line". This "line" is actually a series of straight-line connected surface points that approximate a curved path. CAM2 Q treats a polyline as one feature regardless of the number of points, the shape, or direction of the polyline. Each point in the polyline is compared to the nominal surface.

Polylines need a minimum of two points in order to solve.

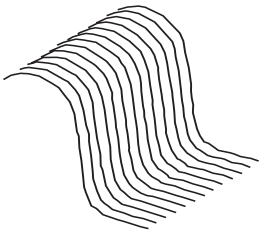
Polyline Groups

Polyline Groups are multiple polylines, or cross-sectional, measurements. CAM2 Q treats a polyline group as one feature regardless of the shape or direction of the polylines.

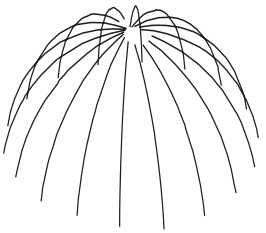
With Polyline Groups, a cross-sectional plane and distance is required to define each polyline in the group. A reading automatically records each time the Probe/SMR crosses one of the cross-sectional "lock" planes, and all of the readings on each plane are then used to create each of the polylines in the group.

Each polyline in the group needs a minimum of two points in order to solve.

Polyline Group Examples



Parallel Lock Planes



Radial Lock Planes

ACTIVITY: Configure a Measurement Mode

In this activity, we will select the Measure Mode for the next few activities.

1. Device Control panel	Press the ***P*** hot key.
2. Settings	Click the ***Measurement Mode Setting***, or ***Settings,*** button.
3. Distance Based Mode	Select the ***Distance Based Mode*** and change the distance to `0.50mm`. Click ***OK***.

☛ *These are the available measurement modes:*

- **Single Point Mode**: The measurement device takes a single point.
- **Distance Interval Mode**: The measurement device takes points at a specified distance interval (specify the distance in the device settings).
- **Time Interval Mode**: The measurement device takes points at a specified time interval (specify the time in milliseconds).

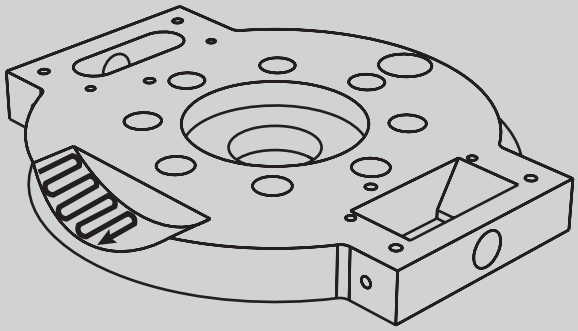
4. Close	Click the ***Close (X)*** button in the upper right corner of the Device Control panel.

ACTIVITY: Parallel Lock Plane Surface Measurement

In this activity, we will set up a parallel lock plane measurement, then "scan" a cross-section of the model.

1. Measure mode	Make sure that you are in ***Measure*** mode.
2. Measure a Plane	Measure the plane perpendicular to arc G as Plane 2. ☛ *Try switching to* **Distance Interval Mode** *and "scanning" this plane feature by adding readings every time the Probe/SMR moves 0.5mm .*
3. Measure polyline group	On the ***Measure*** menu, select ***Other***, then ***Polyline Group***.

We must configure the polyline group properties before we can see any data.

4. Polyline Group 1 properties	Select the ***Project to Surface*** check box, and select ***CAD Assembly*** from the drop-down list. Select the ***Automatically Create Nominal*** check box. Select the ***Polyline Lock*** check box and select ***Parallel Planes*** from the drop-down list. ***Base Plane***: Plane 2 - Actual. ***Base Distance***: 0.00mm. ***Distance Between Planes***: 5.00mm. ***Number of Planes***: 20. Click ***Create*** and ***Close***
5. Measure the surface	Make sure that the Measurement Mode is set to **Distance Interval**. Touch the Probe/SMR to curved surface "G", press the ***Front*** button, or ***G*** key, and "scan" the curved surface in a front to back direction perpendicular to Plane 2 - Actual.  Press the ***Back*** button, or ***H*** key, to complete the scan.

Feature Whiskers

Whiskers are single lines that visually represent distance data of a feature. There are two types of whiskers:

- Whiskers - The Whisker shows the deviation of each reading to the feature.
- Nominal Whiskers - The Whisker shows the deviation of the measured feature to its nominal.

Select a feature from the ***Features*** panel, or the ***Main*** Window, and on the ***View*** menu select ***Feature Whiskers*** to access either Whisker command. You can also access these commands by right-clicking a feature. The whiskers change color according to tolerance condition:

- Green - feature is in tolerance
- Red - measurement is above the upper tolerance
- Blue - measurement is below the lower tolerance

ACTIVITY: View And Adjust Polyline Whiskers

In this activity, we will view and adjust the polyline you created in the prior activity.

1. Select Polyline Group 1	In the ***Features*** panel, select ***Polyline Group 1***.
2. View feature whiskers	On the ***View*** menu, select ***Feature Whiskers***, then select ***Whiskers***.
3. Increase the size of feature whiskers	On the ***View*** menu, select ***Feature Whiskers***, then select ***Increase***. ☛ *You can also press the* **+** *(Plus Sign) key.*
4. Rotate CAD	Click on the screen and drag the mouse to rotate the ***Main*** window view to see the Whiskers. Press the ***8*** hot key to reset the isometric view.
5. Decrease the size of feature whiskers	On the ***View*** menu, select ***Feature Whiskers***, then select ***Decrease***. ☛ *You can also press the* **-** *(Minus Sign) key.*

ACTIVITY: Reordering Feature Data

In this activity, we will put deviations in order, from lowest to highest, remove readings on the outliers, and re-calculate the feature using the remaining readings.

1. Select Point Cloud 1	On the ***Features*** panel, select ***Point Cloud 1***.
2. Readings tab	Click on the ***Readings*** tab.
3. Click on column heading	Click on the ***DevFrom Fit*** column heading to reorder the readings.
4. Uncheck readings	In the ***On*** column, clear the check box for the outlier (larger deviation) readings to remove them from the calculation.
5. Recalculate	Click ***Apply*** to recalculate the remaining readings.
6. Repeat with Polyline Group 1	On the ***Features*** panel, select ***Polyline Group 1*** and repeat this activity.

ACTIVITY: Creating And Saving A Report

In this activity, we will create a measurement report and save it to a .csv file.

1. Access Report view	Click the ***Report*** tab.
2. ***Add features*** to the report	On the ***Features*** panel, on the ***Active Report List***, double-click on the features to add to the report in the ***Not In Report*** section. You can also select the check box to the left of the feature to move it.
3. Save Report	Click the ***Export Document*** icon and select ***CSV Document***. Click ***OK*** to accept all the default export options, and save the report with the name `My Report 2` in your folder.

ACTIVITY: Save The Measurement File

In this activity, you will save your work in the measurement file.

1. ***Save***	From the ***File*** menu, select ***Save***.
2. Your folder	This updates your `Surface Inspect.fcd` file.

Lesson 2: Scanning for Reverse Engineering

Reverse Engineering, once stigmatized as a method to "copy" or "steal" a product, now has many positive applications in manufacturing. The primary functions of Reverse Engineering are to create data to refurbish or manufacture a part for which there is no CAD data, or for part inspection (comparing a fabricated part to its CAD model or to a standard item). Outside of the manufacturing world, Reverse Engineering is widely used to document and measure historical artifacts, create models for animation in games and movies, reproduce crime scenes, and generate data for use in a wide array of medical applications.

Lesson Objectives

Using the specialized surface measurement commands you will be able to:

- Measure to reverse engineer a part
- Output the data to an appropriate file format

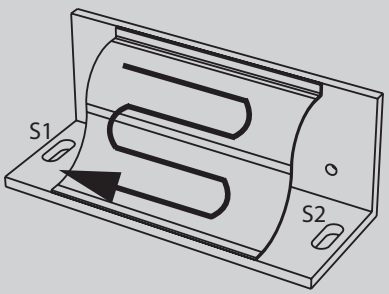
ACTIVITY: Create A New Coordinate System

In this activity, we will measure features to put a coordinate system on the demo part.

1.	Attach the bracket to the top of the training part.	
2.	New file	Open a new file.
3.	Measure plane	On the ***Measure*** menu, select ***Plane***. Measure the top of the base part as a plane. Cancel the command.
4.	Measure front edge	On the ***Measure*** menu, select 2D ***Line***. Measure the front edge of the bracket as a line.
5.	Measure left edge	Measure the left edge of the bracket as a line. Cancel the command.
6.	Create coordinate system	On the ***Coordinate Systems*** menu, select ***Create Using the Wizard***.
7.	Advanced tab	Click on the ***Advanced*** tab.
8.	Plane, Line, Line	Create a coordinate system using Plane 1, Line 1, and Line 2. Click ***OK***.

ACTIVITY: Measure Features and Export Data

In this activity, we will measure multiple polylines to define the free form surface of the bracket.

1. Measure polyline group	On the ***Measure*** menu, select ***Other***, then select ***Polyline Group***.
2. Configure the properties of Polyline Group 1	Polyline Lock: Parallel Planes Base Plane: YZPlane_World Base Distance: 30.00mm. Distance Between Planes: 5.00mm. Number of Planes: 30. Click ***Create***, then click ***Close***.
3. Measure the surface	Place the Probe/SMR on the surface and press the ***Front*** button, or ***G*** key, to scan the surface. Cover the part from top to bottom, moving left to right.  Press the ***Back*** button, or ***H*** key, to complete the scan.
4. Increase point size	On the ***Edit*** menu, click ***Preferences.*** In the ***Preferences*** dialog box, click ***Display*** and ***CAD View*** tab. Change the ***Size and Scale: Point size*** to Large. Click ***OK***.
5. Check Polyline Group	Zoom in and look at the points in the Polyline Group, and check for large gaps between the points.
6. Add additional readings	On the ***Feature*** panel, select ***Polyline Group 1***, right-click and select ***Add Readings***. Place the Probe/SMR on the surface. Press the ***Front*** button, or ***G*** key, to scan the surface again to fill the large gaps in the polylines. Press the ***Back*** button, or ***H*** key, to complete. You can repeat this command as often as you need.

You need to measure more features to completely reverse engineer this part; however, not all of them are free-form surfaces that you measure with the Polyline Group command. You should be able to see planes, circles, round slots, etc. that you measure using simple feature commands.

7. Measure back face	On the ***Measure*** menu, select ***Plane***. Measure the back face of the bracket part.
8. Measure the hole	On the ***Measure*** menu, select ***Circle***. Change the ***Measure Mode*** to ***Single Point***. Measure one, or all, of the circles on the back face of the bracket part.

If you prefer, continue measuring the remaining features on the bracket part.

9. ***Save As***	From the ***File*** menu, select ***Save As***.
10. Your folder	Browse to your folder and type `RE_Bracket.fcd`.
11. ***Save***	Click ***Save***.
12. Export file	From the ***File*** menu, select ***Export***, ***Measurement Data***, ***To CAD File***.
13. Export Wizard	In the ***Export CAD*** wizard, select and move all the features you created (- Actual) to the right window. Click ***Next***.
	Export Format: IGES Export Unit: Millimeter Click ***Next***.
	Click the ***Browse*** button Browse to your folder and type `RE_Bracket.igs`. Click ***Save***.
14. Close the Wizard	Click ***Finish***.

CAM2Q

Module 13:
Data Import/Export

In this module, you will become familiar with exporting your measurement results to different file formats, and importing a data file into a measurement file.

Prerequisite: Review Activities.

MODULE OBJECTIVES

You will be able to:

- Locate the different import and export options
- Export/Import Points to a text file

CAM2Q

PRE-ACTIVITY:

Option 1 If training files are available	Option 2 If starting with a new file
Open Nominals 2.fcd	Open a new file

	Add and measure/construct features. Add/associate their nominals:
	● Plane 1
	● 2D Line 1 (across the front of the part on the handle side)
	● Circle 1
	● Circle 3
	● Circle 5
	● Circle 7
	● Cylinder 9
	● Bolt Circle
	● Intersection Point (of Plane 1 and Cylinder 9)
	● Coordinate System using:
	  ● Plane 1
	  ● Line 1
	  ● Bolt Circle
	  ● Align to World

ACTIVITY: Export Features To Saved Files

In this activity, we will export some actual (measured) points, nominal points, and all other features to three separate .txt file.

1. Access the Export Point Data Wizard	On the ***File*** menu, click ***Export***, then select ***Points***. The ***Export Point Data*** wizard appears.
2. Select features	Select all of the "- Actuals" features, and move them to the right side of the wizard window by clicking the single arrow button. Then click ***Next***.
3. Export configuration	Take note of the current Export Configuration options, column order, and separators. Click ***Next***.

The export configuration information is important to ensure you import the points properly.

4. Export file name	Click ***Browse***. Select your file folder. Name the file `Actuals.txt` and click ***Save***. Click ***Finish***.

The text file saves to the computer and automatically opens in Notepad.

5. Close Notepad	Click the ***Close*** button (X) in the upper right corner of the Notepad window.
6. Access the Export Point Data Wizard	On the ***File*** menu, click ***Export***, then select ***Points***. The ***Export Point Data*** wizard appears.
7. Select features	Select all of the "- Nominal" features, add move them to the right side of the wizard window by clicking the single arrow button. Then click ***Next***.
8. Export configuration	Take note of the current Export Configuration options, column order, and separators. Click ***Next***.

The export configuration information is important to ensure you import the points properly.

9. Export file name	Click ***Browse***. Select your file folder. Name the file `Nominals.txt` and click ***Save***. Click ***Finish***.

The text file saves to the computer and automatically opens in Notepad.

10. Close Notepad	Click the ***Close*** button (X) in the upper right corner of the Notepad window.
11. Access the Export Point Data Wizard	On the ***File*** menu, click ***Export***, then select ***Points***. The ***Export Point Data*** wizard appears.
12. Select features	Move all of the features to the right side of the wizard window by clicking the ***Select All*** button. Then click ***Next***.

13. Export configuration	Take note of the current Export Configuration options, column order, and separators. Click ***Next***.

The export configuration information is important to ensure you import the points properly.

14. Export file name	Click ***Browse***. Select your file folder. Name the file `Features.txt` and click ***Save***. Click ***Finish***.

The text file saves to the computer and automatically opens in Notepad.

15. Close Notepad	Click the ***Close*** button (X) in the upper right corner of the Notepad window.
16. Save your file with a new name	On the ***File*** menu, click ***Save As***. Enter `Data Export.fcd` as the file name.

ACTIVITY: Import Points From A Saved File

In this activity, we will import some nominal points from a .txt file.

1. New File	On the ***File*** menu, click ***New***.
2. Access the Import Point Data Wizard	On the ***File*** menu, click ***Import***, then select ***Points***. The ***Import Point Data*** wizard appears.
3. Select file	Click ***Browse*** and locate ***Features.txt***, select the file click ***Open***. Click ***Next*** to continue.
4. Import Configuration	Make sure that Import Configuration is the same as the Export Configuration and click ***Next***.
5. Select features	Select all of the "- Nominal" features and click the single arrow button to move the features to the right side of the wizard window. Select the ***Import as nominal*** option.
6. Import the Points	Click ***Finish***.

The nominal information from the text file imports directly to features and adds to your measurement file. Now you are ready to measure coordinate system features and add measurements to these nominal features.

CAM2Q

Module 14:
Iterative Alignment Using Surfaces

In this module, you will become familiar with creating an Iterative Alignment using surface point measurements and nominal surfaces.

Prerequisite: Iterative Alignments.

MODULE OBJECTIVES

You will be able to:

- Create specific Nominal points on a CAD surface for use in an alignment.
- Measure and remeasure the surfaces of the part to refine the results of an alignment.

CAM2Q

Using Surfaces Instead Of Features

If your part has fewer than three point-reducible features, you will likely have enough nominal CAD surface data in a CAD model to create an alignment. Use Surface Point measurements and CAD surface data to align the part with your measurement device.

Previously, you used point-reducible features with associated nominals to create an iterative alignment. These measured features were the same geometry type as the nominal features (circles, slots, spheres, etc.). Now, instead of a point-reducible feature, we will use a surface point measurement feature and a nominal surface feature to create a best fit iterative alignment.

ACTIVITY: Performing An Alignment Using Surfaces

In this activity, we will perform an alignment by picking nominals from CAD and associating them to nominal surfaces.

1. New file	Open a new file.
2. ***Import CAD***	From ***\CAM2 Q v1.5 Training Files,*** import Demo Part Q.x_t.
3. Define nominals	On the ***Pick from CAD*** menu, click ***Vector Point***.
4. Points on the top of the CAD	Click on the top of the CAD model surface near P1. Click three more points near P2, P3 and P4.
5. Points on the left of the CAD	Click on the left side of the CAD model surface near Circle 11. Click one more point on the left side.
6. Points on the front of the CAD	Click on the front of the CAD model surface to the right of Arc G. Click ***Close***.
7. Select the features	In the ***Features*** panel, select the vector point features.
8. Add measurements	Right-click and select ***Add Measurement***.
9. Measure the surfaces	Measure the surfaces with a single reading for each vector point as shown in the Measurement Window panel. Place the probe on the correct surface in the general area of the nominal point.
10. Select the vector point features	In the ***Features*** panel, select all vector points.
11. Align to the part	On the ***Alignments*** menu, click ***Iterative Alignment***. Click the ***Add Selection*** button to add the vector points.

12. Set Weights	Select the ***Set Weights*** check box to set the vector direction for each point. Notice that each weight value automatically corresponds to the location of each point on the surface. For example, the weight values for Vector Points 1-4 are 0,0,1 and control the Z axis of the alignment.
13. Solve	Click the ***Apply/Solve*** button.

Note the high value of the Max Error, or the error between the measured points, and the nominal surfaces. Now we will remeasure the vector points and automatically refine the alignment, with the goal of improving it by reducing the Max Error.

14. Close	Click the ***Close*** button.
15. Select the features	In the ***Features*** panel, select all vector points.
16. Remeasure the surfaces	From the ***Measure*** menu, click ***Remeasure***.
17. Inspect vector point	You now see the nominal point, the Home In Zone and the current position of your probe. Move the probe to find the nominal point on the top surface of the part. Watch the screen and listen to the sound to help determine if you are in the Home In Zone. When you are in the Home In Zone, click the ***Front*** button (or the ***G*** key) to take a point, then pull away from the surface and click the ***Back*** button (or the ***H*** key).
18. Reset the view	Press the ***E*** hotkey to Zoom All, and ***8*** to set the Isometric view.
19. Check the alignment	Press the ***A*** hot key and review the Max Error value - it should be lower than before. If it is, you have just improved your alignment by remeasuring the surfaces while being guided to the exact nominal surface location. Click ***Close***.
20. Save	On the File menu, click ***Save As***. Name the file `Iterative2.fcd`.

CAM2Q

Module 15:
Tool Building

In this module, you will become familiar with using your measurement device to properly position a part in an assembly.

Prerequisite: none.

MODULE OBJECTIVES

You will be able to:

- Use the Feature Window to position a part correctly in space relative to another CAD surface.

CAM2Q

PRE-ACTIVITY:

Option 1 If training files are available	Option 2 If starting with a new file
Open Advanced 2.fcd	Open a new file
Select all of the features, then select ***Add Readings.*** Measure the features.	• Import ***Demo Part Q.x_t***
	Add and measure: • Plane 1 • 2D Line 1 (across the front of the part on the handle side) • Circle 9

Feature Window - Digital ReadOut

In addition to the ***Measurement Window*** panel that includes a Digital ReadOut (DRO) for the current measurement, CAM2 Q also has a ***Feature*** window that shows the distance of the Probe/SMR to a specific feature. Use a ***Feature*** window to:

- See the live deviations of a specific feature based on the current location of the Probe/SMR.
- Guide a part into position using the position of a feature.

You can have multiple ***Feature*** windows open in a measurement file, or a single window which shows the distance values for the closest feature. The values in the ***Feature*** window change color based on tolerances. You can also show/hide the individual values of the DRO to view only the important values of your feature.

ACTIVITY: Positioning The Bracket

In this activity, you will accurately position the bracket relative to the part and verify the accuracy with part measurements.

1. Construct a measured coordinate system	On the ***Coordinate Systems*** menu, click ***Create Using the Wizard***. Select ***Plane 1*** as the ***Plane Feature***. Select ***Line 1*** as the ***Line Feature***. Select ***Circle 9*** as the ***Point Feature***. Check the ***Align with: World*** box. Click ***OK***

2. Position the bracket	Position the bracket on the demo part, slot side down and lightly fasten it using the set screws.
3. Key-In Point	On the ***Key-In*** menu, select ***Point***. Type in the following values: x: -50.85mm - FaroArm x: -50.85mm - FARO Laser Tracker y: -40.64mm - FaroArm y: -40.64mm - FARO Laser Tracker z: 13.26mm - FaroArm z: 8.20mm - FARO Laser Tracker (1.5" SMR) z: 16.12mm - FARO Laser Tracker (0.875" SMR)
4. Key-In Point	On the ***Key-In*** menu, select ***Point***. Type in the following values: x: 50.85mm - FaroArm x: 50.85mm - FARO Laser Tracker y: -40.64mm - FaroArm y: -40.64mm - FARO Laser Tracker z: 13.26mm - FaroArm z: 8.20mm - FARO Laser Tracker (1.5" SMR) z: 16.12mm - FARO Laser Tracker (0.875" SMR)
5. Feature Window for Point 1	On the ***Features*** panel, select ***Point 1***, then from the ***View*** menu select ***Show Feature Window***. ☛ *You can also press the **D** hot key.*
6. Tolerance tab	With ***Point 1*** still selected on the ***Features*** panel, click the ***Feature Information*** tab. Change all of the ***Lower*** and ***Upper*** tolerances to -0.500mm and 0.500mm.
7. Place Probe/ SMR	Place the Probe/SMR on the bracket's left-front set screw.

For the next step, you want the d3D and X,Z,Y values in the ***Feature*** window DRO to be green and as close to 0.00 as you can get them. You will now compare the numbers to the nominal position you specified when keying in the point.

8. Move bracket	If necessary, move the bracket until the numbers are close to 0.00 and green.
9. Feature Window for Point 2	On the ***Features*** panel, select ***Point 2***, then press the ***D*** hot key. ☛ *You will need to move this second window if it opens on top of the first window.*

10. Feature Information tab	With ***Point 2*** still selected on the ***Features*** panel, click the ***Feature Information*** tab. Change all of the ***Lower*** and ***Upper*** tolerances to `-0.500mm` and `0.500mm`.
11. Place Probe/ SMR	Place the Probe/SMR on the bracket's right-front set screw.
12. Move bracket	If necessary, move the bracket until the numbers are close to 0.00 and green.

Move back to ***Point 1***, recheck the position, and adjust if necessary. Then recheck ***Point 2***. Keep rechecking each point until you move the bracket part into the correct position.

13. Close ***Feature*** window	Close the Point 1 - Nominal ***Feature*** window.
14. Unlock from Feature	Click the ***Lock to Feature*** (slanted push pin) button to unlock the ***Feature*** window from Point 2.
15. Recheck both points	Move the Probe/SMR to each point and recheck the values. Notice that the ***Feature*** window automatically shows the values of the feature closest to the Probe/ SMR.

ACTIVITY: Save The Measurement File

In this activity, you will save your work in the measurement file.

1. ***Save As***	From the ***File*** menu, select ***Save As***.
2. Your folder	Browse to your folder and type `Tool Building.fcd`.
3. ***Save***	Click ***Save***.

CAM2Q

Module 16:
Coordinate Systems Using Offset and Rotation

In this module, you will become familiar with constructing coordinate systems using advanced techniques.

Prerequisite: Coordinate Systems.

MODULE OBJECTIVES

You will be able to:

- Establish an XYZ coordinate system on a part that reflects the values on the blueprint or CAD model.

CAM2Q

Rotations and Offsets

An inspection drawing with datum geometries usually requires a coordinate system alignment in which the part coordinate system is created using the measured datum features and aligned to the assembly (World coordinate system). The assembly coordinate system must be in the location defined by the datum features to correctly align the part to the assembly.

For example, you have a part that has its own coordinate system and you need to report the measurement results of the part in the coordinate system of the assembly. After you select the features for a coordinate system, you can enter Rotation and Offset values. This creates the part coordinate system and then automatically moves and/or rotates that coordinate system to match the assembly coordinate system.

When you select both Offset and Rotation, you can also select whether to offset or rotate first.

ACTIVITY: Create And Offset A Coordinate System

In this activity, you will create a measured coordinate system and offset it from the part's origin.

1. New file	Open a new file.
2. Import CAD	Import the ***Demo Part Q.x_t*** CAD file.
3. ***Measure*** or ***Template*** mode	Choose either Measure or Template mode. ☛ *Remember that in **Measure** mode the **Add Readings** command is automatically started after you add a feature from the **Measure** menu. In **Template** mode, you will need to select the features and then start the **Add Readings** command.*
4. Add Features	On the ***Measure*** menu, click ***Plane***, ***2D Line***, and ***Circle***.
5. Add Readings	Measure and rename the features as ***Plane 1***, ***Line 1*** and ***Circle 9***.
6. Construct a measured coordinate system	On the ***Coordinate Systems*** menu, click ***Create Using the Wizard***. On the ***Basic*** tab: ● Select ***Plane 1*** as the Plane Feature. ● Select ***Line 1*** as the Line Feature. ● Select ***Circle 9*** as the Point Feature. Clear the ***Align with:*** check box.
7. Add offset	Select the ***Offset*** check box and enter `-150` for X, `-65` for Y, and `0` for Z.
8. Create the coordinate system	Click ***OK***.

Hot Keys
F3: Plane
F4: 2D Line
F10: Circle

You can also press the Chot key to start the Coordinate System Wizard.

9. Labels	On the ***Features*** panel, select ***World*** and ***Coordinate System 1***. Right-click and select ***Label*** to show their labels in the Main Window.

Zoom Out and look at the ***Coordinate System 1*** icon to see that is *not* in the center of ***Circle 9.***

10. Alignment to the part	On the ***Alignments*** menu, click ***Coordinate System Alignment***. ● Select ***Coordinate System 1*** as the Measured Coordinate System. ● Select ***World*** as the Nominal Coordinate System. Click ***Apply/Solve***. Click ***Close***.

Notice that ***Coordinate System 1*** and World are now aligned, origin to origin.

You can also press the *A* hot key to start the Alignments command and then click the Coordinate System button.

ACTIVITY: Create And Rotate A Coordinate System

In this activity, you will create a new coordinate system and apply rotation.

1. Construct a measured coordinate system	Press the ***C*** hot key. ● Select ***Plane 1*** as the Plane Feature. ● Select ***Line 1*** as the Line Feature. ● Select ***Circle 9*** as the Point Feature. Clear the ***Align with:*** check box.
2. Add rotation	Check the ***Rotation*** check box and enter `0` for X, `0` for Y, and `90` for Z.
3. Create the coordinate system	Click ***OK***.

Zoom Out and look at the ***Coordinate System 2*** icon to see that is rotated around the center of ***Circle 9*** (upper-right corner of the part).

4. Alignment to the part	On the ***Alignments*** menu, click ***Coordinate System Alignment***. ● Select ***Coordinate System 2*** as the Measured Coordinate System. ● Select ***World*** as the Nominal Coordinate System. Click ***Apply/Solve***. Click ***Close***.

ACTIVITY: Create An Offset And Rotated Coordinate System

In this activity, you will create a new offset coordinate system and apply rotation.

1. Construct a measured coordinate system	Press the ***C*** hot key. ● Select ***Plane 1*** as the Plane Feature. ● Select ***Line 1*** as the Line Feature. ● Select ***Circle 9*** as the Point Feature. Clear the ***Align with:*** check box.
2. Add offset	Select the ***Offset*** check box and enter `-0` for X, `70` for Y, and `0` for Z.
3. Add rotation	Check the ***Rotation*** check box and enter `90` for X, `0` for Y, and `0` for Z.
4. Create the coordinate system	Select the ***Rotate then Offset*** option button. Click ***OK***.
5. Alignment to the part	On the ***Alignments*** menu, click ***Coordinate System Alignment***. ● Select ***Coordinate System 3*** as the Measured Coordinate System. ● Select ***World*** as the Nominal Coordinate System. Click ***Apply/Solve***. Click ***Close***.

Zoom Out and look at the ***Coordinate System 3*** icon to see that is rotated and offset.

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Module 17:
GD&T

In this module, you will become familiar with inspecting your part using the Geometric Dimensioning and Tolerancing (GD&T) system.

Prerequisite: Basic Measurement, Coordinate Systems, Nominals and Tolerances, Working with CAD modules.

MODULE OBJECTIVES

You will be able to:

- Apply GD&T tolerances to check a part.
- Assign datums and use them in specific GD&T tolerances.
- Edit features to only report GD&T tolerance results.
- Create a measurement report with GD&T tolerance results.

CAM2Q

GD&T in CAM2 Q

In CAM2 Q, you can add GD&T tolerance values to each feature to compare part measurements to one or more datum(s) and ensure that specific design and tolerance requirements are being met.

Some GD&T tolerances, such as Orientation, Runout, and Position, require a datum, or a theoretically exact plane or line (axis) from which a dimensional measurement is made. Form and Profile GD&T tolerances do not require a datum because they use simple geometry for comparison.

Some GD&T tolerances are only applicable to certain features; Therefore, the ***GD&T Tolerances*** panel only shows the available tolerances for a selected feature. The CAM2 Q help file details the steps to add GD&T tolerances to any feature.

When you add GD&T tolerances to features, other tolerance values are usually not necessary. In the ***Feature Information*** panel, you can clear, or not use, any unwanted tolerance value.

ACTIVITY: Review GD&T Tolerances Help Topics

In this activity, we will review the GD&T tolerancing information contained in the CAM2 Q help file.

1. New File	On the ***File*** menu, click ***New***.
2. Open the CAM2 Q Help file	On the ***Help*** menu, click ***Contents***. On the ***Contents*** tab, double-click the ***GD&T Tolerances*** topic.
3. Review GD&T Tolerances	Review the ***GD&T Tolerances*** topic. To review the next topic, click on the ***Next*** (>) button at the top of the screen, or select another topic from the menu tree in the ***Contents*** tab. Notice the feature type in the first step of each topic, and whether a datum is required.
4. Close the CAM2 Q Help file	Click the ***Close*** (X) button in the upper right corner of the CAM2 Q Help file.

ACTIVITY: Establish Datums

In this activity, we will establish datums for use in GD&T calculations.

1. New File	On the ***File*** menu, click ***New***.
2. Import CAD	On the ***File*** menu, select ***Import***, then select ***CAD***.
3. Select file	From the ***\CAM2 Q v1.5 Training Files*** folder, select ***Demo Part Q.x_t.*** Click ***Open***.
4. ***Measure*** or ***Template*** mode	Choose either Measure or Template mode. ☛ *Remember that in **Measure** mode the **Add Readings** command is automatically started after you add a feature from the **Measure** Menu. In **Template** mode, you will need to select the features and then start the **Add Readings** command.*
5. Add Features	On the ***Measure*** menu, click ***Plane***, ***2D Line***, and ***Circle***.
6. Add Readings	Measure and rename the features as Plane 1, Line 1 and Circle 9.
7. Assign Datums	● Right-click ***Plane 1***, mouse-over ***Assign a Datum***, select ***Plane A***. ● Right-click ***Line 1***, mouse-over ***Assign a Datum***, select ***Line B***. ● Right-click ***Circle 9***, mouse-over ***Assign a Datum***, select ***Plane C***.
8. Construct a measured coordinate system	On the ***Coordinate Systems*** menu, click ***Create Using the Wizard***. ● Select ***Plane 1*** as the Plane Feature. ● Select ***Line 1*** as the Line Feature. ● Select ***Circle 9*** as the Point Feature. Select the ***Align with:*** check box.
9. Create the coordinate system	Click ***OK***.

Make sure to add the optimal, not the minimal, number of points for each feature. Refer to Module 3, Lesson 1

ACTIVITY: Customize Labels

In this activity, we will review and customize the label options.

1. Hide Label	On the ***Features*** panel, select ***Coordinate System 1***. Right-click and clear the ***Label*** menu item to hide the label.
2. Show Labels	On the ***Features*** panel, select the plane, line, and circle. Right-click and select ***Label*** to show the feature's simple label. Right-click again and select ***Detailed Label*** to show the feature's detailed label.
3. Select Feature	On the ***Features*** panel, select ***Plane 1***.
4. Hide tolerances	On the ***Feature Information*** panel, clear the check boxes in the first column for every value except ***Flatness***.
5. Select Feature	On the ***Features*** panel, select ***Line 1***.
6. Hide tolerances	On the ***Feature Information*** panel, clear the check boxes in the first column for every value except ***Straightness***.
7. Select Feature	On the ***Features*** panel, select ***Circle 9***.
8. Hide tolerances	On the ***Feature Information*** panel, clear the check boxes in the first column for every value except ***Diameter*** and ***Circularity***.
9. ***Preferences*** command	On the ***Edit*** menu, select ***Preferences***. Select ***Display*** and the ***CAD View*** tab. In the ***Label Style*** drop-down, select ***Show Detailed Label Only***. Click OK.

ACTIVITY: Determine Perpendicularity

In this activity, we will determine the perpendicularity of the left edge of the part to Datum B.

1. Create and Measure the left edge	On the ***Measure*** menu, click ***2D Line.*** Measure the left edge of the part near Circle 11. Cancel the command.
2. Select the feature	From the ***Features*** panel, select ***Line 2***.
3. GD&T Panel	On the ***Edit*** menu, select ***GD&T...***
4. Create a new GD&T tolerance	On the ***GD&T Tolerances*** panel, click the ***Perpendicularity*** icon ⊥.
5. Tolerance to Datum B	Click in the right control frame and select ***B*** (Datum B).

For the remainder of this module, you will be using the ***GD&T Tolerances*** panel often. You may find it helpful to dock it to one of the ***Main*** window edges, or just close it and reopen it when you need it.

6. Hide tolerances	On the ***Feature Information*** panel, clear the check boxes in the first column for every value except ***Perpendicularity***.

ACTIVITY: Determine Parallelism

In this activity, we will determine the parallelism of the back edge of the part to Datum B.

1. Create and Measure the back edge	On the ***Measure*** menu, click ***2D Line.*** Measure the back edge of the part. Cancel the command.
2. Select the feature	From the ***Features*** panel, select ***Line 3***.
3. Create a new GD&T tolerance	On the ***GD&T Tolerances*** panel, click the ***Parallelism*** icon //.
4. Tolerance to Datum B	Click in the right control frame and select ***B*** (Datum B).
5. Adjust tolerance values	On the ***Feature Information*** panel, clear the check boxes in the first column for every value except ***Parallelism***.

ACTIVITY: Determine Position

In this activity, we will determine the position of Circles 1, 3, 5, 7.

1. Create and Measure ***Circle 1***	On the ***Measure*** menu, click ***Circle***. Rename and measure ***Circle 1***. Cancel the command.
2. Select the feature	From the ***Features*** panel, select ***Circle 1***.
3. Create a new GD&T tolerance	On the ***GD&T Tolerances*** panel, click the ***Position*** icon ⌖.
4. Material Condition	Click in the far right control frame and select ***RFS***.
5. Repeat on Circles 3, 5, 7	Repeat the above steps for Circles 3, 5, 7.
6. Select circles	On the ***Features*** panel, select Circles 1, 3, 5, 7.

7. Adjust tolerance values	On the ***Feature Information*** panel, clear the check boxes in the first column for every value except ***Diameter*** and ***Position***. Scroll down the ***Feature Information*** panel to edit the information for all four circles.

ACTIVITY: Configure a Measurement Mode

In this activity, we will set up the Measurement modes for the next few activities.

1. Device Control panel	Press the ***P*** hot key.
2. Settings	Click the ***Measurement Mode Setting***, or ***Settings,*** button.
3. Distance Based Mode	Select the ***Distance Based Mode*** and change the distance to `0.50mm`. Click ***OK***.

☛ *These are the available measurement modes:*

- **Single Point Mode**: The measurement device takes a single point.
- **Distance Interval Mode**: The measurement device takes points at a specified distance interval (specify the distance in the device settings).
- **Time Interval Mode**: The measurement device takes points at a specified time interval (specify the time in milliseconds).

4. Close Device Control Panel	Click the ***Close (X)*** button in the upper right corner of the Device Control panel.

ACTIVITY: Determine Profile of a Line

In this activity, we will determine the profile of a line, or curve.

1. Measure a plane	On the ***Measure*** menu, click ***Plane***. ☛ *Notice the **Mode** is set to now **Distance Interval** and the Distance is 0.500mm. You can change the mode back to single point before measuring, or "scan" the plane with these settings.* Measure the plane perpendicular to G as Plane 1. Cancel the command.
2. Create ***Polyline 1***	On the ***Measure*** menu, click ***Other***, click ***Polyline***. Select the ***2D Plane*** check box, and select ***Plane 2 - Actual*** from the drop-down list. ***Minimum Scan Distance***: `0.50mm` ***Auto Nominal Max Distance***: `1.00mm` ***Maximum Scan Distance***: `5.0mm` ***Probe Compensation***: Use Sampled End Click ***Type***: Free Hand ***Measure on CAD Side***: Selected ***Select Surface***: `CAD Assembly` Click ***Create*** and ***Close***.
3. Measure the surface with a single line	Make sure that the Measurement Mode is set to **Distance Interval**. Touch the Probe/SMR to curved surface "G", press the ***Front*** button, or ***G*** key, and "scan" a single line on the curved surface from left to right.
4. Select the feature	From the ***Features*** panel, select ***Polyline 1***.
5. Create a new GD&T tolerance	On the ***GD&T Tolerances*** panel, click the ***Profile of a Line*** icon ⌒.
6. Adjust tolerance values	On the ***Feature Information*** panel, clear the check boxes in the first column for every value except ***Line Profile***.

ACTIVITY: Creating A Report

In this activity, we will create a measurement report and save it to a .csv file.

1. Select Features	On the ***Features*** panel, select the three datums, the two lines, the four circles, and the polyline.
2. Access Report view	Click the ***Report*** tab.
3. Tabular layout	At the bottom of the ***Report*** window, select ***Tabular*** from the drop-down list.
4. View results	Use the scroll bar or the page navigation icons to view each page of the report, and see the measurement results in the GD&T format.

ACTIVITY: Save The Measurement File

In this activity, you will save your work in the measurement file.

1. ***Save As***	From the ***File*** menu, select ***Save As***.
2. Your folder	Browse to your folder and type `GD&T.fcd`.
3. ***Save***	Click ***Save***.

CAM2Q

Module 18:
Using Multiple Devices

In this module, you will become familiar with inspecting a part using more than one measurement device.

Prerequisite: Basic Measurement, Move Device modules.

MODULE OBJECTIVES

You will be able to:

- Properly connect to and switch between multiple devices
- Add readings using multiple devices

CAM2Q

Module 18: Using Multiple Devices

You can connect and use multiple measurement devices simultaneously, with a single software license, without changing computers and merging data. You can connect any combination of FaroArms and/or FARO Laser Trackers.

This procedure to connect and define multiple devices involves measuring a set of point-reducible features with each device, assigning features from the second device as nominals to the features from the first, and then using the ***Move Device*** command to define the position.

ACTIVITY: Configuring Two Devices

In this activity, we will connect two FaroArms and measure with both of them.

1. New file	On the ***File*** menu, select ***New***.
2. Review Help	Press the ***F1*** hot key. Click on the ***Index*** tab. Type `multiple devices` in the ***Type in the keyword to find*** box. Double-click ***multiple devices*** from the list. Review the procedure for using multiple devices. ☛ *If you are using two FARO Laser Trackers, you must assign two additional hot keys for the second Laser Tracker. These new keys will function the same as the **G** and **H** keys since **G** and **H** are currently assigned to the first device.*

After you locate the point-reducible features that you want to use, ensure that each device can reach each feature.

3. Connect both devices	Connect all cables for each devices to your computer, power on your devices and computer, and start CAM2 Q. You may have to open the ***Device Control Panel*** and manually connect any device that has not automatically loaded.
4. Manage both devices	On the Device menu, select ***Device Manager***. If the second device is not in the Devices/Positions window, click the ***Add*** button and select the second device from the ***Serial Number*** drop-down list. ● Select the first device and rename it ***Device 1***. ● Select the second device and rename it ***Device 2***. Click ***Close*** to exit the ***Device Manager*** dialog box.
5. Measure features with ***Device 1***	Measure four points, one plane, and one circle. Use ***Device 1*** to measure each feature.

6. Measure features with ***Device 2***	On the ***Features*** panel, select all six features. Right-click and select ***Add Readings From***, Select ***Device 2***. Measure the features with ***Device 2***.
7. Solve Device Positions	• From the ***Device*** menu, select ***Device Manager***. The ***Device Manager*** panel opens and displays all of the connected devices. • In the ***Devices/Positions*** window, select Device 2's ***Device Position 1***. • Click the ***Add Features*** button to add all of the features to this dialog box. • Click the ***Solve*** button. CAM2 Q fits the two device positions together and reports the fit error. Look at the fit error (Error) for each feature and the maximum error (Max Error) for the results.

When you are analyze the fit error and the maximum error, both of these numbers should be within you part's fit tolerance. If necessary, you can remeasure any of the features with ***Device 2*** directly from this dialog box. Select individual features and click the ***Remeasure Selected*** or the ***Remeasure All*** button.

Special Conditions with Multiple Devices

When you have multiple devices connected to CAM2 Q, there are additional commands and options that are available.

- Default Device
  In the Device Manager panel, select a device and click the ***Make Default*** button. Measurement commands and the ***Add Readings*** command will always use the default device.

- Add Readings From
  On the ***Features*** panel, right-click a feature. The ***Add Readings From*** command will now list all available devices.

- QuickTool command properties
  Each QuickTool command can be assigned to a any device. On the ***QuickTools*** panel, select a command and then in the ***Basic*** tab of the ***Properties*** section choose a device from the ***Device*** drop-down. When playing the QuickTool, the first command for each device is started and multiple ***Measurement*** windows open.

Appendix I:
FaroArm Device Setup Checklist

Temperature Effect On A Part

- ❒ Part has been allowed to soak in the inspection environment to adjust to the air temperature.
- ❒ Measurements will be taken where the air temperature is relatively constant.
- ❒ Software is set to appropriately adjust for part expansion/contraction (see software user guide).
- ❒ The part is evenly exposed to light.

Temperature Effect On The Device

- ❒ The device temperature has been allowed to stabilize.

Movement

- ❒ The device and part are fastened and stationary.

Vibration Effects

- ❒ Vibrations (forklifts driving by, stamping machine movement etc.) have been avoided.

Mounting Stability

- ❒ Detailed instructions provided with mount (tripod, magnetic, or vacuum) were carefully followed.

If Using C-Clamps:

- ❒ The surface is clear of any debris which can cause the base to rock.
- ❒ Clamps are arranged on opposite sides of the base - 180 degrees from each other.

If Using A Custom Mount:

- ❒ The plate is large enough to mount the FaroArm using the Magnetic Mount or C-clamps.
- ❒ The plate is thick enough to prevent flexing.
- ❒ The plate is securely mounted to the part/fixture.
- ❒ A dial indicator was used to determine to verify the rigidity of the plate.

Appendix II:
Hot Keys

The following is a list of Hot Keys, or keyboard shortcuts, that you can use to quickly start some CAM2 Q commands.

Hot Key	Function
Enter	Repeat the Last command
Arrow keys 2,4,6,8 on numeric keypad	Pans the view in the specified direction.
F1	Help
F2	Measuring a Point
F3	Measuring a Plane
F4	Measuring a 2D Line
F5	Measuring an Inspect Surface
F10	Measuring a Circle
F11	Measuring a Cylinder
F12	Measuring a Sphere
Insert	Add Readings
G	Record a Measurement Reading
H	Record an End Click Reading
Backspace	Delete the Last Measurement Reading
SHIFT + A	Measure All
SHIFT + R	Remeasure
Alt + Enter	Edit Properties
4	Front View
5	Side View
6	Top View
7	Isometric View SE

Hot Key	Function
8	Isometric View SW
9	Isometric View NE
0	Isometric View NW
I Plus Sign (+) on numeric keypad	Zoom In
CTRL + =	Zoom In (Fine)
O Minus Sign (-) on numeric keypad	Zoom Out
CTRL + -	Zoom Out (Fine)
E	Zoom All
W	Zoom To Window
7 on numeric keypad	Rotate Around X Counterclockwise
9 on numeric keypad	Rotate Around X Clockwise
1 on numeric keypad	Rotate Around Y Counterclockwise
3 on numeric keypad	Rotate Around Y Clockwise
0 on numeric keypad	Rotate Around Z Counterclockwise
Period (.) on numeric keypad	Rotate Around Z Clockwise
S	Switch between the three shaded and wireframe views
Shift + Alt + Enter	Full Screen
Number Sign (#)	Grid
R	Reset View
L	Arrange Labels
ESC	Cancels the Current Measurement
SHIFT + R Mouse Mouse Wheel	Dynamic Zoom
R Mouse	Dynamic Pan
L Mouse	Dynamic Rotate
ESC	Cancels the Current Measurement

Hot Key	Function
P	Device Control Panel
D	Show Feature Window
A	Create and Edit Alignments
C	Coordinate System
M	Move Device
Delete	Delete any selected item
Enter	Repeat Last Command
Home	Home (Tracker Only)
T	Motor On Off (Tracker Only)
B	Set Backsight (Tracker Only)
V	Drive Beam (Tracker Only)
CRTL + N	New file
CRTL + O	Open file
CRTL + S	Save file
ALT + F4	Exit CAM2 Q

Appendix III:
Glossary of Terms

TERM	FARO Definition
3D	Three dimensional
3D distance	The distance between a nominal feature's center and a measured/constructed feature's center.
3D dynamic	Refers to the rotation of a computer model of a part in virtual space.
3D image	The three dimensional representation of an object
3D modeling	The creation of a three dimensional mathematical model of an object; typically performed using digitizing devices.
3D surface	The three dimensional representation of the boundary of a region in space
A	
Accuracy	The deviation between the measured value and the nominal or actual value.
Actual	A feature or measurement that exists and is not merely potential or theoretical.
Alignment	The establishing of a coordinate system on a part, typically in reference to a coordinate measurement machine.
Angle	The figure defined by two lines extending from the same point
Apex	In reference to an angle, the point from which the two lines of the angle emanate. Vertex.
Arc	A segment of a circle
B	
Basic Measurement Strategy, CAM2 Q	• Look at what you need to measure and consider the following questions: • What should you include in the formal Report? • Which features (geometric elements) do you need to capture? • Which features can you Measure directly (i.e. planes, circles, cylinders, lines, etc)? • Which features do you need to Construct (e.g. bolt hole circle) • Which Dimensions do you need? (e.g. circle center to circle center) • What logical sequence should you use to move around the part and capture the features? For example, left to right, top to bottom, or measure all planes first, then circles, etc.

TERM	FARO Definition
Best fit	Creating a feature that fits a set of points so each point is the shortest distance to the feature.
Bisect	To divide into two equal parts
Bisector	A feature (line or plane) which is created and equidistant at all points from two existing features.
Bolt hole pattern	A group of holes which is intended to accept bolts from a mating part in an assembly.
Button	Refers to the switches on the FaroArm
C	
CAD	Computer-Aided Design
CAD/CAM	CAD refers to computer-aided design. CAM refers to computer aided modeling. Some software packages perform both functions and are referred to as CAD/CAM.
CAE	Computer-Aided Engineering
Calibration	The procedure by which the FaroArm is optimized to perform accurate measurements. Thousands of data points are captured to determine the true geometry and kinematics of each FaroArm. This electronic "finger print" is stored on a chip in the base of the device.
CAMM (CAM2)	Computer-Aided Manufacturing Measurement
Capture (of data)	Digitizing. Storing data points in the computer system.
Circle	A closed planar curve in which every point on the curve is equidistant from the center.
Circularity	The quantity that refers to the fit of the input data points to the resultant circle.
CMM	Coordinate Measuring Machine
Coincident Points	Points that have the same XYZ value.
Collinear	Data points which lie on the same line.
Compensated points	Refers to the compensation for the radius of the ball probe on the FaroArm.
Computer aided design (CAD)	Software which replaces or augments manual drafting and design. Allows parts to be designed using computer software, and stored in an electronic database.
Cone	A basic or prismatic feature defined as a region generated by revolving a right triangle about one of its legs.
Conicity	The deviation of measured data points from the resultant cone.
Construction	A feature developed from measured features

TERM	FARO Definition
Coordinate Measuring Machine	CMM. These machines capture 3D data from objects to give the position (XYZ) of the object.
Coordinate System	A system of representing points in a space of given dimensions by coordinates, such as the Cartesian coordinate system or the system of celestial longitude and latitude.
Coplanar	Points or features which lie in the same geometric plane.
Corner	The point resulting from the intersection of three planes.
Counter-balance	The internal system which supports the weight of the FaroArm allowing for greater operator control and reduced operator fatigue.
Curvature	The deviation of a feature from a straight line.
Curvature change	The change in the slope of a curved feature.
Custom probes	Probes other than those that are supplied with the FaroArm as a standard. Customers can manufacture their own probes, and calibrate them using the 1" reference sphere.
Cylinder	A geometric feature formed by extruding a circle along its centerline in a direction normal to its plane.
Cylindrical	A part which is similar to a cylinder in shape.
Cylindricity	The deviation of measured data points from the resultant cylinder.
D	
Datum	A datum (plural datums or data) is a reference from which measurements are made. In engineering and drafting, a datum is a reference point, surface, or axis on an object against which measurements are made.
Datum Coordinate	The XYZ values of a feature used to establish an alignment.
Degrees of Freedom (DOF)	The FaroArm is termed to have 6 or 7 degrees of freedom. Each axis of rotation is defined as a degree of freedom.
Device	A piece of equipment or a mechanism designed to serve a special purpose or perform a special function. Measurement device; FaroArm or FARO Laser Tracker.
Dial indicator	Dial indicators are instruments used to accurately measure a small distance. They may also be known as a Dial Gauge, Dial Test Indicator (DTI), or as a "clock".
Diameter	The width of a circular of cylindrical featre.
Dimension	The relationship (distance or angle) between two geometric features.
Dimensioning	The function of determining the relationship between geometric features.

TERM	FARO Definition
Distance	The degree of separation between geometric features.
Drawing	The engineering documentation that indicates how a part is to be manufactured, assembled, or installed.
DRO	Digital ReadOut. The display of XYZ coordinates on the screen.
E	
Edge points	Data points which lie on the edge or border of a part.
Elbow	Refers to the joint of the FaroArm at the intersection of the two long tubes.
Ellipse	A geometric feature resulting from the intersection of a plane with a cylinder at an angle other than 90 degrees.
End Click	To accept collected readings by either clicking the FaroArm BACK button or for the Laser Tracker pressing the H hot key.
Entity	A feature or object in a CAD database.
Evaluation / to evaluate	The review of a product or process for suitability for its intended purpose.
External shape	The boundary which encloses a geometric form.
F	
Fabrication drawing	An engineering drawing that specifies how a part is to be manufactured.
Feature	Refers to an attribute of a product or a geometric object.
Fit	How easily two mating parts can be assembled.
Flatness	The deviation of the measured data points from the resultant plane.
Form	The maximum bandwidth (Max. Error added to the Min. Error) of error that a set of measured points deviates from the true form of the resultant feature calculated from that set of points.
G	
GD&T	Geometric Dimensioning and Tolerancing. A system of specifying engineering design and drawing requirements with respect to actual function and relationship of part features.
Geometric features	A shape that can be expressed using standard shapes i.e. circle, line, and arc.
H	
Handle	Refers to the pistol style handgrip of the FaroArm.
Hardware	Refers to the mechanical portion of a computer-based system. Opposite of software.

TERM	FARO Definition
Hardware lock	Dongle or port lock. Provides software security using a computer chip that attaches to the parallel (printer) port on a computer.
Hot Keys	One keystroke commands which invoke a software function.
I	
Inputting	Refers to the keying in of data.
Intersection	A meeting or crossing at a point.
Isometric	A "3D" view of a part from a direction which is equiangular from each coordinate axis.
Iterative Alignment	A means of aligning measured data to CAD data by best fitting through many iterations.
J	
Joint	The meeting place of two adjacent articulating units. Refers specifically to the transfer cases on the FaroArm.
K	
Key-in	To manually enter data using a keyboard.
Key-in alignment	The software function that creates a new coordinate system based upon translation and rotation relative to an existing coordinate system.
L	
Layer	A software tool used to group features and entities.
Line intersect	A type of alignment based upon a plane and two lines.
Line-to-plane	A type of angular dimension.
M	
Max. (maximum) Error	The largest distance from a reading above or outside a best-fit feature.
Maximum Material Condition	(MMC) Maximum material condition is that condition of a part feature wherein it contains the maximum amount of material within the stated limits of size. For a hole, the MMC is defined as the condition when the diameter is at the lowest end of its tolerance band. For a stud, the MMC is defined as the condition where the diameter is at the highest end of its tolerance band.
Measure	To capture data points to determine the size, position, and form of feature.
Mid-Point	A point which is equidistant from two other points.
Min. (minimum) Error	The largest distance from a reading below or inside a best-fit feature.
MT	Material Thickness

TERM	FARO Definition
N	
NIST	National Institute of Standards and Technology
Nominal	The theoretical value for a feature.
Normal	The vector which defines a direction perpendicular to a curve or surface at a specific point.
O	
Off-line	Any function which does not require a specific device to be connected.
Offset	To create a feature that is equidistant by a specified distance at all points.
On-line	A function which indicates that a specific device must be connected to perform a specific function.
Orientation	In reference to Geometric Dimensioning and Tolerancing, it is the type of dimension that pertains to the orientation of one feature relative to a datum feature. Such as Parallelism, Perpendicularity and Concentricity.
Origin	The point from which the axes of a coordinate system emanate.
Orthogonal coordinates	Coordinates which are perpendicular.
P	
Pan	In CAD, to change the position of the viewing window relative to the model.
Parallel	Two lines or planes which are equidistant at all points.
Perpendicular intersect	A type of alignment which requires a plane, line and point, where the origin is defined at the projection of the point on to the line.
Plane	A geometric feature defined by a point and a vector. A flat surface.
Plane-to-Plane	A type of distance dimension which yields the perpendicular distance between the centroid of the second plane and the first plane.
Point reducible feature	A feature that has a repeatable center point such as a circle or sphere.
Point-to-Line	A dimension type which yields the perpendicular distance between the point and the line.
Point-to-Point	A dimension type which yields the 3D distance between two points.
Polyline	A multi-segmented line.
Position	The location of a feature in a Cartesian coordinate system.

TERM	FARO Definition
Prismatic	Taking the shape of a prism or 3 dimensional object whose faces are represented by regular geometric features.
Prismatic features	A feature that can be represented by regular geometric shapes. The opposite of curves and surfaces.
Probe	End effector or stylus. The part of the FaroArm that touches the measurement piece. The FaroArm can use a variety of probes including hard probes, touch-trigger, and laser probes.
Probe compensation	The ability of the software to account for the radius of the probe. Gross errors can result if this is not done correctly.
Project to	To reproduce in a prescribed direction.
Q, R	
Readout	The visual display of data on a computer screen.
Re-capturing	In reference to digitizing, the re-measurement of erroneous data points.
Rectangular	A polygonal shape whose corners intersect at right angles.
Repeatability	The ability of a device to obtain consistent results. Although the terms are generally used interchangeably, repeatability differs from accuracy in that a device can consistently obtain the wrong result.
Resolution	The number of decimal places that a measurement device can reliably display.
Reverse engineering	The creation of design or CAD data from an existing or prototype part.
Rotation	Angular motion about a specified point or axis.
Roundness	The deviation of the measured data points from the true form of the resultant circle.
S	
Scanning	To capture large quantities of data points quickly.
Slot	An elongated hole with round or squared ends.
SMR	Spherically Mounted Retroreflector
Solve, to	To calculate the results of a problem.
Spherical	A shape which is similar to that of a sphere.
Spherically Mounted Retroreflector (SMR)	The probe for the FARO Laser Tracker.
Sphericity	The amount by which the measured data points deviate from the resultant sphere.

TERM	FARO Definition
Spline	A smooth curve which cannot be represented by a simple mathematical function.
Straightness	The amount by which the measured points deviate from the resultant line.
Stream	A continuous input of data points.
Surface	The outer boundary of an object.
Surface off-set	An output surface that is equidistant from an input surface at all points.
Surfacing	The technique of creating 3D models by defining the outer boundary of objects in virtual space.
T	
Temperature Compensation	The ability of a measurement device to adjust to changes in ambient temperature. A measurement device that is temperature compensated will maintain its accuracy through a wide range of temperatures. A device that is not temperature compensated cannot.
Thread	A screw thread. A machined surface with a helical shape. Normally parts are fastened together with a screw and a threaded hole or a threaded bolt and nut.
Thread mount	A metal piece that is used as an interface between a mating thread and a tripod or a mating thread and a table mountable device.
Tolerance	A zone of accuracy in both size and placement of a feature. For example, a hole of a diameter of 1.00" +/- 0.01 located at the x, y, z of 1.000, 1.000, 0.000 +/- 0.005 means a 1 inch diameter hole can be between 1.01 and 0.99 in size and the location can be plus or minus 0.005 inches from the specified nominal location.
Touch trigger probes	An electro-mechanical device that has a hard ball probe on the end of a pivoting straight shaft. When any movement of shaft is sensed at the pivot, an electrical message is sent to command the CMM to take a data point.
Traditional CMM	There are many types of CMMs that fall into this category. These include the bridge type, cantilever type, gantry type, etc. These are floor-mounted machines that require you to bring the parts to the CMM, unlike FARO's "portable" CMM, the FaroArm®.
True Position	Refers to a GD&T callout that specifies how a feature such as a hole is to be positioned on a part as well as toleranced. A true position callout may also specify additional tolerance or "bonus" if the hole is over-sized or if a shaft is under-sized (maximum or least material condition; MMC or LMC).

TERM	FARO Definition
U	
Unit Vector	A directional line with a non-dimensional magnitude of one. The line may be associated with the normal vector of a surface at a specific location. The vector is described in relation to the current alignment using the letters I, J, and K that are associated to X, Y, and Z.
USB	Universal Serial Bus (USB) is an external peripheral interface standard for communication between a computer and external peripherals over an inexpensive cable using biserial transmission. USB is intended to replace existing serial ports, parallel ports, keyboard, and monitor connectors and be used with keyboards, mice, monitors, printers, and possibly some low-speed scanners and removable hard drives.
V	
Vector	A term used to describe the direction of a line or object that may or may not have a magnitude. The line may be associated with the normal vector of a surface at a specific location. The vector is described in relation to the current alignment using the letters I, J, and K that are associated to X, Y, and Z.
Vertex	The point of intersection of two sides of an angle.
Volume	The amount of space occupied in three dimensions.
W	
Wrist	Refers to the last grouping of joints on the end of the FaroArm.
X, Y	
X, Y, Z	Refers to the Cartesian Coordinate System for three-dimensional space.
Z	
Zoom, to	A command used to change the view of the design on the screen, specifically to show a closer view.

Appendix IV:
Technical Support

FARO Technologies, Inc. is committed to providing the best technical support to our customers. Our Service Policy is detailed in Appendix C: Industrial Products Service Policy of any FARO manual. If you have any problem using one of our products, please follow these steps before contacting our Technical Support Team:

- Be sure to read the relevant sections of the documentation to find the help you need.
- Visit the FARO Customer Service area on the Web at www.faro.com to search our technical support database. This is available 24 hours a day 7 days a week.
- Document the problem you are experiencing. Be as specific as you can. The more information you have, the easier the problem will be to solve.
- If you still cannot resolve your problem, have your device's Serial Number available before calling.

Support Hours

- North America:
  8:00 a.m. to 8:00 p.m. Eastern Standard Time (EST).
- Europe:
  8:00 a.m. to 5:00 p.m. Central European Standard Time (CET).
- Asia:
  8:30 a.m. to 5:30 p.m. Singapore Standard Time (SST).
- Japan:
  9:00 a.m. to 5:00 p.m. Japan Standard Time (JST).
- China:
  8:30 a.m. to 5:30 p.m. China Standard Time (CST).
- India:
  9:30 a.m. to 5:30 p.m. India Standard Time (IST).
- You can also e-mail or fax any problems or questions 24 hours a day.

Phone

- North America:
  800 736 2771, +1 407 333 3182 (Worldwide)
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  infoindia@faro.com

E-Mails or Faxes sent outside regular working hours

(8:00 a.m. to 5:00 p.m., Monday through Friday) usually are answered before 12:00 p.m. the next working day. Should our staff be on other calls, please leave a voice mail message; calls are always returned within 4 hours. Please remember to leave a detailed description of your question and your device's Serial Number. Do not forget to include your name, fax number, telephone number and extension so we can reach you promptly.

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Contact Information

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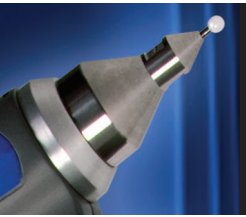

CAM2®

FaroArm:
Training Workbook

FARO® Institute