



## **PC-DMIS Touch 3.0 - CMM User Manual**

May 18, 2015



(For use with a T-Gage Bridge Coordinate Measuring Machine)



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# PC-DMIS Touch Overview

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## Introduction: About PC-DMIS Touch

PC-DMIS Touch is a precision measurement software made specifically for touch-enabled tablets and screens so that you can create and measure part files on a standard T-Gage CMM.

Before you can use PC-DMIS Touch, you need to install it onto your computer. The installation is controlled by License Management Software (LMS). For installation instructions, see the [ReadmeForTouch.pdf](#) that comes with the Touch installation files.

To get started, view the "[Getting Started](#)" chapter. For an overview of the documentation, see the "[About this Documentation](#)" topic.

For general usage instructions, see the Getting Started Guide that comes with the software's installation files.

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## About this Documentation

While documentation already exists inside the actual software itself as interactive help callout buttons of simple text, this documentation here provides additional details, explanations, and diagrams beyond what can be covered in a simple text-based callout.

If this is your first time using the software, you should see the topics in the "[Getting Started](#)" section of topics first.

The topics under the "Common Tasks" chapter detail the concepts, procedures, and reference topics you can use to create, learn, and run a part file in order to end up with a report of measurement results. Generally, you can follow these in the order they are listed.

The topics under the "General Reference Topics" chapter are useful reference topics that are not directly applicable to just one section.

References between topics are provided in the documentation as needed.

**Note:** This documentation is geared toward using the software on a touch-enabled device or screen. Therefore, it so uses terminology such as "tap", "swipe", "pinch", "stretch", and so on. However, you can also run the software on a standard desktop configuration with a mouse and keyboard.

## Getting Started

- [About Getting Started](#)

## Common Tasks

- [Calibrating a Probe](#)
- [Selecting a Probe and Tip](#)
- [Using a Probe Changer](#)
- [Working with Part Files](#)
- [Measuring Features](#)
- [Creating an Alignment](#)
- [Working with Faces](#)
- [Defining Dimensions](#)
- [Constructing Features](#)
- [Using DCC Mode](#)
- [Running Your Part File](#)
- [Working with Reports](#)

## General Reference Topics

- [About Touch Gestures](#)
- [About Nominals, Tolerances, and Axes](#)
- [The Commands List](#)
- [About the Status Icons](#)
- [About Decimal Places](#)
- [The Settings Tile](#)
- [The Settings Screen](#)
- [The Graphics Window](#)
- [About Advanced Properties](#)
- [About Temperature Compensation](#)
- [The Tools Screen](#)
- [About Keyboard and Keypad Input](#)
- [Common Jog Box Buttons](#)

# Common Tasks

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## Getting Started

### About Getting Started

This chapter helps you run the PC-DMIS Touch software for the first time. You should have already installed the software and configured its license as instructed. If not, please refer to the installation instructions in the Readme file that came with your software.

To get started, follow in order these topics below:

1. [How to Start the Software](#)
2. [How to Update the Software](#)
3. [The User Interface](#)

You should then be at the [Home screen](#) and should have a basic understanding of how the user interface works.

If you need further help, consult the rest of this documentation, or tap the help button  inside the software itself to view help callouts on any screen element in the software. If you want to view an electronic version of the printed quick start guide that came with your software, tap the help button and then tap the book button . The quick start guide provides an overview of the software of the software in a tutorial format.

**Related Topics:** [The Splash Screen](#), [The Home Screen](#), [The User Interface](#)

## How To

### How to Start the Software

**Objective:** To start PC-DMIS Touch.

**Concepts:** [Splash Screen](#)

**Prerequisites:** You must have an installed version of PC-DMIS Touch.

**Follow this procedure:**

1. Tap **Start** and then point to the program folder containing the executable.
2. Tap the PC-DMIS Touch shortcut to access the splash screen.
3. If needed, from the splash screen, [change the language](#) the software runs in.
4. If needed, from the splash screen, [update the software](#).
5. Tap anywhere to launch the software.
6. Once the software launches, it connects to PC-DMIS and to your measurement device. While the software connects to the controller and initializes the CMM, the [New tile](#), [Files tile](#), and [Tools tile](#) are inaccessible. In place of the **New** and **Files** tiles, the software displays connection status messages and other prompts. A message asks which probe is currently mounted on the CMM. The software highlights in a white box the probe it assumes is connected to the CMM.
7. *If a probe changer is defined*, it lists the probes held by your [probe changer](#).
  - a. Specify the probe currently connected to the CMM by tapping on the correct probe.
  - b. If no probe is attached to the CMM, meaning the probes are unloaded into their slots on the probe changer, tap **NOPROBE**. You can also use **NOPROBE** to move a probe attached to the CMM back to the probe changer.
8. *If a probe changer is NOT defined*, it displays a list of all available probes.
  - a. Filter the list as needed.
  - b. Specify the probe currently mounted on the CMM by tapping on the correct probe.
9. Tap **Done** to proceed.
10. Once the software connects to the CMM and the mounted probe is specified, the **New**, **Files**, and **Tools** tiles are enabled for use. When you [create a new part file](#), the software inserts the chosen [probe tip command](#).

## How to Change the Language

**Objective:** To change the language you see when you run the software.

**Concepts:** [The Splash Screen](#)

**Prerequisites:** You must have an installed version of PC-DMIS Touch.

**Follow this procedure:**

1. Tap **Start** and then point to the program folder containing the executable.
2. Tap the PC-DMIS Touch shortcut to access the splash screen.
3. Tap the list in the upper-left corner of the screen, and select the desired language.
4. Tap anywhere on the splash screen to launch the software in the selected language.

**How to Update the Software**

**Objective:** To download and install new software updates.

**Concepts:** [The Splash Screen](#)

**Prerequisites:** You must have an installed version of PC-DMIS Touch and an active Internet connection. You must have administrator privileges to download and install any updates; however a user with standard privileges can check if an update exists.

**Follow this procedure:**

1. Start PC-DMIS Touch. See "[How to Start the Software](#)".
2. From the splash screen, tap **Information** .
3. Tap **Check for updates** to show the Hexagon software updater.
4. If an update appears, tap **Download Update**.
5. Once it downloads, tap **Install Update**, and follow the instructions in the installation wizard.

**Reference Topics****The User Interface**

**Terminology Notes:** To help explain the user interface, the following terminology conventions are used in this documentation:

- **Tile** - Tiles are groupings of the most recently selected and the most popular items. They only exist on the top level of screens. You can swipe through the tiles horizontally.
- **Screen** - The term "touch screen" refers to the actual hardware screen on your mobile device that you touch with your finger to interact with the software.

The term "screen" preceded by descriptive or user interface text (such as "Home screen", "Files screen", "Reports screen", and so on) refers to a specific area in the user interface that covers the named functionality.

Some screens reside on top of other screens. They are used to create alignments, measurements, dimensions, constructions, and so on.

- **Buttons** - These are touchable areas on tiles and screens, or on toolbars that take you to another screen or perform a specific function.
- **Toolbar** - A row or column of buttons in its own container for quick-access functions.

The main user interface for PC-DMIS Touch is laid out in a series of tiles and screens that you can [swipe](#) horizontally in order to view and access available items. Tapping a button in a tile or a button on a toolbar opens the associated screen or performs the associated action.

Here's a brief overview of the screens and toolbars and their purposes:

- **Splash** screen - This is the first screen that appears when you launch PC-DMIS Touch but before you launch the main application and connect to your measurement device. You use it to change the language, update the software, or connect to your device. Once you tap on this screen, the software connects to the device and displays the **Home** screen. For more information on the splash screen, see "[The Splash Screen](#)".
- **Home** screen - This is the main screen in the software and is accessed when the software connects to PC-DMIS. Tapping  from the lower level screens returns you to the **Home** screen. As you swipe to the left the **Home** screen contains the tiles in the top level of screens. For more information, see "[The Home Screen](#)".

### Levels of Tiles or Screens

There are three levels of tiles and screens:

- **Top Level** - The top level, known as the **Home** screen, contains these tiles: [New tile](#), [Files tile](#), [Reports tile](#), [Tools tile](#), and [Settings tile](#). Swipe left or right as needed. Tiles only exist at this top level of the user interface. They show only the most recent and most popular items. On the **Files** and **Reports** tiles, if the item or functionality you seek is not on the

tile, tap on the **More Files** or **More Reports** buttons to open the associated screen from the middle level.

- **Middle Level** - The middle level contains these screens: [New File screen](#), [Files screen](#), [Reports screen](#), [Tools screen](#), and [Settings screen](#). Swipe left or right as needed. These are similar to the tiles on the top level except that they contain a comprehensive view of all of the available options for the selected item. You can quickly get back to the top level

**Home** screen by tapping **Home**  from any middle level screen.

- **Bottom Level** - The bottom level contains these screens: [Properties screen](#), [Edit screen](#), [Report screen](#), and [Run screen](#). Swipe left or right as needed.

In addition, some screens lie on top of other screens. These are those screens:

- [Define Clearance Cube screen](#)
- [New Dimension screen](#)
- [New Construction screen](#)
- [New Alignment screen](#)
- [New Measurement screen](#)
- [Print Preview screen](#)
- [Edit screen](#) (for commands)
- [Probe Calibration screen](#)
- [Add Calibration Tool screen](#)
- [Probe Changer Setup screen](#)
- [Probe Changer Calibration screen](#)
- [Probe Assignment screen](#)
- [Probe Builder screen](#)

### Other User Interface Elements

- [The Graphics Window](#) - This window appears within many screens to graphically represent features and the part's orientation.
- [The Commands List](#) - This list appears within many screens on the right pane. You can use it to see and select commands available in your part file.
- [Nominals, Tolerances, and Axes](#) - This pane appears within the [New Measurement](#), [New Dimension](#), and [New Construction](#) screens, allowing you to change nominal and tolerance values and control what information is sent to the report.
- [Decimal Places](#)

- [Status Icons](#) - These icons appear above the Commands list. They detail the status of the software, its connection to the device, and other information.

## The Splash Screen

This is the first screen that appears after you start PC-DMIS Touch but before you tap the screen to launch the main application. Before connecting to the device and launching the main application, you can use this screen to change the language for the software, view the current version, and update the software."

- To shut down PC-DMIS Touch, tap **Turn Off** .
- To change the language, tap the list in the upper-left corner of the screen, and select the desired language.
- To view information about this version of the software, tap **Information** .
- To check for software updates, tap **Information** , and then tap **Check for updates**.

If you tap anywhere else on the screen, the software displays the [Home screen](#) in the currently selected language, and then it attempts to connect to the CMM.

**Related Topics:** [How to Update the Software](#), [How to Change the Language](#), [How to Update the Software](#)

## The Home Screen

The software opens to this main screen whenever you start the software. While the software connects to the controller and initializes the CMM, the [New tile](#), [Files tile](#), and [Tools tile](#) are inaccessible. In place of the **New** and **Files** tiles, the software displays connection status messages and other prompts. Once the machine is connected, and you choose the probe currently connected to the machine, you can use those tiles as usual.

This screen makes up all of the tiles on the top level. Tapping **Home**  from any middle-level screen also shows the home screen.

As you swipe horizontally in **Home** screen, it contains the following tiles that provide access to other screens:

- [New](#) - Creates a new part file
  - [Files](#) - Shows the four most recently accessed part files
  - [Reports](#) - Shows the most recently viewed reports
  - [Tools](#) - Shows the most recently accessed or most popular hardware tools
  - [Settings](#) - Shows the most recently accessed or most popular settings
- 

## Calibrating a Probe

### About Probe Calibration

Probes need to be calibrated before they can be used to measure the part. Calibration tells the software the position of the probe tip in 3-D space. Probe calibration involves the probe tip taking touches on a measurement sphere, called a "calibration tool". You should perform a probe calibration whenever your probing hardware changes, such as installing a new probe tip, or when a calibrated probe grows stale.

The software checks to see if the current probe is calibrated whenever you [create a new part file](#), open an existing part file, [learn a feature](#), or begin to [run a file](#).

- A message in red that says the probe is "uncalibrated" means the probe tip has never been calibrated. You cannot run a part file with that probe until the probe is calibrated.
- A message in white that says the probe needs to be "recalibrated" means your probe is still calibrated, but the calibration is growing stale. You should probably recalibrate, but the software gives you the option of deciding to continue using the probe or not.
- The probe status icon also indicates a probe's calibration status.

If the calibration tool has been moved since the last time you calibrated the probes with it, during calibration you need to locate the tool by taking a single point on its top.

During calibration, the software compensates for temperature.

**Related Topics:** [How to Calibrate a Probe](#), [Probe Calibration Screen](#), [Installed Probes](#), [About Temperature Compensation](#), [The Probe Information Screen](#)

## How To

### How to Calibrate a Probe Tip

**Objective:** To measure the calibration sphere so that the software knows the probe tip's position in 3D space. PC-DMIS Touch calibrates all the tips on all probes attached to the probe changer and the machine.

**Concepts:** [About Probe Calibration](#), [The Probe Calibration Screen](#), [About the Probe Changer](#)

**Prerequisites:** You must have a calibration tool attached to your CMM table. The probes to calibrate must be attached to the machine or currently stored in a port on the probe changer. The probe changer must be calibrated.

#### Follow this procedure:

1. Launch the software and verify the current probe. For more information, see "[How to Start the Software](#)".
2. Ensure that the calibration tool (measurement sphere) you want to use is attached to the CMM table in the proper location.
3. From the **Home** screen, under the **Tools** tile, tap **Calibrate**  to access the **Probe Calibration** screen.
4. From the left pane, follow the instructions for each step. From the right pane, make all your selections. After you make the proper selection, the software automatically advances to the next step.
5. If you have not already done so, from the probe diagrams, tap the probe that is currently loaded onto the machine. It has a white border, like this:



6. For **Tool**, from the right pane, you see a list of colors. Select a color that corresponds to where the physical calibration tool is attached on your CMM. Each color represents a pre-defined position for your calibration tool (or sphere). If a yellow caution symbol appears next to the calibration tool, it means that the software doesn't know its location yet, and when you first

- try to calibrate with that tool, the software asks you to take a point on the top of the sphere to locate it.
- If you need to use a calibration tool that isn't in the list, tap **Add Tool** to add a new tool to the list. For information on adding a new calibration tool, see "[How to Add a Calibration Tool](#)".
  - After you select a calibration position, and verify the current probe, you are ready to calibrate the probe.
7. For **Calibrate**, from the right pane, tap **Start**.
  8. Follow the instructions on the screen. The following occur during calibration:
    - The **Start** button changes to a **Stop** button. You can tap **Stop** at any time to stop the calibration process and any machine motion. If you do this while the CMM probes the calibration sphere, the software displays a red "The host sent an ABORT command" error message to indicate that the process was aborted.
    - The probe in the left-most port is the master probe. It is always measured first. Calibration then moves from left to right until all the probes are calibrated. A probe highlighted in **Blue** means it is currently being calibrated.
    - If the calibration tool is not in the same place as when you last calibrated your probe or probes, the software prompts you to measure a point on the top of the tool's sphere.
      - a. Use the jog box to move the probe above the sphere, and then measure a point on the sphere's top. The software prompts you to press DONE on your jog box.
      - b. From the jog box, press the DONE  button.
      - c. The software then drives the machine in DCC mode and finishes measuring other points on the sphere.
    - Probes are automatically swapped in and out of the probe changer as they are calibrated.
  9. After the calibration finishes, the software displays the calibration results.
    - **Green** - means the probe has passed its calibration without any warnings or errors.
    - **Red** - means the probe has failed its calibration.
  10. For **Completed**, tap **Done**. Once you've calibrated the probe successfully, you can learn features for a new part file or execute existing part files.

## How to Add or Delete a Calibration Tool

**Objective:** To define a new calibration tool and position; or to remove a calibration tool you no longer need.

**Concepts:** [About Probe Calibration](#), [The Probe Calibration Screen](#), [The Add Calibration Tool Screen](#)

**Prerequisites:** [Start PC-DMIS Touch](#)

### To add a calibration tool, follow this procedure :

1. Access the **Add Calibration Tool** screen. From the **Probe Calibration** screen, tap **Add Tool**.
2. From the left pane, follow the instructions for each step. From the right pane, make all your selections. Depending on your choices, the software may automatically advance to the next step.
3. For **Tool ID**, choose the ID. From the right pane, either tap **Color** and then choose a color to represent your calibration tool; or tap **Custom** to define a custom ID. Once you make your selection, it appears under the **Tool ID** text on the left pane.
  - If you tapped a color ID, proceed to the **Shank Vector** step below.
  - If you tapped **Custom**, in the **ID** box, type a name for the new calibration tool, and then tap **Next**.
4. For **Shank Vector**, define the direction of the shank holding the calibration sphere. From the right pane, either tap **Axis** and choose the direction from a predefined axis; or tap **Custom** to define a custom IJK vector. Once you make your selection, it appears under the **Shank Vector** text on the left pane.
  - If you tapped **Axis**, tap the machine axes that corresponds to the direction of the calibration tool's shank: **Zplus**, **Zminus**, **Yplus**, **Yminus**, **Xplus**, **Xminus**. Proceed to the **Size** step below.
  - If you tapped **Custom**, tap **Shank I**, **Shank J**, and **Shank K** in turn to define the shank's vector. In each box, type the value making up the IJK vector. Tap **Next** to proceed.
5. For **Size**, from the right pane, tap **Diameter** and type a value that defines the nominal size of the calibration sphere on the probe calibration tool. The measurement units for this value is based on the units selected the

last time you created a part file. Tap **Next** to proceed. Once you define your selection, it appears under the **Size** text on the left pane.

6. For **Complete**, review the information in the left pane and tap **Done** to add the custom calibration tool. The calibration tool position is added into the right pane of the **Probe Calibration** screen. A yellow caution symbol appears next to the calibration tool indicating that the software doesn't know its location yet. When you use that tool in a calibration for the first time, the software asks you to take a point on the top of the sphere to locate it.

**To delete a calibration tool, follow this procedure:**

1. Access the **Probe Calibration** screen. For more information, see "[The Probe Calibration Screen](#)".
2. From the right pane, under **Select the calibration tool position**, tap the delete button  next to the calibration tool you want to delete. The calibration tool is removed from the list.
3. Tap **Cancel** to close the screen, or continue using the **Probe Calibration** screen to finish a calibration, and use **Done** to close the screen.

## Reference Topics

### The Probe Calibration Screen

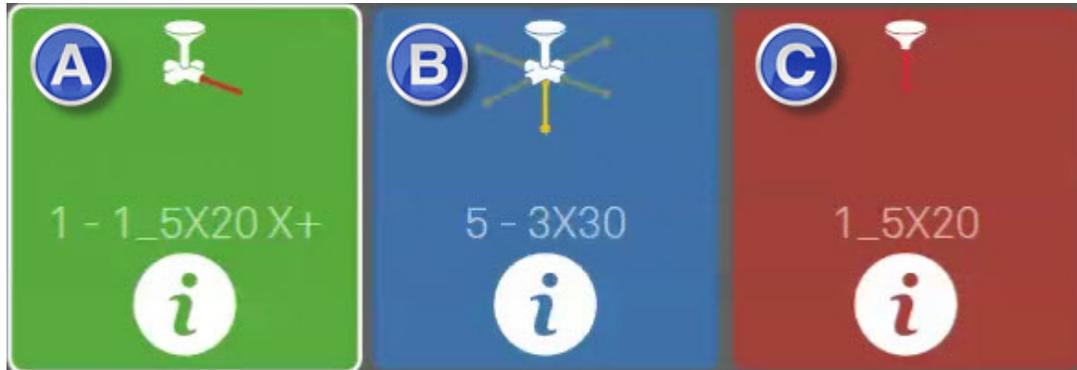
You should access this screen when you want to calibrate probes. To access this screen, from the **Tools** tile, tap **Calibrate Probe**:



The screen contains these items:

- **Header** - This shows "Probe calibration".
- **Left Pane** - The left pane shows the required steps you need to take to calibrate one or more probes stored on the probe changer. The highlighted step shows the current action you must take. Once an item is selected, it appears beneath the completed step, and a check mark appears next to that step.
- **Probe Diagrams** - The area between the left and right panes shows diagrams of the different probe ports located on the probe changer. You can use this area to verify the probe currently loaded on the machine

(indicated by a white border), view information on each probe in a probe changer, and view the calibration results. The following image represents a possible configuration of probes in the three ports:



- A. Port 1. The probe in the left-most port is considered the master probe, because the probe in port 1 is always calibrated first. Calibration moves from left to right until all the probes in the probe changer and attached to the machine are calibrated. If the master probe fails calibration, the calibration process stops.
- B. Port 2. The probe in this port is calibrated after port 1.
- C. Port 3. The probe in this port is calibrated after port 2.

The information icon under each probe  displays detailed information about the Probe loaded in that port.

The different colors indicate the status of the calibration and which probe is attached to the machine:

- **Green** - means the probe has passed its calibration without any warnings or errors.
  - **Red** - means the probe has failed its calibration.
  - **Blue** - means the probe is currently being calibrated.
  - A yellow tip means the tip's calibration is stale.
  - A white tip means the tip's calibration is good.
  - A white border around a probe means that probe is the one currently attached to the machine. You can tap on a probe to let the software know which one is currently attached.
- **Right Pane** - The right pane initially shows some colors in a list. Each color represents a pre-defined position for your calibration tool (or sphere). A yellow caution symbol appears next to a calibration tool if the software

doesn't know its location yet. When you use that tool in a calibration for the first time, the software asks you to take a point on the top of the sphere to locate it. If you need to add a new calibration tool, you can click **Add Tool**. You can also delete tools by tapping the delete button:



- After you select a calibration position, and verify the current probe, you are ready to calibrate the probe. From the right pane, you can tap **Start** begin the calibration process or tap **Stop** to abort the calibration process.

**Related Topics:** [About Probe Calibration](#), [How to Calibrate a Probe](#), [The Probe Information Screen](#), [The Add Calibration Tool Screen](#), [How to Add a Calibration Tool](#)

## The Add Calibration Tool Screen

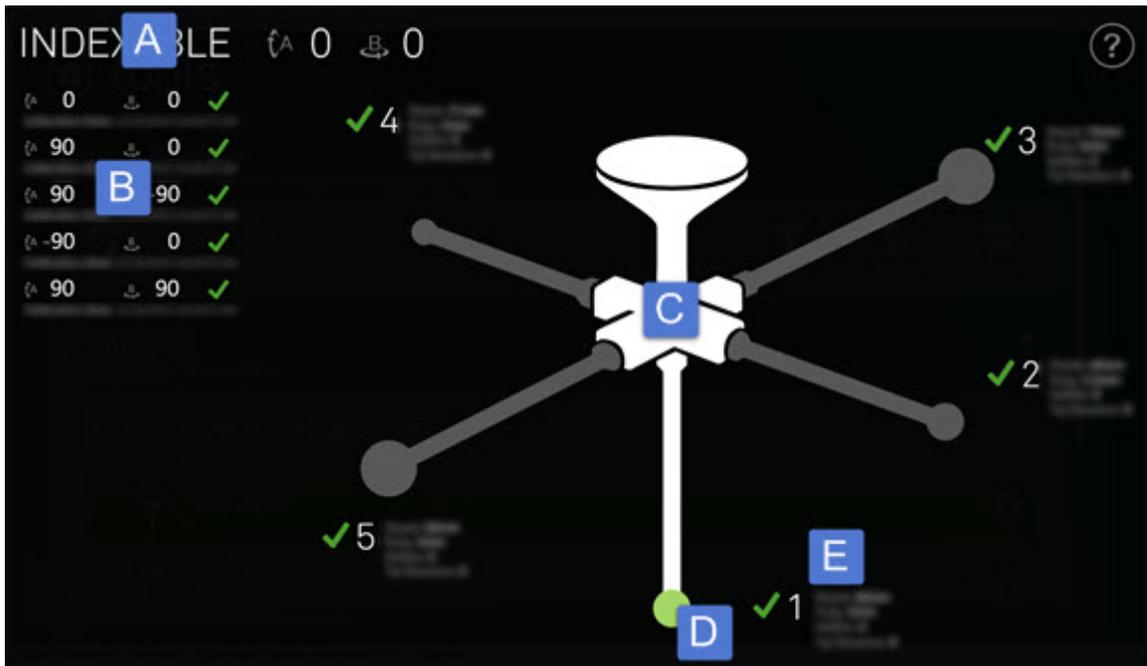
You should access this screen if you need to add a calibration tool that isn't in the current list of available tools. To access this screen, from within the **Probe Calibration** screen, tap **Add Tool**.

The screen contains these items:

- **Header** - This shows "Add calibration tool".
- **Left Pane** - The left pane shows the required steps to take to define a new calibration tool. The highlighted step shows the current action you must take. As you progress through the screen, your selections are displayed beneath each step.
- **Right Pane** - The right pane changes as you proceed through the steps on the left pane. Use this pane to make all your selections to define the tool's location ID, vector, and size.

**Related Topics:** [How to Add a Calibration Tool](#), [The Probe Calibration Screen](#)

## The Probe Information Screen



The Probe Information screen displays a diagram and information about the probe. The Probe Information screen shows the following items:

- A. The probe's name.
- B. The probe's AB angles (if any) and calibration date. You can click on an AB angle to see the specific calibration information for each tip at that angle. If a probe does not support multiple angles, the calibration date appears beneath the probe name.
- C. Diagram of the probe body.
- D. Diagram of the probe tip. The active probe tip shows in a brighter hue than other tips.
- E. A number for each tip used on the probe. The following information appears next to each tip number:
  - **Shank** - The shank's length
  - **Ruby** - The ruby tip's diameter
  - **StdDev** - The tip's standard deviation
  - **Tip Deviation** - The tip's probe radius deviation

### About Symbols and Colors

The software uses colored symbols, tips, and text to communicate the calibration status.

-  A green check mark appears if the tip is calibrated and ready to use.
- A white tip with a green ball, like that shown in the example image above, indicates a good calibration that is not stale.
- A yellow tip indicates a stale calibration.
-  A red warning symbol and a red tip indicate that a probe is not calibrated or has failed calibration.
- Red foreground text on the tip number indicates a failed calibration. In this case the corresponding standard deviation value or tip deviation value also turns red.

The Probe Information screen appears whenever you tap the a probe information icon:



The probe information icon is available from these locations:

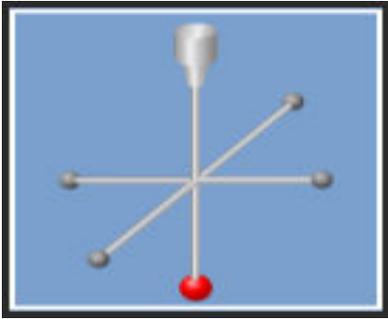
- [The Probe Assignment screen](#)
- [The Probe Command in the Commands list](#)
- [The Probe Calibration screen](#)
- [The Probe Builder screen](#)
- [The Tools screen](#)

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## Working with Probes, Tips, and Angles

### About Probes, Tips, and Angles

By default, PC-DMIS Touch expects that your probe stylus and tip point straight down from the CMM's ram (pointing in the Z- direction). However, you might have a probe stylus or multiple stylii that point in a different angle, as in the case of the star cluster probe where five tips point in different directions, like this:

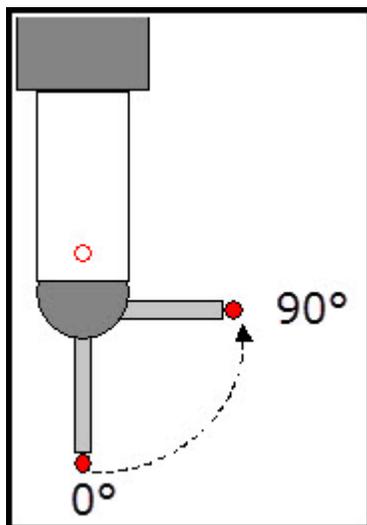


Depending on what you're measuring, you may need to define a different probe type, tip, or angle. For example, if the current probe tip is pointing in a certain direction, and you want to measure a feature or surface also facing in that same direction, you would need to tell the software to use a probe tip or angle that would be able to physically access that feature.

### Custom Probes and Probe Angles

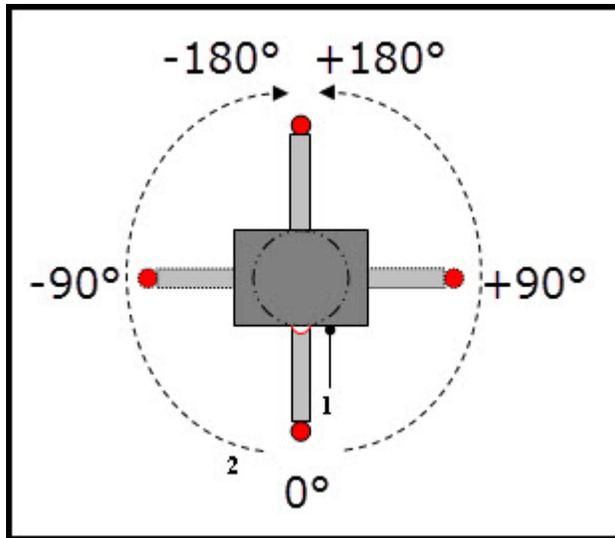
PC-DMIS Touch ships with [fixed probes](#) (probes that don't rotate), but it supports indexable probes. An indexable probe contains two rotating joints termed A and B that act like a wrist and can provide many AB angle combinations. For example, it can measure AB positions from  $0^\circ$  to  $90^\circ$  in the A axis and from  $-180^\circ$  to  $180^\circ$  in the B axis.

*A axis angle range:*



Front view of Probe with B =  $90^\circ$

*B axis angle range:*



Top view of Probe with  $A = 90^\circ$

- 1 - Front
- 2 - B Axis

A defined probe angle acts as a container of sorts. If you have a probe with multiple tips, such as a star cluster probe, those tips are contained within each defined angle. To define an indexable probe with specific angles, see the "[How to Define or Edit a Probe](#)" topic.

### Probe Tips

The available tips are the same probe tips contained within the three slots of the LSPX1C probe changer. For more information, on the LSPX1C probe changer, see "[About Probe Changers](#)". You can choose **NOPROBE** to cause the software to dock the current probe with the probe changer.

Probe tip commands in the Commands list have an icon next to them depicting the probe tip used. A star cluster probe, for example, looks like this:



### The Active Tip

The current probe tip command is called the "active tip". It is indicated with a brighter shank with a green tip if calibrated. All measurement commands that follow the active probe tip command are measured by that tip. Knowing which

probe tip is your active tip is also useful when measuring features on your part, because you'll have a better idea of how to move the probe when taking manual hits.

### Probe Tip Status Icon

As you tap on different commands in the Commands list, or as features are measured when running a file, a probe tip [status icon](#) appears above the Commands list. This icon depicts the probe tip the software uses when running the selected command. If the active tip for a selected command differs from the current tip physically attached to the CMM, the status icon displays inside of a circle with an up arrow, like this:



You can tap this warning icon to load the active probe from the probe changer. A prompt appears, and if you tap to confirm the probe change, the CMM moves to the probe changer and loads the new probe.

**Related Topics:** [How to Load a Probe from the Probe Changer](#), [How to Select a Probe Tip](#), [How to Learn Features to Measure](#), [About Probe Calibration](#), [How to Define or Edit a Probe](#), [Installed Probes](#), [How to Chose a Specific Angle on a Probe Tip Command](#), [How to Delete a Probe Definition](#)

## How To

### How to Insert or Modify a Probe Tip Command

**Objective:** Insert or modify a command into the part file that changes the probe tip used for subsequent features.

**Concepts:** [About Probes, Tips, and Angles](#)

**Prerequisites:** You must first create a part file, and [calibrate the probe tip](#). Your [probe changer](#) must be defined, positioned, and calibrated.

**Follow this procedure to add a new probe tip command:**

1. From the **Edit** screen, from the bottom toolbar, tap **Probe** . A probe tip command is inserted into the Commands list and it is placed in Edit mode.
2. From within the probe tip command, tap on the available probe images to select the desired probe. The available tips are the same probe tips contained within the three slots of the LSPX1C probe changer. For more information, see "[About the Probe Changer](#)".
3. If you choose **NOPROBE**, the probe currently attached to the CMM is moved back to dock with the probe changer. This also occurs when the software runs the **NOPROBE** tip command during Run Mode.
4. Tap **Accept** . An icon of the selected tip appears to the left of the probe tip command.
5. If the selected probe tip differs from the probe tip attached to the CMM, a prompt appears informing you that you must first move the probe to a safe location.
6. Move the probe to a safe location above the part.
7. Click **OK** to have the CMM move to the probe changer and attach the new probe.

**Follow this procedure to modify an existing probe tip command:**

1. From the Commands list, tap an existing probe tip command.
2. Tap **Edit Probe**  to place the command in Edit mode.
3. From within that command, tap on the available probe images to select the desired probe. The available tips are the same probe tips contained within the three slots of the LSPX1C probe changer. For more information, see "[About the Probe Changer](#)".
4. If you choose **NOPROBE**, the probe currently attached to the CMM is immediately docked with the probe changer. This also occurs when the software runs the **NOPROBE** tip command during Run Mode.
5. Tap **Accept** . An icon of the selected tip appears to the left of the probe tip command.
6. If the selected probe tip differs from the probe tip attached to the CMM, a prompt appears, informing you that you must first move the probe to a safe location.
7. Move the probe to a safe location above the part.

8. Click **OK** to have the CMM move to the probe changer and attach the new probe.

**Note:** Whenever you change a probe command (angles or probes), the probe status icon at the top of the screen changes to have a circle drawn around it. This indicates that what you have mounted on the CMM no longer matches the active probe command in the part file. To change the probe on the CMM so that it matches the current probe command, you need to tap that probe status icon. For more information on the probe status icon, see "[About the Status Icons](#)".

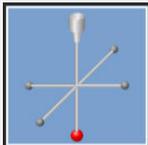
Subsequent commands use the defined probe tip until the next probe tip command.

For information on choosing a specific tip from a multi-tip probe, such as a star-cluster probe, see "[How to Choose the Active Tip on a 5-way Star Cluster Probe](#)".

For information on how to add a new command with a selected tip, see "[How to Insert or Modify a Probe Tip Command](#)".

## How to Choose a New Tip on a Star-Cluster Probe

**Objective:** To choose one of the tips on a star-cluster probe. A star-cluster probe has multiple tips pointing in different directions, such as this 5-way star-cluster probe:



**Concepts:** [About Probes, Tips, and Angles](#)

**Prerequisites:** You must first [create a part file](#), and [calibrate the probe tip](#). Your [probe changer](#) must be defined, positioned, and calibrated. You must have a [probe tip command](#) for the star cluster probe added into the part file.

### Method 1:

1. From the Commands list, tap an existing probe tip command for the star-cluster probe you want to modify.

2. Tap **Edit Probe**  to place the command in Edit mode.

3. Tap on the probe image for the star cluster probe repeatedly to cycle through and highlight the desired tip.
4. Tap **Accept**  to use the modified tip command. The image next to the probe tip command changes to reflect the new tip.

### Method 2:

1. From the Commands list, tap an existing probe tip command for the star-cluster probe you want to modify.
2. Tap **Tip**  to see an image of the probe, its tips, and the current tip.
3. Tap the left arrow  or the right arrow  to cycle through and highlight the desired tip.
4. Tap **Accept**  to use the modified tip command. The image next to the probe tip command changes to reflect the new tip.

### Method 3:

1. From the bottom toolbar, tap **Tip**  to insert a new probe tip command based off the current probe tip command into the Commands list but with the interface open to choose a new tip on a multi-tip probe.
2. Tap the left arrow  or the right arrow  to cycle through and highlight the desired tip.
3. Tap **Accept**  to use the modified tip command. The image next to the probe tip command changes to reflect the new tip.

## How to Define or Edit a Probe

**Objective:** To create a new probe definition in the software for unique hardware or to edit an existing probe's definition.

**Concepts:** [About Probes, Tips, and Angles](#), [The Probe Builder Screen](#)

**Prerequisites:** [Start PC-DMIS Touch](#) and select the mounted probe.

**To create a new probe or edit an existing probe, following this procedure:**

1. From the **Tools** tile on the **Home** screen, tap **Tools**  to access the **Tools** screen.
2. Tap **Probe Builder**  to access the **Probe Builder** screen.
3. From the left pane, follow the instructions for each step. From the right pane, make all your selections. Depending on your choices, the software may automatically advance to the next step.
4. For **Probe**, you can create a new probe or edit an existing probe.
  - a. To create a new probe, from the right pane, tap **New Probe** to proceed to the **Define** step.
  - b. To edit an existing probe, from the right pane, select the probe you want to edit, and tap **Next** to access the **Define** step.
5. The middle of the screen shows the a probe preview window and the component selection interface.
6. For **Define**, in the middle of the screen, use the component selection interface to build or edit the probe.
  - Build a Probe:
    - a. If the software highlights **No probe defined**, from the right pane, click the plus symbol to define a new probe component. On some configurations, PC-DMIS Touch automatically defines this first component. In this case, it highlights **Empty Connection #1**, the next available connection point for the next component you need to define.
    - b. For each empty connection point, select the empty connection. Then from the right pane, tap **Add** . Use the **Probe description** list to define the component for that connection point. Continue doing this until you define components for all empty connection points.
  - Edit a Probe:
    - a. From the component selection interface, tap a component you want to remove or modify.
    - b. *To remove a probe component entirely*, from the right pane, tap **Remove** . When you remove a component, the software also removes any components under it.

- c. *To modify the selected component*, from the right pane, tap  **Edit**. Then use the **Probe description** drop-down list to select the new component.
7. In the **New Probe Name** box, define a new probe name, or keep the existing name if you're editing an existing probe. Tap **Next** to proceed to the next step.
  8. If your probe is an indexable probe and supports angles, the software proceeds to the **Angles** step in the left pane. If your probe is a fixed probe, it proceeds to the **Completed** step.
  9. For **Angles**, from the right pane, define the angles you want to use. You can choose some common predefined angles, create a custom angle, or do both.
    - a. *To select common predefined angles*, from the right pane, tap on one of the check boxes to add those angle intervals into the **Included Angles** list on the right pane. The predefined angles are grouped by Angles with A at 45 degrees and angles with A at 90 degrees. Within each of those groupings, you can choose to add 45 or 90 degree intervals by marking the appropriate **45** or **90** degree check box.
    - b. *To create a custom angle*, from the right pane, tap **Custom Angle** then type a specific **A** and **B** angle, and tap **Add Angle**. Tap **Next** to add the custom angle into the **Included Angles** list on the right pane.
    - c. To delete an angle from the **Included Angles** list, tap the trash can button.
    - d. When you're done defining the angles for the probe, tap **Next** to proceed to the **Completed** step.
  10. For **Completed**, tap **Done** to close the **Probe Builder** screen. If the probe is one of the three probes assigned to ports on the probe changer, you can tap the **Calibrate** button that appears to start the probe calibration process right away.

**Note:** You must calibrate the probe before you can use it. For information on probe calibration, see "[About Probe Calibration](#)".

## How to Choose a Specific Angle on a Probe Command

**Objective:** To choose a specific angle for a probe command so you can cause your tip to point in different directions.

**Concepts:** [About Probes, Tips, and Angles](#)

**Prerequisites:** You must have a probe with an articulated head that supports rotating to different angles, such as an indexable probe. You must [define the probe](#) in the software and the available angles you want to use. You must have [created a part file](#) and have [inserted a probe tip command](#) for the probe into the file's Commands list.

### Method 1:

1. From the **Edit** screen for the part file, from the Commands list, tap on an existing probe tip command to edit.
2. From the probe tip command, tap **Select Angle** .
3. From the list of available **A** and **B** angles, tap on the one you want to use.
4. Tap **Accept**  to use the new angle in your tip command. The name of the probe tip command changes to reflect the new angle.

### Method 2:

1. From the bottom toolbar, tap **Angle**  to insert a new probe tip command based off the current probe tip command into the Commands list but with the interface open to choose a new tip on a multi-tip probe.
2. From the list of available **A** and **B** angles, tap on the one you want to use.
3. Tap **Accept**  to use the new angle in your tip command. The name of the probe tip command changes to reflect the new angle.

**Note:** Whenever you change a probe command (angles or probes), the probe status icon at the top of the screen changes to have a circle drawn around it. This indicates that what you have mounted on the CMM no longer matches the active probe command in the part file. To change the probe on the CMM so that it matches the current probe command, you need to tap that probe status icon. For more information on the probe status icon, see "[About the Status Icons](#)".

## How to Delete a Probe Definition

**Objective:** To delete a probe file.

**Concepts:** [About Probes, Tips, and Angles](#), [The Tools Screen](#)

**Prerequisites:** [Start PC-DMIS Touch](#) and select the mounted probe.

**Follow this procedure:**

1. From the **Tools** tile on the **Home** screen, tap **Tools**  to access the **Tools** screen.
2. From the **Tools** screen, under **Probes**, you can see a list of available probes.
3. Swipe up and down on the screen until you locate the probe you want to delete.
4. Next to the probe, tap the **Delete** button  to delete that probe.

## Reference Topics

### The Probe Builder Screen

You should access this screen when you want to define a new custom probe, edit an existing probe definition, or define an indexable probe's angles. To access this screen, from the **Tools** tile, tap **Tools**.



Then from the **Tools** screen, tap **Probe Builder**.



The screen contains these items:

**Header** - This initially shows "Edit Probe" until you select a probe to edit or define a new probe. Then the software appends the probe name.

**Left Pane** - The left pane shows the required steps you need to take to create or edit your probe. The highlighted step shows the current action you must take.

**Probe Preview window** - The area between the left and right panes shows a preview window of the probe as you build or edit it. Initially this contains nothing until you begin the creation or edit process. You can use the sliders to the left and bottom of the window to three dimensionally rotate the probe.

**Probe Description** - Next to the Probe Preview window, you can see the **Probe description** area. This contains the probe component selection interface. It contains a drop-down list and a list box:

- The drop-down list - You can use the drop-down list to define the current component.
- The list box - The box directly beneath the drop-down list shows the various hardware components that define the probe. When you create a new probe, it initially shows **No probe defined**. Once you start defining the probe components, it then shows any empty connection points you still need to define (listed as **Empty Connection #1** and so forth). You can select on a component to highlight it. Once highlighted, you can remove or modify that component.

**Right Pane** - For the different steps, the right pane contains these settings you can use to further define your probe:

- **New Probe** - Creates a new probe definition.
-  **Add** - Adds the selected probe component into the **Probe description** list box. This opens the **Probe description** drop-down list for you to select the component. This button activates once you highlight an empty component (shown as **Empty Connection #1**, **Empty Connection #2**, and so forth).
-  **Edit** - Edits the selected probe component from the list box with a new component that you select from the **Probe description** drop-down list, and removes any components beneath it. This button activates once you highlight a component.
-  **Remove** - Removes the selected probe component and any components beneath it. This button activates once you highlight a component.
- **New Probe Name** - Shows the probe's name. You can tap on the box to edit the probe name. By default, a new probe has a name of "Probe1", "Probe2", and so on.
- **Define the probe's angles** - Provides tools to add predefined angles or custom angles that you want your probe to use. This only appears if you use an indexable probe that supports angles.

For **45** degree A Angle increments

- **45** - Adds all the 45 degree angles in B for all the 45 degree angles in A.
- **90** - Adds all the 90 degree angles in B for all 45 degree angles in A.

For **90** degree A Angle increments -

- **45** - Adds all the 45 degree angles in B for all the 90 degree angles in A.
- **90** - Adds all the 90 degree angles in B for all 90 degree angles in A.
- **Custom Angle** - Shows an A and B box where you can type a specific custom angle.
- **Included Angles** - Shows all the angles you want to use in the probe definition. You can remove specific angles with the trash can button.
- **Calibrate** - You can tap this button to start the probe calibration process. This button only appears if the probe you selected to edit is one of the three probes assigned to ports in the probe changer.

**Related Topics:** [About Probes, Tips, and Angles](#), [How to Define or Edit a Probe](#)

## Installed Probes

The TIGO CMM comes with many probes in its initial kit. PC-DMIS Touch contains pre-configured probe files for each physical probe. You can then tell the software to use specific probes when measuring the part. This topic lists the default installation location and the available probe files.

### Default Installation Location

These are installed to this directory, where <version> is your current version of the software:

C:\Users\Public\Public Documents\WAI\PC-DMIS\2014.1

### Available Probe Files

- 1 - 1\_5x20 X-.prb
- 1 - 1\_5x20 X+.prb

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1 - 3x30 X-.prb  
1 - 3x30 X+.prb  
1 - 5x50 X-.prb  
1 - 5x50 X+.prb  
1\_5x20.prb  
2 - 3x30 X.prb  
2 - 3x30 Y.prb  
3 - 3x30 X.prb  
3 - 3x30 Y.prb  
3x30.prb  
3x30, 1\_5x20 X-.prb  
3x30, 1\_5x20 X+.prb  
3x30, 3x30 X-.prb  
3x30, 3x30 X+.prb  
3x30, 5x50 X-.prb  
3x30, 5x50 X+.prb  
4 - 3x30.prb  
5 - 3x30.prb  
5x20.prb  
5x50.prb  
5x100.prb

**Related Topics:** [About Probe Calibration](#), [About Probes, Tips, and Angles](#), [About the Probe Changer](#)

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## Using a Probe Changer

### About Probe Changers

**Terminology Note:** Probe changers are also sometimes called a "tool changer" or a "tip changer" or a "probe rack". This documentation always refers to it as a "probe changer".

A probe changer is a frame or rack that holds probes of different makes and styles with styli of different lengths and diameters. There may be times when you need to use a different probe tip for a specific situation. For example, you may want a probe that uses a longer stylus for measuring features that a shorter stylus cannot reach.

Your CMM comes with the LSPX1C probe changer. It initially has three ports to hold different hardware, but you can expand it to hold more than three. For information on how to do this, consult the documentation that comes with the probe changer.



### LSPX1C Probe Changer

Inside PC-DMIS Touch, you must define and calibrate your probe changer and then define the hardware attached into the ports before the software can interact with the probe changer to pick up or drop off any hardware.

- Probe changer *setup* tells the software what type of probe changer you have, the number of ports, the mount point location, and the docking speed to use during pickup or drop-off operations. For information on setting up the probe changer, see "[How to Define a Probe Changer](#)".
- Probe changer *calibration* tells the software where the probe changer and each probe changer port is located in 3-D space. For information on calibrating the probe changer, see "[How to Calibrate a Probe Changer](#)".
- Probe changer port *assignments*, tells the software what type of hardware is attached in each port. For information on port assignments, see "[How to Assign Probe Changer Ports](#)".

Once your probe changer is properly defined and calibrated, you can have the CMM's ram automatically load any of the probes stored in the ports by adding a command to load a specific probe tip when editing a part file. For more information, see "[How to Load a Probe from the Probe Changer](#)".

**Related Topics:** [About Probes, Tips, and Angles](#), [The Probe Changer Setup Screen](#), [The Probe Changer Calibration Screen](#), [The Probe Assignment Screen](#)

## How To

### How to Define a Probe Changer

**Objective:** To tell the software the type of probe changer to use, the number of ports it has and to define any other related settings.

**Concepts:** [About Probe Changers](#), [The Probe Changer Setup Screen](#)

**Prerequisites:** Start the software.

#### Follow this procedure:

1. From the **Home** screen, on the **Tools** tile, tap **Setup**  to open the **Probe Changer Setup** screen.
2. From the left pane, follow the instructions for each step. From the right pane, make all your selections. Depending on your choices, the software may automatically advance to the next step.
3. For **Type**, from the right pane, select the probe changer type and then tap **Next**.
4. For **Ports**, from the right pane, tap either **+** or **-** to increase or decrease the number of ports and then tap **Next**.
5. For **Mount Point**, you must have a probe mounted on the CMM. You need to define a safe location and angle above and in front of the probe changer that the ram moves to and rotates to before dropping off and picking up probes from the ports. You can do this in either of these ways:
  - From the right pane, manually tap the **X**, **Y**, and **Z** boxes and use the numeric keypad that appears to define a value for each.
  - Or use the jog box to manually position the physical probe in the desired location, and then on the jog box press **DONE**  or from the right pane, tap **Read Machine**.
  - From the right pane, for **Probe Angle**, manually tap the **A** and **B** boxes and define the safe rotation angle that the probe rotates to prior to interacting with the probe changer.
6. Tap **Next** to proceed.
7. For **Settings**, if you don't want to use the default value, from the right pane, tap the **Docking Speed** value to define a new docking speed. This

docking speed is a percentage of the maximum machine speed. The default is 5%.

8. Tap **Next** to proceed.
9. For **Completed**, tap **Done** to finish. If the changes you made require a calibration, the **Calibrate Probe Changer** screen automatically opens. For information on probe changer calibration, see "[How to Calibrate a Probe Changer](#)".

## How to Calibrate a Probe Changer

**Objective:** To locate where the probe changer and its ports are located in 3-D space.

**Concepts:** [About Probe Changers](#)

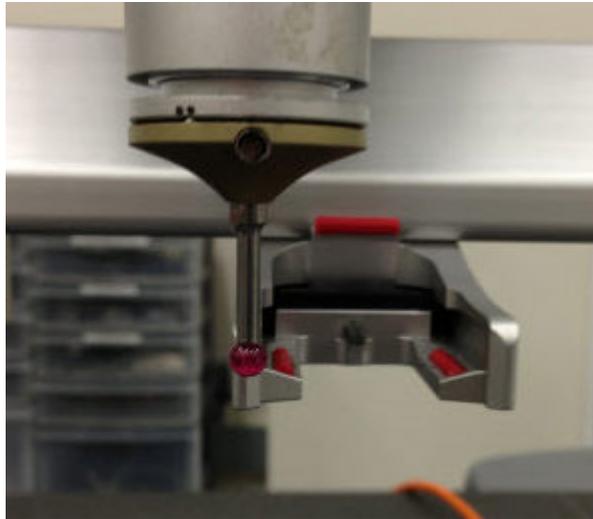
**Prerequisites:** Start the software. You must have a [defined](#) probe changer.

### Follow this procedure:

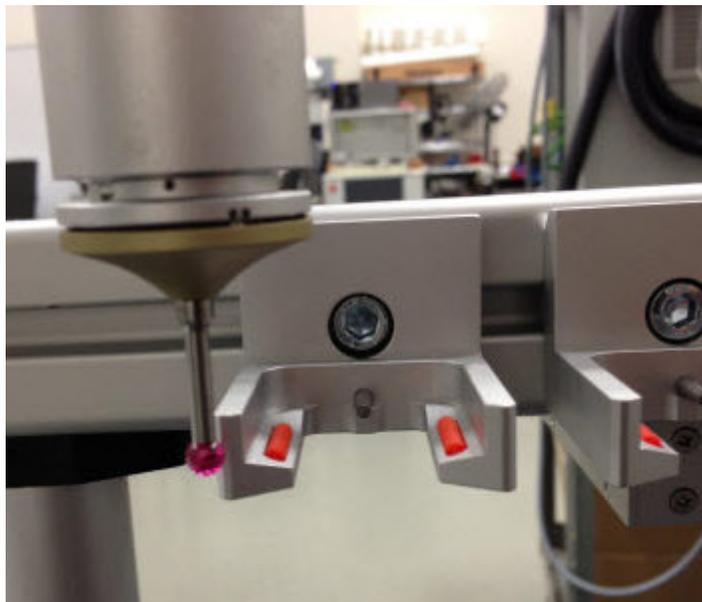
1. From the **Home** screen, on the **Tools** tile, tap **Calibrate** (for probe changers)  to open the **Probe Changer Calibration** screen. This button is only enabled if a probe changer is already defined. For information on defining a probe changer, see "[How to Define a Probe Changer](#)".
2. For **Probe**, from the right pane, use the filter controls to locate and select the probe to use in the probe changer calibration. The probe doesn't need to be calibrated. Tap **Next** to proceed with the next step. Note that specific probe selection is only needed if you don't have a probe already mounted on the CMM, or if you want to use a different probe. If you want to use the probe already attached on the CMM, just wait a few seconds to automatically progress to the next step.
3. For **Mount Probe**, manually attach the selected physical probe to the CMM and tap **Next** to proceed. This step is only needed if the selected probe isn't already mounted on the CMM. If the software detects the probe is already mounted, it skips this step.
4. For **Remove Probes**, manually remove any hardware from all the slots on the physical probe changer and tap **Next** to proceed. This is only needed if you're doing a re-calibration of the probe changer. If this is your first time defining and calibrating a probe changer, the software skips this step.

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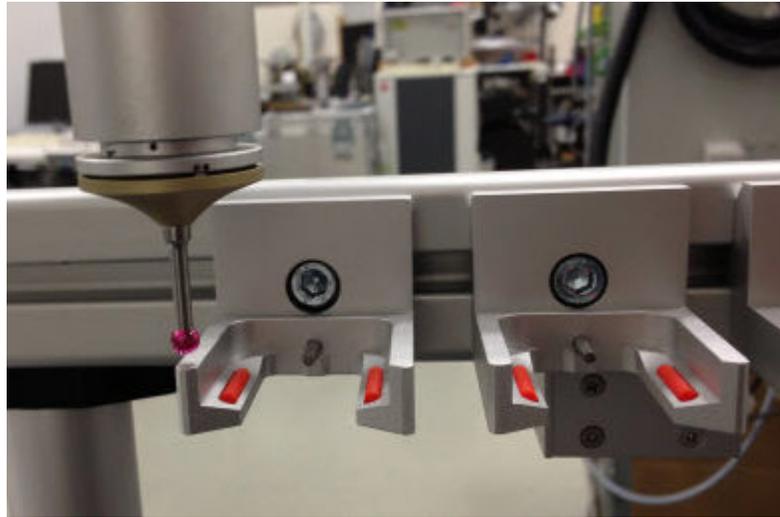
5. For **Calibration**, follow the on-screen animations and instructions. Use the jog box and manually move the machine and take points on the probe changer hardware as directed:
  - a. Open all the lids on the ports by pushing them all the way back.
  - b. Ensure that hardware is removed from each port.
  - c. Probe a manual point on the front surface of the left side of the last port. Be sure to avoid the tapered corner.



- d. Probe a manual point on the front surface of the left side of the first port. Again, be sure to avoid the tapered corner.



- d. Probe a manual point on the top surface on the left side of each port attached to the probe changer. This manual point should be taken near the front but far enough back to avoid any tapered corner.



- e. The computer now drives the machine to finish the calibration.
6. Tap **Done** to close the **Calibrate Probe Changer** screen.

## How to Assign Probe Changer Ports

**Objective:** To assign specific probes to the available probe changer ports.

**Concepts:** [The Probe Assignment Screen](#), [About Probe Changers](#)

**Prerequisites:** [Start PC-DMIS Touch](#). You must have a [defined](#) probe changer.

### Follow this procedure:

1. Access the **Probe Assignment** screen. From the **Home** screen, under the **Tools** tile, tap **Assign** . This button is only enabled if a probe changer is already defined. For information on defining a probe changer, see "[How to Define a Probe Changer](#)".
2. From the left pane, follow the instructions for each step. From the right pane, make all your selections.

3. For **Dock Probe**, follow the prompt that appears. Move the probe to safe location, and then to tap **OK** to have the CMM move to the probe changer and place the probe currently attached on the CMM into an empty port on the probe changer. You can also tap **Cancel** to close the prompt and from the right pane, tap **Start** later to dock the probe. The software skips this step if it determines you don't need to dock the current mounted probe.
4. For **Assign**, do the following:
  - a. From the right pane, filter the list of probes as needed until you find the one you want to add. For information on the filters, see "[The Probe Assignment Screen](#)".
  - b. *To add a probe to the probe changer*, from the right pane, drag and drop the probe onto one of the three port positions in the center of the screen. The probe is removed from the list and assigned to that port.
  - c. *To rearrange probes in existing ports*, drag and drop the probes already assigned among the available ports.
  - d. *To remove a probe from the probe changer*, drag and drop it somewhere outside of the port diagram altogether. The probe appears back in the list.

Note that the left-most port is always the master port; so whatever probe is assigned to that port is always calibrated first.

4. Repeat the above step until you have added the probes you need.
5. Tap **Next** to proceed to the next step.
6. For **Completed**, physically attach the probes into the available ports in the probe changer.
7. Tap **Next** to proceed to the next step.
8. For **Calibrate Probes**, tap **Done** to calibrate the probes. The screen changes to the **Probe Calibration** screen. For information on probe calibration, see "[About Probe Calibration](#)".

## How to Load a Probe from the Probe Changer

**Objective:** To load the probe stored in a specific port of the probe changer.

**Concepts:** [About Probe Changers](#)

**Prerequisites:** From the **Home** screen, open a file.

When you select a new probe tip within a part file, that probe is loaded from the probe changer. For information, see the "[How to Select a Probe Tip](#)" topic.

## Reference Topics

### The Tools Tile

The **Tools** tile shows the most recently accessed or commonly used hardware tasks. Tap on one of the buttons on this tile to open the selected tool or perform the action. The **Tools** tile is disabled until you define the mounted probe.



**Tools** - Opens the **Tools** screen to view all the hardware tools at once. Using this screen, you can then select the desired tool. To access the

**Tools** screen, from the top toolbar, you can also tap **Tools** . For information on this screen, see "[The Tools Screen](#)".



**Calibrate (Probe)** - Calibrates a probe and defines a calibration tool if needed. For information, see "[How to Calibrate a Probe](#)".



**Assign** - Assigns probe hardware to each probe changer port. For information, see "[How to Assign Probe Changer Ports](#)". This button is only enabled if a probe changer is already defined.



**Calibrate (Probe Changer)** - Calibrates the probe changer. For information, see "[How to Calibrate a Probe Changer](#)". This button is only enabled if a probe changer is already defined.



**Setup** - Defines the probe changer and its settings (type, number of ports, mount point location, and docking speed). For information, see "[How to Define a Probe Changer](#)".

**Related Topics:** [About Probe Changers](#)

### The Probe Changer Setup Screen

You should access this screen when you want to define a new probe changer or modify settings for the current probe changer. To access this screen, from the **Tools** tile, tap **Setup**.



The screen contains these items:

- **Header** - This shows "Probe changer setup" unless a probe changer type is already defined, then it shows "<type> setup" where <type> is the probe changer type.
- **Left Pane** - The left pane shows the required steps you need to take to configure the probe changer. The highlighted step shows the current action you must take.
- **Port Diagrams** - The area between the left and right panes shows diagrams of the different slots located on the probe changer. These can increase or decrease when you change the number of ports. They also provide a relative location of where to define your mount point.
- **Right Pane** - For the different steps, the right pane contains these settings used to define your probe changer:
  - **Select your type** - Defines the probe changer type. Currently, these include the LSPX1C, LSPX1H, and LSPX1SF.
  - **Enter the number of ports** - Defines the total number of ports. You can tap + to increase the number by one or - to decrease the number by one.
  - **Define mount point** - The **X**, **Y**, and **Z** boxes define the mount point location. The mount point is safe location in front of and above the probe changer that the CMM's ram moves to before and after performing a drop-off or pickup operation.
  - **Probe angle** - The **A** and **B** boxes define the A and B angle positions that the probe rotates to before it moves to drop off or pick up anything from the probe changer. This only appears if the probe mounted on the CMM supports rotating to different angles.
  - **Docking Speed** - The speed used when performing a drop-off or pickup operation. This is a percentage of the top machine speed. The default is 5%.

**Related Topics:** [About Probe Changers](#), [How to Define a Probe Changer](#)

## The Probe Changer Calibration Screen

You should access this screen after you've defined a probe changer and you're ready to calibrate it. To access this screen, from the **Tools** tile, tap **Calibrate** (for probe changers).



The screen contains these items:

- **Header** - This shows "Calibrate Probe Changer 1".
- **Left Pane** - The left pane shows the required steps you need to take to calibrate the probe changer. The highlighted step shows the current action you must take. If the software considers a step unnecessary, it puts a check mark next to that step and progresses to the next step.
- **Animation Area** - The area between the left and right panes shows port diagrams and probe animations that can help guide you through completing the current step.
- **Right pane** - Provides instructions for each step.
  - For the **Select Probe** step, you need to select a probe do the calibration. It provides a list of probes to select from. For information on sorting and filtering the probe list, see "[The Probe Assignment Screen](#)".
  - For the **Calibrate** step, it also provides a **Start** button to begin the actual calibration. This changes to a **Stop** button once calibration starts if you want to stop the calibration before it finishes.

**Related Topics:** [About Probe Changers](#), [How to Define a Probe Changer](#)

## The Probe Assignment Screen

You should access this screen when you want to configure what probes should be in the ports attached to the probe changer. To access this screen, from the **Tools** tile, tap **Assign**.



The screen contains these items:

- **Header** - This shows "Probe Assignment".
- **Left Pane** - The left pane shows the required steps you need to take to configure the probe changer. The highlighted step shows the current action you must take.
- **Port Diagrams** - The area between the left and right panes shows diagrams of the different ports located on the probe changer and the probe assigned to each port.
- **Right pane** - If a probe is connected to the CMM, the right pane initially shows a **Start** button. You need to use this to dock the probe with the probe changer before proceeding. Once the probe docks with the probe changer, or if no probe is attached to the CMM in the first place, this pane then shows all the available probe types. You can scroll through the list to find a probe to assign to a port. From this pane, you can drag and drop probes onto these diagrams to assign probes to ports.

You can use the filters at the top of the pane to show a subset of probes based on one or more of the following:

-  **Filter by Number of Tips** - Filters the available probes by the selected number of tips.
-  **Filter by Tip Direction** - Filters the available probes by the selected tip direction.
-  **Filter by Tip Size** - Filters the available probes by the selected tip diameter.

Note that for any of these filters, a grayed out item in this list means you are not filtering by that item. You can choose any item in the list to re-filter the list of probes by the new item.

The information icon next to each probe  displays detailed information about that probe.

**Related Topics:** [About the Probe Changer](#), [How to Load a Probe from the Probe Changer](#), [How to Assign Probe Changer Ports](#)

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## Working with Part Files

### About Files

A PC-DMIS Touch "File" stores the commands needed to align and measure a part. In order to distinguish between this type of file and other file types on your computer system, this type of file is often termed a "part file" in this documentation. In Learn mode, you can add feature measurement commands and dimensions into your file to teach the software what to measure.

**Important:** Commands are stored in a list called the Commands list. Certain screens and operations allow you to select existing features from your Commands list or from the Graphics window. The insertion point needs to be beneath the commands you want to select. To change your insertion point location, simply swipe up or down in the Commands list to the desired position, and tap on an existing command. All commands above the selected command are then "visible" and available for selection in other screens and operations.

After learning a part, when you're ready to measure it, you can run the file from the **Run** screen. In Run mode, the software guides you through where to take the needed probe hits for each feature measurement command.

To create a new file, from the **Home** screen, on the **New** tile, tap one of the items.

To edit an existing file from the **Home** screen, on the **Files** tile, tap **More Files** to access the **Files** screen.

Then, from the **Files** screen, find the part file to edit, and tap its name.

**Related Topics:** [How to Create a New Part File](#), [How to Edit a Part File](#), [How to Edit a Part File's Attributes](#), [How to Run a Part File](#), [The New Tile](#), [The Files Tile](#), [The Files Screen](#), [The Edit Screen \(for Files\)](#), [About Run Mode and Running Your Part File](#), [The Run Screen](#)

### How To

#### How to Create a New Part File

**Objective:** To create a new part file so that you can start adding measurements and other commands to it.

**Concepts:** [About Files](#), [The New Tile](#), [The New File Screen](#), [The Files Tile](#), [The Files Screen](#)

**Prerequisites:** [Install and start the software.](#)

**Follow this procedure:**

1. Create a new part file by choosing one of the following options:
  - a. Create a temporary part file from the system defaults. You can save this file later:
    - From the **Home** screen, under the **New** tile, tap one of the side buttons: **Dimension**, **Construction**, **Alignment**, or **Measurement**.
  - b. Create your own part file, choosing its name, measurement units, and so on:
    - To access this screen, from the **Home** screen, tap under the **New** tile, tap **File** or tap the probe image.
    - On the **New File** screen that appears, do the following:
      - *Required:* In **Name**, type a name for your part.
      - *Optional:* Tap on any of these items to provide these values:

**Revision Number**

**Serial Number**

**Part Temperature Compensation**

**Measurement Units**

- *Optional:* Associate an image with the part in one of two ways:
  - To open an existing image file, tap  and find the picture you want.
  - To take a new picture and associate it with the part file, tap . This shows the current camera's feed in the container window. To take a snapshot of the image in the view, tap . To cycle between available cameras on your

computer, tap the  icon in the top left of the camera window.

2. Tap **Done** to create the part file.
3. If temperature compensation is enabled for the file, a prompt with a 20 second countdown timer appears, asking you if you've already applied the part temperature sensor. If you haven't, apply it now. If you have, tap **OK**. Once the timer expires, or if you tap **OK**, the current process continues and compensates for temperature. If you tap **Cancel**, the process continues, but it does not compensate for temperature. For information on temperature compensation, see "[About Temperature Compensation](#)".
4. From the [Edit screen](#), you can [learn your part](#).
5. To save your changes, from the bottom toolbar, tap **Save** . If you are using a temporary part file (default.prg), a dialog box appears when you tap **Save** or when you try to leave the bottom level screens so you can define the file name. If the software detects an existing part file of the same name, it asks if you want to overwrite the existing file.

## How to Edit a Part File

**Objective:** To delete commands from a part file; or to modify tolerance or nominal values in a learned feature, constructed feature, or dimension command.

**Concepts:** [The New File Screen](#), [About Files](#), [The Edit Screen \(for Files\)](#), [The Edit Screen \(for Editing Commands\)](#), [The Commands List](#)

**Prerequisites:** You must have an existing part file containing commands, and you must have privileges to make changes to that part file.

### Follow this procedure:

1. On the **Home** screen, from the **Files** tile, tap **More Files**.
3. From the **Files** screen, find the file to edit, and tap its name to open that file.
4. If temperature compensation is enabled for the file, a prompt with a 20 second countdown timer appears, asking you if you've already applied the part temperature sensor. If you haven't, apply it now. If you have, tap **OK**.

Once the timer expires, or if you tap **OK**, the current process continues and compensates for temperature. If you tap **Cancel**, the process continues, but it does not compensate for temperature. For information on temperature compensation, see "[About Temperature Compensation](#)".

5. From the **Edit** screen, you can see the Commands list, a list of commands in an expandable and collapsible tree view. For information about the Commands list, see "[The Commands List](#)".
6. Modify the file by doing one of the following:

#### *Edit a command*

Tap the command to select it from the Commands list. You need to expand a collapsed parent plane (PLN) command if you want to access a child feature stored under that plane. For information on parent planes, see "[About Faces](#)".

- a. Once you've selected the desired command, tap **Edit** .
- b. From the **Edit** screen, modify any nominals or tolerances as needed. For more information, see "[About Nominals, Tolerances, and Axes](#)".
- c. Show or hide axes as desired by tapping on them. Colored axes appear in the report. Empty axes are hidden from the report.
- d. Tap **Next** to access the **Advanced** items. This step is optional.

On the left pane, you can choose how the feature is measured. If you select **Touch**, the feature is measured with distinct points, triggered by the probe contacting with the part. If you select **Scan**, the probe remains in contact with the part and follows an internal strategy to scan and measure the feature. In the right pane, you can modify the advanced properties for the feature, such as the number of **Points**, the **Depth**, the ClearanceCube face used by the feature, and so on. For information on feature-specific properties you can edit, see "[About Advanced Properties](#)". Tap **Remeasure** if you want to remeasure the feature using the information from the right-pane.

- e. When finished, tap **Done**, or press DONE  on your jog box. The edited feature is recalculated.

### *Delete a command*

In the Commands list, tap on a command to select it, and then tap

**Delete** . You must delete child commands before deleting their parent commands. Once the child commands are deleted, the parent command's delete button appears in the list of commands.

### *Insert a new command*

In the Commands list, tap on an existing command to select it, and then create the new command as usual. The command is inserted just below the currently selected command. If the inserted command is a 2D feature (point, line, circle, round slot, or square slot), it is projected to the parent [Face](#) command immediately above it in the Commands list.

If you insert a new feature where other features already exist, and the attached probe does not match the probe command used for the surrounding features, the software asks if you want to load the probe used by those features. If you tap **Yes**, it inserts the new feature and uses their probe command. If you tap **No**, it adds in the new feature and adds in a new probe command for the currently attached probe. If you tap **Cancel**, the new feature is canceled.

7. Tap **Save**  to save the modified part file.

## **How to Edit a Part File's Properties**

**Objective:** To edit a part file's properties, such as part name, revision number, serial number, temperature compensation, and so on.

**Concepts:** [About Files](#), [The Properties Screen](#), [The New File Screen](#)

**Prerequisites:** An existing part file that was saved using your current version of PC-DMIS Touch.

**Follow this procedure:**

1. On the **Home** screen, from the **Files** tile, tap **More Files**.
3. From the **Files** screen, find the file to edit, and tap **Edit**  to open the file's **Properties** screen. This screen contains the same information as the [New File screen](#). You can change the following properties:
  - The picture used
  - The **Revision Number**
  - The **Serial Number**
  - The part material for **Part Temperature Compensation**
4. Make and save your changes. The process is similar to filling out the information when creating a new part file. For information, see "[How to Create a New Part File](#)".
  - To save the file with the same name, tap **Save** .
  - To save the file with a new name, tap **Save As** .

## Reference Topics

### The New Tile

The **New** tile provides you with options to create a new part file. By default, if you tap one of the following, the part file is a temporary file named default.prg until you save it and give it a new name:

- **Dimension** creates a new part file and displays the [New Dimension screen](#) in order to create dimensions from new features you measure.
- **Alignment** creates a new part file and displays the [New Alignment screen](#) in order to create a new alignment from new features you measure.
- **Measurement** creates a new part file and displays the [New Measurement screen](#) in order to measure new features.
- **Construction** creates a new part file and displays the [New Construction screen](#) in order to create constructed features from new features you measure.

If you want to define the part file's name, units, and other items, tap **File**.

The software immediately displays the [New File screen](#).

## The New File Screen

You should access this screen when you want to create a file with a specific file name, revision number, serial number, measurement units, a picture, and so forth.

To access this screen, from the **Home** screen, under the **New** heading, tap **File**; or under the **New** heading, tap the probe image.

The screen contains these items:

- **File Name** - *Required*: This defines the name of the underlying file. When you create a name, the software automatically displays an associated part image if one exists (see "Image" below). You can use the same name as an existing file. If you do, red text warns you that a file of the same already exists. If you re-create the part file, a message appears confirming that you want to overwrite the existing one.
- **Revision Number** - *Optional*: If you type a value, this number is stored in the part file and becomes part of the file name when the file is created.
- **Serial Number** - *Optional*: If you type a value, this number is stored in the part file and becomes part of the file name when the file is created.
- **Part Temperature Compensation** - *Optional*: If this list is enabled, choose your part's material. This makes it so measurements made on your part are compensated for temperature based on the selected material.
- **Measurement Units** - This determines the type of measurement units to use in the part file (inches or millimeters). It defaults to millimeters.
- **Image** - *Optional*: The large picture area to the left of the **Name** box initially shows a picture of a large camera. This indicates that no image is currently associated with the file. You can associate a picture by taking a new picture or by using a picture already on your computer. Once the part file is created, the image is copied to the same directory as the file. The image name is the file's same name with a .png filename extension.
-  - Opens a dialog box so you can to find a picture from your computer's file system to associate with the file.
-  - Activates the tablet's built-in camera. The following icons appear in a subsequent window once the camera is activated:

-  - Takes a picture of what's in your camera's view port and associates it with current part file.
-  - Switches to another available camera on your computer. This is disabled if you only have one camera.
-  - Deactivates your camera.
-  - Deletes the current picture.

**Related Topics:** [The New Tile](#), [The Files Tile](#), [The Files Screen](#), [How to Create a New Part File](#)

## The Files Tile

The **Files** tile shows the four most recently accessed part files. Tapping one of the buttons on this tile opens the selected part file in the [Edit screen](#). From there, you can edit the part file or swipe horizontally to get to the [Run screen](#) to run the file. If you want to work with a different part file that isn't visible on the **Files** tile, tap **More Files** to access the [Files screen](#) to see all available part files.

From this screen, you can see all of the part files and choose the desired one to view, edit, or delete.

## The Files Screen

The **Files** screen shows all of the available part files from the root of the directory that holds your part files. By default, this is located in C:\Users\Public\Documents\WAI\PC-DMIS\<>version>\ where <version> is the version of PC-DMIS installed. From this screen, you can filter the existing files and then edit or delete the file as needed. You should use this screen if you need to edit a file that isn't one of the recent ones listed in the **Files** tile or if you need to delete a part file.

To access this screen, from the **Home** screen, on the **Files** tile, tap **More Files**.

## Search Feature

Initially, this screen lists all of the files. They are sorted in reverse date order. The software automatically filters the files to show only the files with names that contain the set of characters typed in the search box. If you don't type anything, the search functionality shows all of the part files.

## File Info

On the left of the list, you can view each file's associated picture. Tap a picture to show an enlarged view.

You can also see each file's name above the date the file was last accessed.

## Operations

- To open the file for editing, tap its name to open the [Edit screen](#).
- To edit a file's properties, tap **Edit**  to open the [Properties screen](#).
- To delete a file, tap **Delete** .

**Related Topics:** [About Run Mode and Running Your Part File](#), [About Learn Mode and Measuring Features](#), [How to Learn a Part](#), [How to Edit a Part File](#), [How to Run a Part File](#)

## The Properties Screen

You should access this screen when you need to edit your part file's attributes.

To access this screen, from the **Home** screen, under the **Files** heading, tap **More Files**, and then from the **Files** screen, find the file you want to modify, and tap **Edit** .

The screen contains these properties. You can change any of them except for **Measurement Units**:

- The picture used
- The **Revision Number**
- The **Serial Number**
- The part material for **Part Temperature Compensation**

It also contains these buttons at the bottom:

-  **Save** - Saves your changes to the same file name.
-  **Save As** - Saves your changes to a new file name.

For additional information on these properties, and the picture and camera controls, see the "[The New File Screen](#)" topic.

**Related Topics:** [The Files Tile](#), [The Files Screen](#), [The New File Screen](#), [How to Edit a Part File's Properties](#)

## The Edit Screen (for Files)

The **Edit** screen contains a Graphics window, a Commands list, and a bottom toolbar. You can use this screen to view or edit existing part files and populate them with alignment, face, feature, dimension, and other commands needed to [learn a part](#).

If you start taking hits while on this screen, the software immediately displays the **New Measurement** screen.

### Graphics Window

This window displays all of the learned features graphically. See "[The Graphics Window](#)".

### Commands List

To the right of the Graphics window, a special list, called the Commands list in this documentation, contains all of the commands in your part file. For more information see "[The Commands List](#)".

### Bottom Toolbar

This toolbar provides these buttons to add new content into the part file or save changes made:

-  [Dimension](#) - Defines a new dimension. It opens the **New Dimension** screen.
-  [Construction](#) - Constructs a feature from existing features. It opens the **New Construction** screen.
-  [Alignment](#) - Defines a new alignment to establish a new coordinate system. It opens the **New Alignment** screen.

-  **Measurement** - Learns a new feature. It opens the **New Measurement** screen.
-  **Axis** - Defines a new surface or plane within which new features will reside. It opens a toolbar of six faces that are based on either the current alignment's axis planes or the machine's coordinate system.
-  **DCC** - Inserts a **DCC Mode** command into the part file that you can set to **On** (to have the computer control machine motion) or **Off** (to have you manually control the machine's motion).
-  **Probe** - Inserts a new probe tip command into the Commands list but with the interface open to choose one of the probes assigned to your probe changer. You can tap on one of the probes and then modify the command to choose a specific tip or angle. The software uses the specified probe command to know what hardware and rotation angle to use to measure any features that follow it in the Commands list.
-  **Angle** - Inserts a copy of the current probe tip command into the Commands list but with the interface open to choose a new angle. This button appears if the active probe supports multiple angles (such as an indexable probe).
-  **Tip** - Inserts a copy of the current probe tip command into the Commands list but with the interface open to choose a new tip on a multi-tip probe. This button appears if the active probe supports multiple tips attached to the same probe body (such as a star-cluster probe).
-  **Save** - Saves the part file with the same name.
-  **Save As** - Saves the part file with a new name.

**Related Topics:** [How to Learn a Part](#), [How to Define a Dimension](#), [How to Construct a Feature from Other Features](#), [How to Learn Features to Measure](#), [How to Edit a Part File](#), [The Graphics Window](#), [About Faces](#), [About DCC Mode and Manual Mode](#), [About Probes, Tips, and Angles](#)

## The Edit Screen (for Editing Commands)

Use this **Edit** screen in Learn mode to edit the nominal values, and the upper and lower tolerance values of a learned feature, constructed feature, or dimension.

This screen appears whenever you select a feature and then tap  from the [Edit screen](#) on the bottom level of screens.

The screen contains these items:

- **Left Pane** - The left pane shows the current feature or dimension you're editing.
- **Graphics window** - The Graphics window between the left and right panes shows a drawing of all of the learned features in your part file in relation to each other. It shows the current selected feature highlighted in yellow. For more information on the Graphics window, see "[The Graphics Window](#)".
- **Right Pane** - The right pane initially shows a grid of actual, nominal, deviation, and tolerance values. You can use this pane to change the nominals and tolerances, and to show or hide axes from the report. For more information, see "[About Nominals, Tolerances, and Axes](#)". If you are editing a feature, and you tap **Next** again; or if in the left pane, you tap on **Advanced**, the contents of the right pane change to show the advanced properties for the feature.

On the left pane, you can choose how the feature is measured. If you select **Touch**, the feature is measured with distinct points, triggered by the probe contacting with the part. If you select **Scan**, the probe remains in contact with the part and follows an internal strategy to scan and measure the feature. In the right pane, you can modify the advanced properties for the feature, such as the number of **Points**, the **Depth**, the ClearanceCube face used by the feature, and so on. For information on feature-specific properties you can edit, see "[About Advanced Properties](#)". Tap **Remeasure** if you want to remeasure the feature using the information from the right-pane.

**Related Topics:** [The Edit Screen \(for Files\)](#), [How to Edit a Part File](#)

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## Measuring Features

### About Learn Mode and Measuring Features

Before you can measure anything with the software, the software needs to know what it must measure. In Learn mode the software can learn what features you want it to measure. You teach it the features to learn by taking hits on different

features when in Learn mode. Learned features are stored in files that you can then run (or measure) in [Run mode](#).

The software is placed in learn mode from the **Home** screen whenever you create a new part file or edit an existing part file. This happens whenever you tap any of these items under the **New** tile on the **Home** screen:

- **File** - Tap this to create a new part file with a part name, an image, and other details. See "[How to Learn a Part](#)".
- **Dimension** - Tap this to create a new default part file and then specify dimensions for features. You can define the nominal and tolerance values. See "[How to Define a Dimension](#)".
- **Alignment** - Tap this to create a new default part file and then tell the software the location and position of your part in 3D space. See "[How to Create a Part Alignment](#)".
- **Measurement** - Tap this to create a new default part file and then tell the software what features you want it to measure in a new part file. See "[How to Learn Features to Measure](#)".
- **Construction** - Tap this to create a new default part file and then construct new features that don't exist physically on your part from other learned features. See "[How to Construct a Feature from Other Features](#)".

**Related Topics:** [How to Learn a Part](#), [How to Learn Features to Measure](#)

## How To

### How to Learn a Part

**Objective:** To align the part with the machine and teach the software what features on your part to measure and dimension.

**Concepts:** [About Learn Mode and Measuring Features](#), [About Files](#), [The New File Screen](#)

**Prerequisites:** [Create new part file](#). Ensure a probe is attached and [calibrated](#).

#### Follow this procedure:

1. Once you have a new part file created, you need to first create an alignment. From the bottom toolbar, tap **New Alignments** and then follow the procedure in "[How to Create a Part Alignment](#)".



2. Once you have an alignment, from the bottom toolbar you need to define features to measure. Tap **New Measurement** and then follow procedure in "[How to Learn Features to Measure](#)".



3. If desired, tap **New Construction** to create new features from other features by following the procedure in "[How to Construct a Feature from Other Features](#)".



4. Once features are in your part program, you need to check to see if they are within acceptable tolerances. Tap **New Dimensions** and then follow the procedure in "[How to Define a Dimension](#)" to add the desired dimensions into the part file.



5. The file is now ready to run. See "[About Run Mode and Running Your Part File](#)".

6. To save your changes, from the bottom toolbar, tap **Save** . If you are using a temporary part file (default.prg), a dialog box appears when you tap **Save** or when you try to leave the bottom level screens. Use the dialog box to define the file name.

## How to Learn Features to Measure

**Objective:** To teach the software what features Run mode should measure.

**Concepts:** [About Learn Mode and Measuring Features](#), [The New Measurement Screen](#). Features taught to the software are stored in the part file and then measured later when you run the file. To teach the software, you need to manually measure the feature yourself once with your probe.

**Prerequisites:** You must have a probe attached to the machine or to the [probe changer](#). You must successfully [calibrate](#) any probe tips you are going to use. You must have defined an [alignment](#).

**Follow this procedure:**

1. Ensure that the insertion point is where you want it in the Commands list by tapping on the desired command. For more information on the insertion point, see "[The Commands List](#)" topic.
2. Access the **New Measurement** screen. You can do this by doing one of the following:
  - Create a brand new part file and open the screen. From the **Home** screen, on the **New** tile, tap **Measurement**:



- Edit an existing part file and then from the bottom toolbar, tap **New Measurement**:



- Or just start probing points with your measurement device.
3. The left pane shows the steps you need to take to measure a feature. The highlighted heading shows the current needed action.
  4. If temperature compensation is enabled for the file, and at least two hours has passed since the last temperature compensation, a prompt with a 20 second countdown timer appears asking you if you've already applied the part temperature sensor. If you haven't, apply it now. Once the timer expires, or if you tap **OK**, the current process continues and compensates for temperature. If you tap **Cancel**, the process continues, but it does not compensate for temperature. For information on temperature compensation, see "[About Temperature Compensation](#)".
  5. If you want to measure a two-dimensional feature (point, line, circle, or slot), you must first define a plane, called a "Face", that the feature is projected onto. Until a Face is defined, those two-dimensional features remain unavailable for selection. See "[Faces](#)" for more information.

You can add a Face by doing one of the following:

- In the **Measurement** screen, after you probe points for a plane feature, in the right pane, tap **New Face**.



The Face is inserted at the end of the Commands list.

- If you have defined an alignment, from the bottom toolbar, tap **Axis**:



Then tap one of the six axis plane buttons (**X+**, **X-**, **Y+**, **Y-**, **Z+**, or **Z-**) generated from the alignment:



The Face is inserted after the alignment command. Its name is based on the alignment and the chosen axis plane.

6. **Measure.** Begin probing points on your part to measure your features. If you need help on where to probe the points, before probing any points, from the right pane, tap on the feature type to see an animated diagram. For each point you take, move the probe to the feature, use the jog box and carefully touch the feature with the probe tip. The touch trigger records a point. As you start measuring, and the location and number of probed points are interpreted by the software, the right pane highlights possible features in either white or yellow:

- White highlighted features represent the possible valid measurements for the current probed points.
- The yellow highlighted feature is the current guessed feature. Next to the guessed feature, the software shows the current number of probed points.

In addition, on the left pane under **Measure**, the software shows a preview of the current measurement. The preview shows the feature's XYZ centroid, as well as data for its position, size, and form. You can then determine to proceed with the measurement or not.

7. Continue probing points for all of the points needed for the desired feature. Each point is stored temporarily in a point buffer until the feature is

- created. You can remove all the point from the buffer by tapping **Previous** or remove the last point by pressing ERASE HIT  on your jog box.
8. If you want to override the guessed feature, in the right pane, tap a valid feature type.
  9. Once the points are taken, the screen changes to show the feature's name.
  10. If you need to change the feature name, in the **Feature Name** box, type a new value. After the feature is selected, tap **Next** or **Done**.
    - Tap **Done** to automatically create the feature, accepting the default nominals and tolerances. A feature created in this way does not appear in the report. This helps speed up the creation of commands when you don't need to display a feature in the report.
    - Tap **Next** to proceed to edit the feature's nominals and tolerances.
  11. After the feature is selected, tap **Next** to proceed. You can also press DONE  on your jog box to proceed.
  12. If you tapped **Next**, the software temporarily creates the feature and draws it in the Graphics window. An icon of the feature and the feature's name appear in the left pane under **Select/Measure**.
  13. **Edit**. This step is optional. You can modify tolerances and nominals and choose what axes to send to the report. For more information, see "[Nominals, Tolerances, and Axes](#)".
    - In each box you want to edit, tap the nominal or tolerance box, and then use the on-screen keypad to type the desired value. The available axes and boxes vary, depending on the type of feature just learned. For more information on editing this information, see "[About Nominals, Tolerances, and Axes](#)". Continue doing this until you make the changes you need.
    - Tap the colored axis row that has a small dot to its left to show or hide that axis. This way you can control what information ends up in the report. You cannot hide axes without dots. Colored axes appear in the report. Empty axes do not.
  14. Tap **Next** to proceed You can also press DONE  on your jog box to proceed. For information on the advanced information, see "[About Advanced Properties](#)".
  15. **Advanced**. This step is optional.

On the left pane, you can choose how the feature is measured. If you select **Touch**, the feature is measured with distinct points, triggered by the probe contacting with the part. If you select **Scan**, the probe remains in contact with the part and follows an internal strategy to scan and measure the feature. In the right pane, you can modify the advanced properties for the feature, such as the number of **Points**, the **Depth**, the ClearanceCube face used by the feature, and so on. For information on feature-specific properties you can edit, see "[About Advanced Properties](#)". Tap **Remeasure** if you want to remeasure the feature using the information from the right-pane.

16. Until you reach the very last step, if you ever need to redo any of the steps in the left pane, tap the desired step in the left pane, or tap **Previous** until the desired step becomes highlighted.

**Note:** Going back to an earlier step in the left pane resets all of the steps below it.

17. Once everything is done, if you don't like the measurement, you can either use **Previous** to go back to an earlier step and make corrections, or you can cancel it entirely and close the screen by doing one of the following:

- Tap **Cancel**, or tap the white X button in the upper-right corner of the screen.

Or

- Press the ERASE HIT button  on your jog box.

18. If you're satisfied with the measurement, tap **Done** to accept the measurement and close the screen. Or, press DONE  on your jog box.

The new feature is then visible in the part file.

## Reference Topics

### The New Measurement Screen

Use the **New Measurement** screen in learn mode to create and store learned features for future measurement.

This screen automatically appears whenever you take points with your measurement device. It also appears if you do one of the following:

- Create a brand new part file and open the screen by tapping the **Measurement** button from the **New** tile in the **Home** screen:

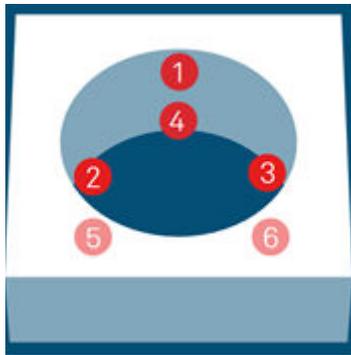


- Edit an existing part file, and then from the bottom toolbar, tap the **New Measurement** button:



The screen contains these items:

- **Left Pane** - The left pane shows the steps you need to take to measure a feature. The highlighted heading shows the current action you must take. In addition, under **Measure**, the software shows a preview of the current measurement. You can then determine to proceed with the measurement or not. The preview shows the feature's XYZ centroid, as well as data for its position, size, and form. For information on the symbols that appear, see "[About Nominals, Tolerances, and Axes](#)" and where it discusses "[Available Axes](#)".
- **Probing Diagram and Graphics window** - The area between the left and right panes changes based on the current state of a measurement. When you first come to this screen, if you select a feature from the right pane, an *animated diagram* appears showing where you need to probe points on the feature in order to properly learn that feature. Once you start probing points, this area changes to the graphics window. It shows any points taken during the active measurement as well as all of the learned features in your part file in relation to each other. When the measurement finishes, the Graphics window shows a drawing of the newly learned feature. For more information, see "[The Graphics Window](#)".



Sample diagram showing how to probe points on a cylinder

- **Right Pane** - The right pane initially displays a dimmed view of all of the possible features supported by the software. Once you start measuring, and the location and number of probed points are interpreted by the software, this pane highlights the possible features in white and the current guessed feature in yellow. Next to the guessed feature, the software shows the current number of probed points. At any time, you can override the guessed feature by tapping a white-highlighted feature. When you finish or cancel the measurement, the **New Measurement** screen closes.

If you tap **Next** before tapping **Done**, or if you tap the dot to the left of the feature name, the right pane changes first to show a grid of actual, nominal, deviation, and tolerance values. You can use this pane to change the nominals and tolerances, and to show or hide axes from the report. For more information, see "[About Nominals, Tolerances, and Axes](#)". If you tap **Next** again, or if in the left pane you tap **Advanced**, the contents of the right pane change to show the advanced properties for the feature.

On the left pane, you can choose how the feature is measured. If you select **Touch**, the feature is measured with distinct points, triggered by the probe contacting with the part. If you select **Scan**, the probe remains in contact with the part and follows an internal strategy to scan and measure the feature. In the right pane, you can modify the advanced properties for the feature, such as the number of **Points**, the **Depth**, the ClearanceCube face used by the feature, and so on. For information on feature-specific properties you can edit, see "[About Advanced Properties](#)". Tap **Remeasure** if you want to remeasure the feature using the information from the right-pane.

When measuring or constructing a feature, if both the **Next** and **Done** buttons are available, and you choose **Done**, the feature is created with the default nominal and tolerance values, but it is excluded from the report so you can create the command more quickly. If you want to later display it in the report, you can edit the feature and tap the dot to the left of the feature name. For information on editing a feature in a file, see "[How to Edit a Part File](#)".

Points, lines, circles, and slots first require a defined Face feature. Until a Face is defined, these two-dimensional features remain unavailable for selection. See "[About Faces](#)" for more information.

**Related Topics:** [About Learn Mode and Measuring Features](#), [How to Learn a Part](#), [How to Learn Features to Measure](#)

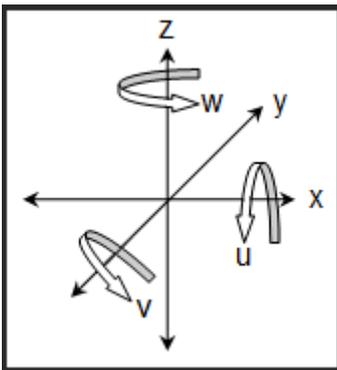
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## Creating an Alignment

### About Alignments

Before you can properly start measuring your part, you must first define an alignment for your part. An Alignment command tells the software where you want your coordinate system's origin position ( $x=0$ ,  $y=0$ ,  $z=0$ ) to be located in relation to your part. This locates the part's position and orientation in 3D space.

A part without any alignment has six degrees of freedom: three degrees of rotation and three degrees of translation.



This diagram shows the six degrees of freedom in 3D space ( $x,y,z,u,v$ , and  $w$ ).

To establish an alignment, you must measure three valid features termed "alignment features" and then use those features to define the X, Y, and Z origin.

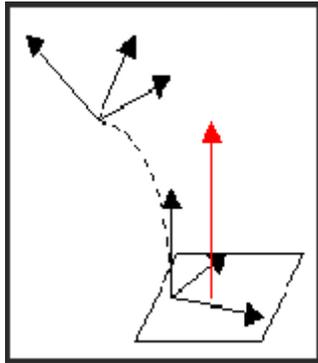
These alignment features "align" each major axis to a selected feature, thereby constraining the six degrees of freedom, and fixing the part in 3D space.

The Table of Valid Feature Types located in the "[Aligning the Part](#)" topic shows the list of valid features.

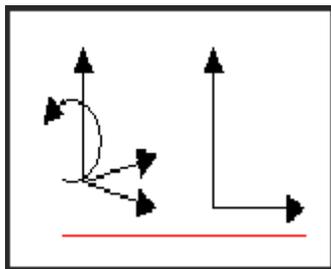
### A Sample Alignment

In this sample alignment, the coordinate system's origin is established from a plane feature and two line features.

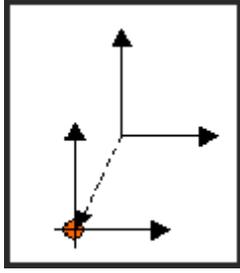
**Plane** - An axis is associated with the normal vector of the plane. This determines the orientation of that axis, and the part is considered leveled to the selected plane. This constrains two degrees of rotation:



**Line** - An axis is rotated to this line about the leveled axis. The rotated axis matches the vector of the selected feature. This constrains one degree of rotation about the leveled axis:



**Line** - The X and Y origin location is moved to the point where the two lines intersect and then is projected in the Z axis onto the plane feature. This constrains three degrees of translation (origin) in the X, Y, and Z axes:



**Related Topics:** [The New Alignment Screen](#), [How to Create a Part Alignment](#), [About Faces](#)

## How To

### How to Create a Part Alignment

**Objective:** To tell the software where the part is located in 3D space and establish the coordinate system in order to learn the part.

**Concepts:** [About Alignments](#), [The New Alignment Screen](#)

**Prerequisites:** You need a part file. Also, you need three valid features to level, orient, and translate the axes; however, you can measure these during this procedure, if needed.

#### Follow this procedure:

1. Ensure that the insertion point is where you want it in the Commands list by tapping on the desired command. For more information on the insertion point, see "[The Commands List](#)" topic.
2. Access the **New Alignment** screen. You can do this by doing one of the following:
  - Create a brand new part file to open the screen. From the **Home** screen, on the **New** tile, tap **Alignment**:



- Edit an existing part file and then from the bottom toolbar, tap **New Alignment**:



From the left pane, follow the instructions for each step. From the right pane, make all your selections. After you make the proper selection, the software automatically advances to the next step.

3. In the left pane, under the highlighted heading, as you proceed, the software provides instructions and lists the type of feature to select or measure.
4. For **Type**, from the right pane, tap an *alignment type*:
  - **Basic** - This creates a basic alignment from three features. Valid features are listed under in the Basic alignment procedure below.
  - **Rotate** - This creates an alignment with a rotation offset.
  - **Translate** - This creates an alignment with a translation offset.
  - **Best Fit** - This creates an alignment best fit between all the selected features.

*For the **Basic** alignment...*

- a. For **Feature 1**, select or measure the first feature.
  - Consult the "[Basic Alignment - Valid Features](#)" table below for valid features.
  - From the filtered list on the right, or from the Graphics window, tap a valid feature to select it.
  - If the desired feature does not exist, measure it now. To measure, probe points on the part with your measurement device. For more information, see "[Learning Features to Measure](#)". Once the feature is measured, it becomes selected.
- b. When you tap or measure a valid feature, the software accepts that feature, and displays it in the left pane. You are prompted to select or measure the next feature. You can also press **Done** on your jog box to proceed. The left pane shows the selected feature.
- c. For **Feature 2** and **Feature 3**, continue selecting or measuring valid features until you have the three alignment features selected. Once the needed features are selected or measured, the screen changes to show the alignment's name.

- d. For **Complete**, if you need to change the alignment's name, tap on the name, and type a new value.
- e. Tap either **Next** or **Done**.
  - Tap **Done** to create the alignment without changing any advanced options.
  - Tap **Next** to set **Advanced** properties in the right pane. For information on the advanced alignment properties, see "[About Advanced Properties](#)".

**Basic Alignment - Valid Features**

Feature 1	Feature 2	Feature 3
Plane	Line	Line
Plane	Line	Circle
Plane	Circle	Circle
Plane	Plane	Line
Plane	Plane	Plane
Cylinder	Plane	Plane
Cylinder	Line	Point

*For the **Rotate** alignment...*

- a. <Instructions provided here once implemented>

*For the **Offset** alignment...*

- a. <Instructions provided here once implemented>

*For a **Best Fit** alignment...*

- a. <Instructions provided here once implemented>

- 5. At any time you can correct or cancel the alignment as needed.

- To correct it, use **Previous** to go back to an earlier step, or from the left pane, tap on the desired step.
- To cancel it entirely and close the screen, do one of the following:
  - Tap **Cancel**.

Or

- Press the ERASE HIT button  on your jog box.

6. If you're satisfied with the alignment, tap **Done** to accept the alignment and close the screen. Or, press DONE  on your jog box.

**Note:** Since most users want to measure items on their alignment's level plane, a Face from the plane used to level the part is automatically inserted into the Commands list immediately after any newly created alignment. For example, if you use the **Xplus** plane as your alignment's level axis plane, and A1 is your alignment name, an axis Face of A1\_XPLUS is inserted immediately after the created alignment.

The alignment command fixes your part in 3D space and establishes the coordinate system. Any feature command added after the Alignment command uses this new coordinate system.

## Reference Topics

### The New Alignment Screen

You should access this screen when you want to create a new Alignment command in your part file. This screen appears if you do one of the following:

- Create a brand new part file and open the screen by tapping the **Alignment** button from the **New** tile in the **Home** screen:



- Edit an existing part file and then tap the **New Alignment** button from the bottom toolbar:



The screen contains these items:

- **Header** - Initially this shows "New Alignment". This text changes to display the type of alignment you choose to create.
- **Left Pane** - The left pane shows the required steps you need to take to define the alignment. The highlighted step shows the current action you must take. The feature icons displayed beneath each step indicate a possible feature type you can measure or select for that step. Once the feature is selected or measured, it appears beneath the completed step and a check mark appears next to that step.
- **Graphics window** - The Graphics window between the left and right panes shows a drawing of all of the learned features in your part file in relation to each other. You can use this to select your alignment features, or as a reference when selecting alignment features from the right pane. When you select a feature from the right pane, the software highlights the selected feature and centers and orients the feature in the Graphics view. See "[The Graphics Window](#)" for more information.
- **XYZ Information** - Beneath the Graphics window, you can see **X, Y, Z** fields. This is a digital readout of the probe's current position.
- **Right Pane** - The right pane initially shows the alignment type to select. Once you select an alignment type, if valid features exist in your part file for the current alignment step, the right pane then displays a filtered list of available alignment features that you can choose from.

Once you choose the features for the alignment, and the software highlights **Complete** on the left, you can tap **Next**, to have the right pane show the alignment's advanced properties so you can make adjustments to the alignment. These include the alignment name, what axes to use in the leveling and rotation, and any values to use for manual offsets. For more information, see "[About Advanced Properties](#)".

**Related Topics:** [How to Create a Part Alignment](#), [About Alignments](#)

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## Working with Faces

### About Faces

Two-dimensional features do not have an orientation. They must be projected onto a specific plane in order to have meaning. This type of plane is called a "Face". Faces can be defined from either a measured surface (a plane feature) or from one of the six axis planes (X+, X-, Y+, Y-, Z+, or Z-).

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- If the insertion point is beneath an alignment command, the Face is based off that alignment's axis. In this case, the Face's name is based on the alignment and the chosen axis plane, and displays as "A1\_ZPLUS", "A1\_YMINUS" and so on.
- If you do not have an alignment, or the insertion point is above an alignment command, the Face is based off of the machine's coordinate system axis. In this case, the Face's name just has the axis label, such as "ZPLUS", "XPLUS" and so on.

Until a Face is defined, these two-dimensional features remain unavailable for selection:

- Points
- Lines
- Circles
- Round slots
- Square slots

Once a Face is defined, you can then measure two-dimensional features. They are added to the part file under the current Face feature and are projected to that plane.

A plane feature can either be its own Face command or under the current Face command. You may want it under the current Face in situations such as viewing an angle or some other distance in a Face view where you don't want it to be its own Face.

**Note:** Three-dimensional features (cylinders, cones, and spheres) do not need to be projected onto a Face.

**Related Topics:** [How to Add a New Face](#), [The Edit Screen \(for Files\)](#), [The New Measurement Screen](#), [How to Learn Features to Measure](#), [About Alignments](#), [The Graphics Window](#)

## How To

### How to Add a New Face Command

**Objective:** Add a new Face command so that two-dimensional features are projected onto that surface.

**Concepts:** [About Faces](#)

**Prerequisites:** You must [measure a plane feature](#).

**Follow this procedure:**

Add a Face command by doing one of the following:

- In the **Measurement** screen, after you take the hits for a plane feature, in the right pane, tap **New Face**:



The Face is inserted at the end of the Commands list.

- From the **Edit** screen, on the bottom toolbar, tap **Axis**:



Then tap one of the six axis plane buttons (**X+**, **X-**, **Y+**, **Y-**, **Z+**, or **Z-**) generated from the alignment:



- If the insertion point is beneath an alignment command, the Face is based off that alignment's axis. In this case, the Face's name is based on the alignment and the chosen axis plane.
- If you do not have an alignment, or the insertion point is above an alignment command, the Face is based off of the machine's coordinate system axis. In this case, the Face's name just has the axis label.

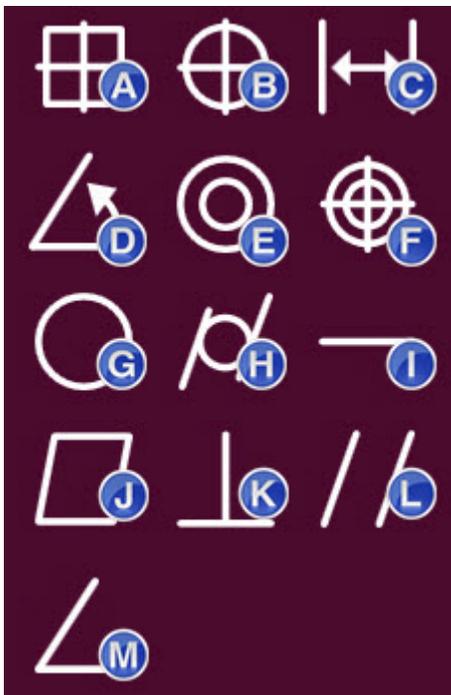
**Note:** Since most users want to measure items on their alignment's level plane, a Face from the plane used to level the part is automatically inserted into the Commands list immediately after any newly created alignment. For example, if you use the **Xplus** plane as your alignment's level axis plane, and A1 is your alignment name, an axis Face of A1\_XPLUS is inserted immediately after the created alignment.

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## Defining Dimensions

### About Dimensions

Once features have been measured or constructed, you can then generate an inspection report. The first step in preparing a report is to calculate dimensions according to specific requirements. Dimensions tell you if a measurement is within acceptable tolerances. A dimension displays the measured value, the deviation from the nominal value, tolerances, and whether the feature is within tolerances. Dimensions are also the main component needed in your part file before you can generate a useful report.



- A. [Location](#)
- B. [Position](#)
- C. [Distance](#) (2D or 3D)
- D. [Angle](#) (2D or 3D)
- E. [Concentricity](#)
- F. [Coaxiality](#)
- G. [Circularity](#)
- H. [Cylindricity](#)
- I. [Straightness](#)
- J. [Flatness](#)
- K. [Perpendicularity](#)
- L. [Parallelism](#)
- M. [Angularity](#)

**Related Topics:** [How to Define a Dimension](#), [The New Dimension Screen](#)

## How To

### How to Define a Dimension

**Objective:** To add a dimension command into your part file so that measurement results can later appear in a report.

**Concepts:** [About Dimensions](#), [The New Dimension Screen](#)

**Prerequisites:** You must have some [learned features](#) in your part file and a [defined alignment](#).

**Follow this procedure:**

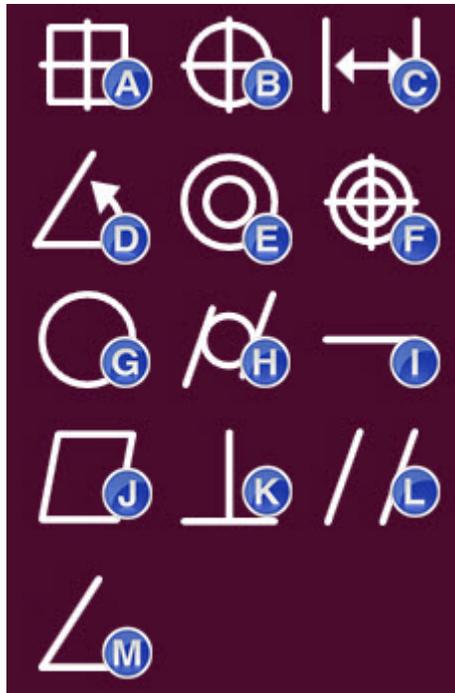
1. Ensure that the insertion point is where you want it in the Commands list by tapping on the desired command. For more information on the insertion point, see "[The Commands List](#)" topic.
2. Access the **New Dimension** screen. You can do this by doing one of the following:
  - Create a brand new part file and open the screen. From the **Home** screen, on the **New** tile, tap **Dimension**:



- Edit an existing part file and then from the bottom toolbar, tap **New Dimension**:



3. From the left pane, follow the instructions for each step. From the right pane, make all your selections. After you make the proper selection, the software automatically advances to the next step.
4. For **Type**, from the right pane, tap the type of dimension to create:



- A. Location
- B. Position
- C. Distance
- D. Angle
- E. Concentricity
- F. Coaxiality
- G. Circularity
- H. Cylindricity
- I. Straightness
- J. Flatness
- K. Perpendicularity
- L. Parallelism
- M. Angularity

5. For **Feature**, **Feature 1**, or **Feature 2**, on the left pane, the software indicates the types of features to select or measure.
  - a. From the filtered list on the right or from the Graphics window, tap a valid feature to select it.
  - b. If the needed feature does not exist, measure it now. To measure, probe hits on the part with your measurement device. See "[How to Learn Features to Measure](#)" for more information. Once the feature is measured, it becomes selected.

6. Continue selecting input features or options until you have everything you need for your selected dimension.
  - a. For the Angle dimension, you can also choose to calculate the angle between the first feature and an axis line by choosing **X Axis**, **Y Axis**, or **Z Axis**.
  - b. For the [Position](#) dimension, the software automatically selects the recommended axes to use in the position calculation and displays them on the left pane. If you want to change the axes used, from the left pane, tap **Axes**. Then from the right pane, select or deselect the desired axes. The software shows the selected axes in the report.
7. Once the needed features are selected or measured, the screen changes to show the following:
  - The dimension's name
  - The dimension's measured data
  - Nominal and tolerance information
  - Other options if any
8. For **Complete**, use the right pane to modify the information in the above step as needed:
  - a. In each box you want to edit, tap the nominal or tolerance box, and then use the on-screen keypad to type the desired value. The available axes and boxes vary, depending on the type of feature being dimensioned. Continue doing this until you make the changes you need. For more information on nominals and tolerances, see "[About Nominals, Tolerances, and Axes](#)".
  - b. The software evaluates the measured value, the allowed tolerances, and the nominal value. It displays the axes in different colors: it colors in-tolerance axes in **green** and out-of-tolerance axes in **red**.
  - c. Tap any colored axis row that has a small dot to its left to show or hide that axis. This way you can control what information ends up in the report. You cannot hide axes without dots. Colored axes appear in the report. Empty axes do not.
  - d. Modify other options as needed:

*For Distance*

Under **Solution**, choose the distance solution to use: **3D** (the 3D distance), **X** (the distance in X), **Y** (the distance in Y), or **Z** (the distance in Z).

If a Distance dimension involves a circle feature, under **Distance Options** choose what to do with the circle's radius in that measurement: add the radius, subtract the radius, or don't use the radius at all.

For more information and examples on the distance options, see "Distance".

### *For Angle*

Under **Angle Type**, determine how to calculate the angle between two features by choosing **2D** (input features are projected onto the current face) or **3D** (the true angle between the input features).

If you need to report the alternate or complementary angle, under **Angle Solutions**, tap the desired angle to report.

For more information and examples on the angle options, see "Angle".

9. At any time you can correct or cancel the dimension as needed:
  - To correct it, use **Previous** to go back to an earlier step, or from the left pane, tap on the desired step.
  - To cancel it entirely and close the screen, tap **Cancel**.

10. If you're satisfied with the dimension, tap **Done** to accept the dimension and close the screen. Or, press DONE  on your jog box.

11. If you want to accept the dimension but keep the screen open to create another dimension, tap  instead.

The dimension is then visible in the part file.

## Reference Topics

### The New Dimension Screen

You should access this screen when you want dimensional information for measured data to appear in the report. This screen appears if you do one of the following:

- Create a new part file and open the screen by tapping the **Dimension** button from the **New** tile in the **Home** screen:



- Edit an existing part file and then tap the **New Dimension** button from the bottom toolbar:



The screen contains these items:

- **Header** - Initially this shows "New dimension". After you define the input features for a dimension, this text changes to display the dimension's name.
- **Left Pane** - The left pane shows the required steps you need to take to create the dimension. The highlighted heading shows the current action you must take.
- **Graphics window** - The Graphics window between the left and right panes shows a drawing of all of the learned features in your part file in relation to each other. You can use this window to select the input features used in the dimension. When you select an item from the file list, the software highlights the selected feature in yellow and centers and orients the view. See "[The Graphics Window](#)" for more information.
- **XYZ Information** - Beneath the Graphics window, you can see **X, Y, Z** fields. This is a digital readout of the probe's current position.
- **Right Pane** - During each phase of the dimension process, the right pane changes to contain the dimension types, and input features you can select. After you select the input features, the right pane shows a grid of actual, nominal, deviation, and tolerance values. You can use this pane to change the nominals and tolerances, and to show or hide axes from the report. You can also use this pane to modify additional options available to

some dimensions. For more information, see "[Nominals, Tolerances, and Axes](#)".



The create another button, , creates the dimension but keep the screen open to create another dimension.

**Related Topics:** [How to Define a Dimension](#), [About Dimensions](#)

## Available Dimensions

### Location

This dimension shows the XYZ position of a feature from the coordinate system's origin, parallel to the feature's respective axis. The diameter, angle, and vector are also part of the calculation if they are available to the selected feature.

The Location dimension is only available for selection from the **New Dimension** screen if the selection point in the Commands list follows an Alignment command.

The default output for a Location dimension changes based on the feature type. Consult this table:

Feature	Output Format
Circle	XYZ D (Diameter based on the face)
Cone	XYZ A (Angle)
Cylinder	XYZ D (Diameter) L (Length based on the face)
Line	Output format is based on the axis perpendicular to the axis closest to and related to the face to which the line is projected
Plane	Output format is based on the axis most closely related to the plane

Point	XYZ T (for points on curved surfaces, this is the deviation along the approach vector)
Slot	XYZ D (Diameter) R (Radius) L (Length)
Sphere	XYZ D (Diameter)

Location dimensions are constructed automatically whenever a feature is measured.

### Position

Similar to [Location](#), the Position dimension also calculates the position of the feature to the X, Y, or Z origin parallel to its respective axis. However, it differs from Location because it is only used with features of size. Also, it defines not only a feature's location, but also its orientation.

The Position dimension is only available for selection from the **New Dimension** screen, if the selection point in the Commands list follows an Alignment command. The features in the alignment command become the datum reference features used for the dimension. Position also takes into consideration the tolerance at a certain material condition. The valid inputs for this dimension are points, planes, circles, cylinders, cones, spheres, and round and square slots.

A measured value of 0 for a Position dimension means the feature has the perfect location and position. The tolerance value defines the amount of deviation away from 0 that is allowed for the dimension to still be in tolerance. The tolerance value defines the tolerance zone.

Depending on the type of feature dimensioned, one or more material condition options become available. You can use a material condition to define the allowed tolerance for the selected material condition. These include:

**MMC** (Maximum Material Condition)

**LMC** (Least Material Condition)

**RFS** (Regardless of Feature Size)

If you choose a circle, cylinder, cone, sphere, round slot, or square slot, you will see all three material conditions listed above. If you choose a point or a plane, only RFS is available.

## **Distance**

The distance dimension shows the distance between two features.

The Distance dimension is a three dimensional distance if you select the **3D** option during the **Complete** step of the dimension creation procedure (For more information, see the "Solution" topic below).

The distance is a two-dimensional distance if you select the **X**, **Y**, or **Z** option during the **Complete** step. These axes appear enabled if your first input feature resides below an alignment command. A two-dimensional distance is projected onto the current workplane prior to the calculation. (For more information, see the "Solution" topic below).

## **Solution**

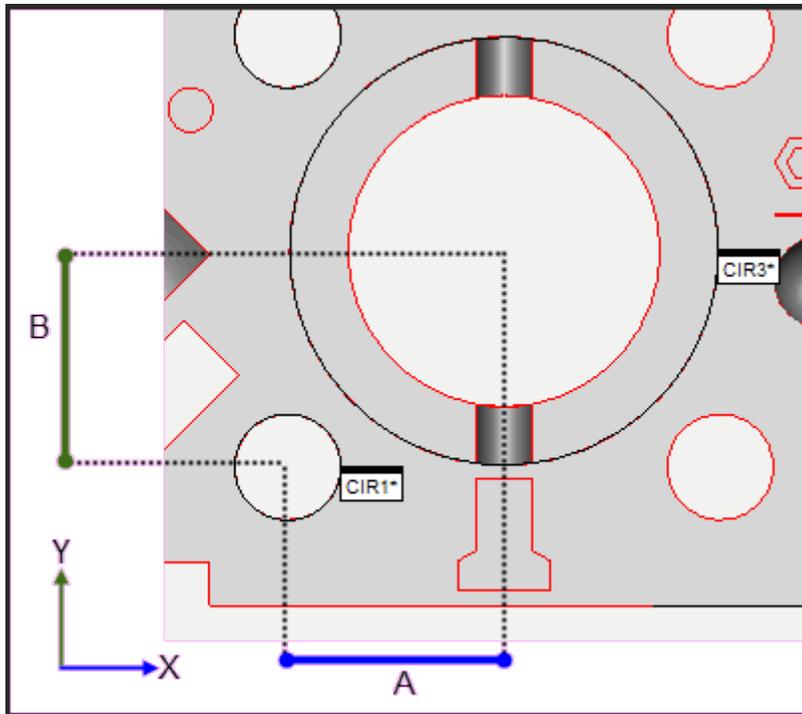
These determine what solution to use for a distance dimension. You can choose a 3D solution by choosing **3D**. Or you can choose a 2D solution by choosing one of the axes options (**X**, **Y**, or **Z**) and calculate a distance parallel to the chosen axis. Since 2D distance projects the features onto the current work plane, the work plane axis is not a valid selection and is automatically disabled.

**3D** - Calculates the distance between the two features three dimensionally. 3D does not do a projection to the work plane. If one of the input features is a line, centerline, or plane, Touch computes the 3D distance normal to that feature. If both features are lines, centerlines, or planes, the second feature is used for the datum. If both features are planes, the 3D distance is the perpendicular distance from the centroid of the first plane to the second plane; the second plane acts as the datum. If neither input feature is a line, centerline, or plane, Touch computes the shortest distance between the two features.

**X** - The distance in X. This is solved between the first selected feature and the second selected feature parallel to the X axis.

**Y** - The distance in Y. This is solved between the first selected feature and the second selected feature parallel to the Y axis.

**Z** - The distance in Z. This is solved between the first selected feature and the second selected feature parallel to the Z axis.

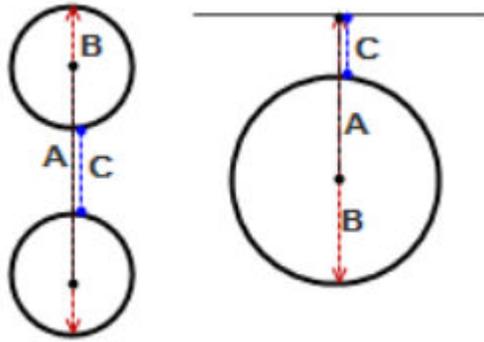


For example, in the above diagram, the distance labeled A is the distance in X (or parallel to X) between CIR1 and CIR3, and the distance labeled B is the distance in Y (or parallel to Y).

### Options

If your distance dimension involves a circle feature, you can decide how to handle the radius in the dimension (add, subtract, or don't use it). The following diagram shows the distance between two circles or between a circle and a line if the radius is added or subtracted:

## PC-DMIS TOUCH CMM

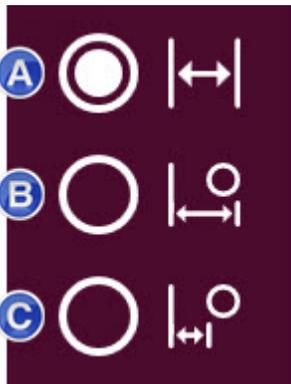


**A** - Normal calculated distance between features (black line). The radius is ignored. This is the default.

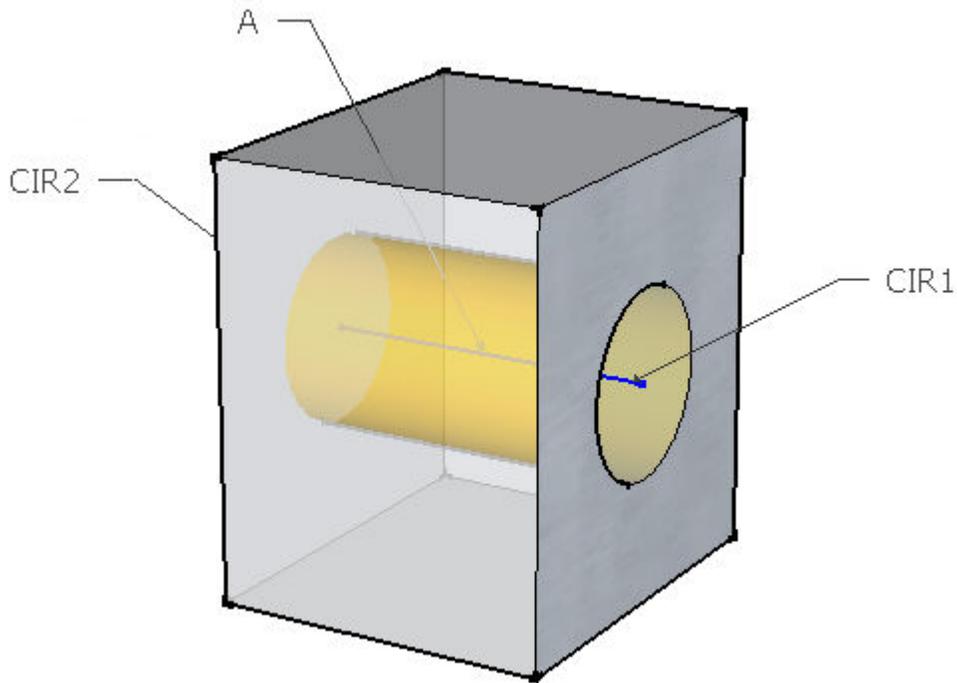
**B** - Add Radius (red line)

**C** - Subtract Radius (blue line)

Below are the corresponding options in the software interface. Use the A, B, and C key above.



### Example

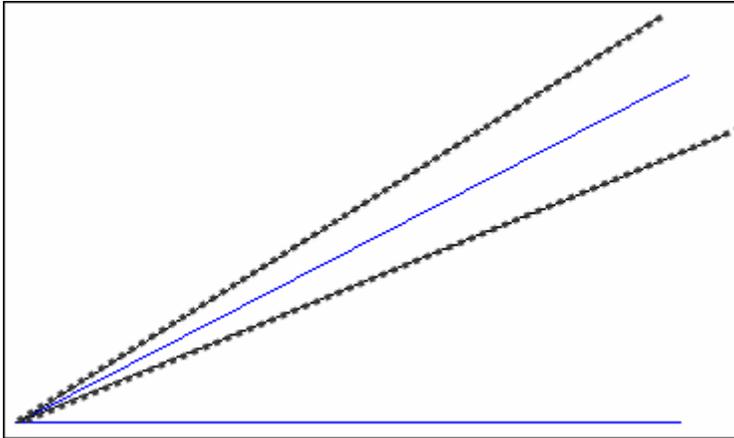


**A** - The reported dimension is the distance between the centers of the two circles (CIR1 and CIR2) at opposite ends of a cylinder.

### **Angle**

This dimension calculates the angle between two valid features (planes, lines, circles, cylinders, cones, round slots, and square slots) or between a single feature and an axis. It then checks if the angle is within tolerances.

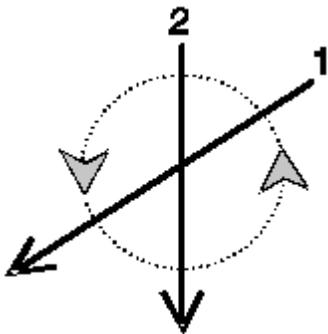
The tolerance band for this dimension is in radians or degrees and forms an angle.



The measured value is an angle (shown in blue). The tolerance band is another angle (shown in dotted black).

If the angle reported by the software is not in the correct quadrant (for example, you want 0.0, not 180.0), type the correct nominal angle in the **Nominal** box. PC-DMIS Touch automatically converts the quadrant to match the nominal angle.

The directions of the features' lines (or vectors) determine the angle that is then measured:



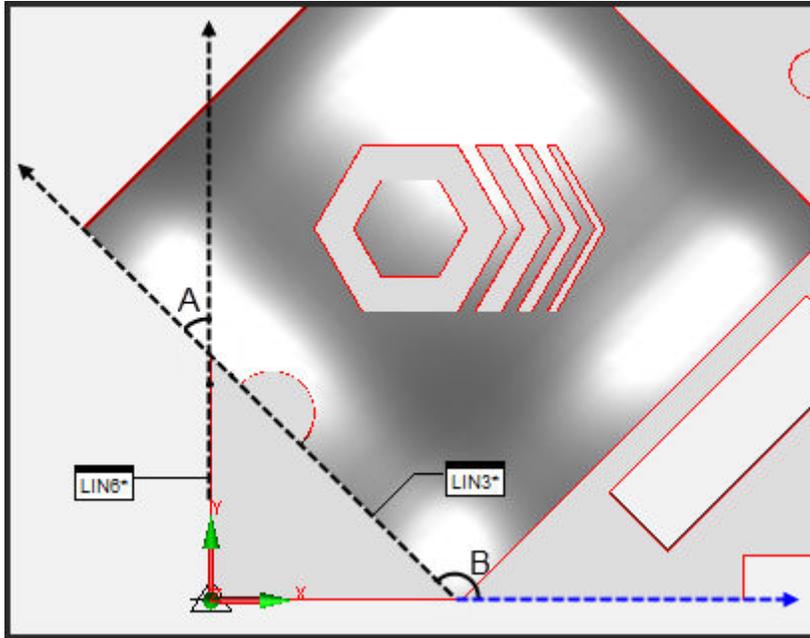
### Angle to an Axis

You can also calculate the dimension between a feature and an axis line. After you select your first feature, the right pane shows the **X Axis**, **Y Axis**, or **Z Axis** lines.

**X Axis** - The angle between the feature and the X axis.

**Y Axis** - The angle between the feature and the Y axis.

**Z Axis** - The angle between the feature and the Z axis.



For example, in the above diagram, an angle between LIN3 and LIN6 in the Z+ axis plane results in an angle of 45 degrees (A), while the angle between LIN3 and the X axis results in 135 degrees (B).

### Angle Type

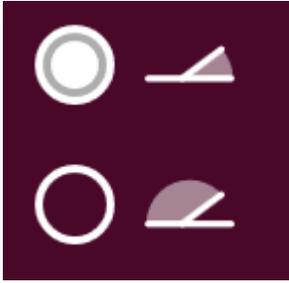
This determines how to calculate the angle between two features. Angle dimension can either be two-dimensional or three-dimensional. If your active face (the selected face or the face for a selected command) is an [axis plane](#), you can choose between either option. If the active face is a measured plane, only **3D** is available. If you select the same axis as the active face, only **3D** is available.

**2D** - This calculates the angle two dimensionally. This means the input feature or features are projected onto the current face before the angle is calculated.

**3D** - This is the true angle between the features. It calculates the angle in three dimensions. This does not do a projection to current face.

### Angle Solutions

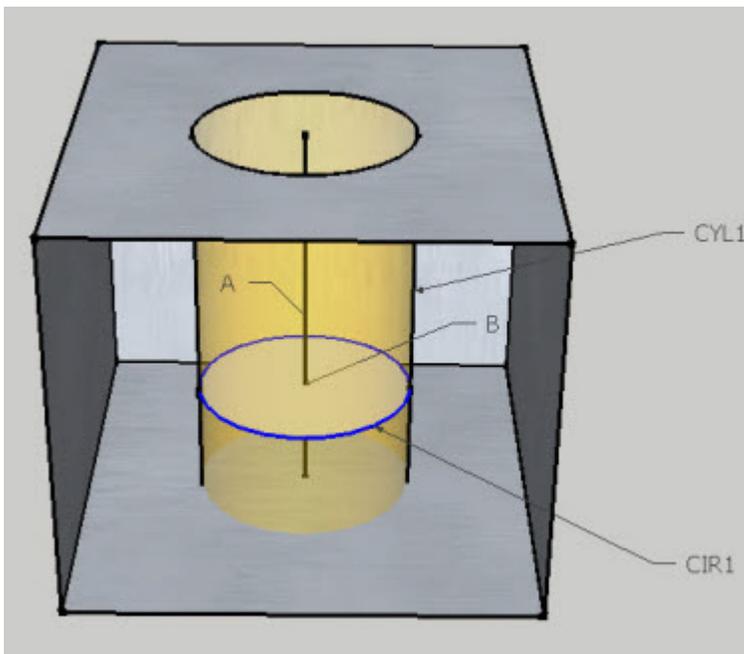
If you need an alternate or complementary angle, under **Angle Solutions**, you can tap the specific angle you want reported:



## Concentricity

This dimension tests how closely two circular features (circles, cylinders, cones, or spheres) share the same XY center point when projected to the same face or plane. The first feature is the feature being evaluated. The second feature is the datum feature. If you choose a sphere for the first feature, you must also choose a sphere for the second feature. The reported value shows the distance deviation between the two center points as a diameter. For example, two center points that are 0.5 millimeters apart results in a concentricity deviation of 1 millimeter. A measured value of 0 means the features are perfectly concentric. You cannot specify a nominal value for Concentricity since this dimension is a comparison between the two features' measured values.

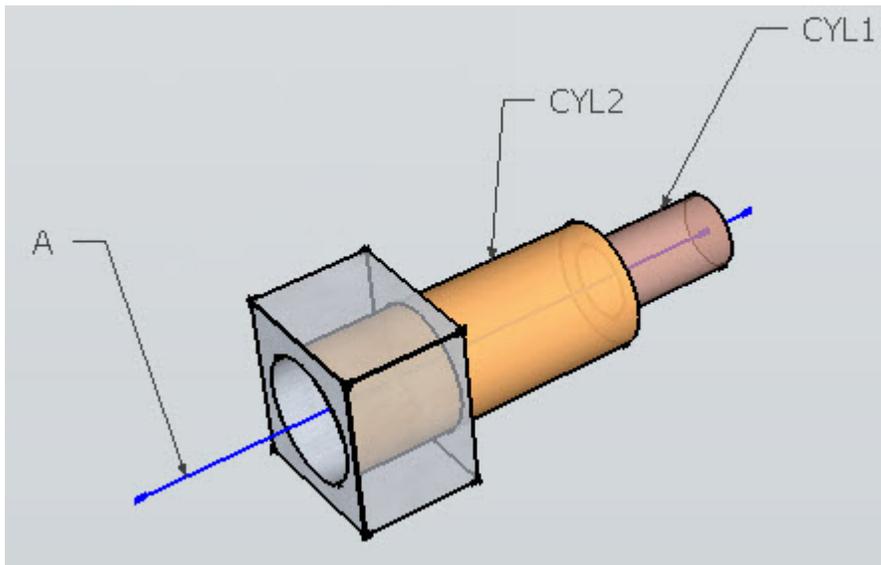
## Example



In this diagram, Concentricity can test if the centers of cylinder CYL1 (A) and CIR1 (B) share the same XY position.

## Coaxiality

This dimension tests how closely two linear features (cylinders, lines, or cones) share the same center line or axis. The first feature is the feature being evaluated. The second feature is always the datum feature. The reported value shows the deviation. A measured value of 0 means the features are perfectly coaxial. You cannot specify a nominal value for Coaxiality since this dimension is a comparison between the two features' measured values.

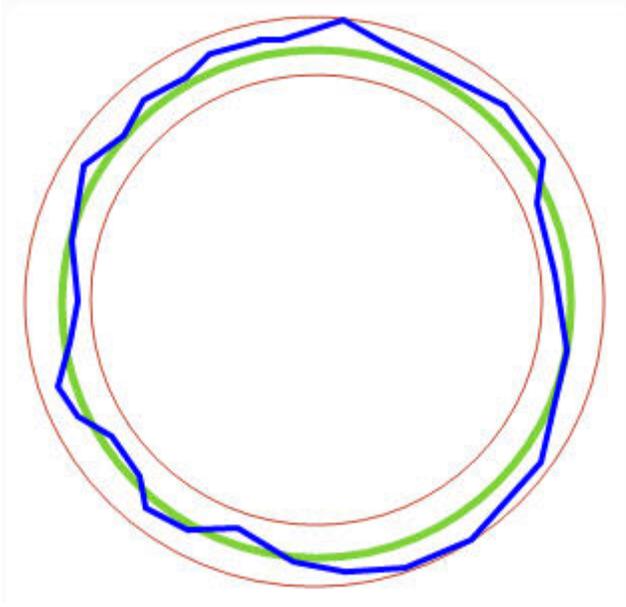


In this diagram, Coaxiality can test if the axes of the two cylinders, CYL1 and CYL2, share the same center line (A).

## Circularity

This dimension tests the circularity or roundness of cones, spheres, or circles. You cannot specify a nominal value for Circularity because it's a Form dimension. A measured value of 0 means the feature is perfectly circular.

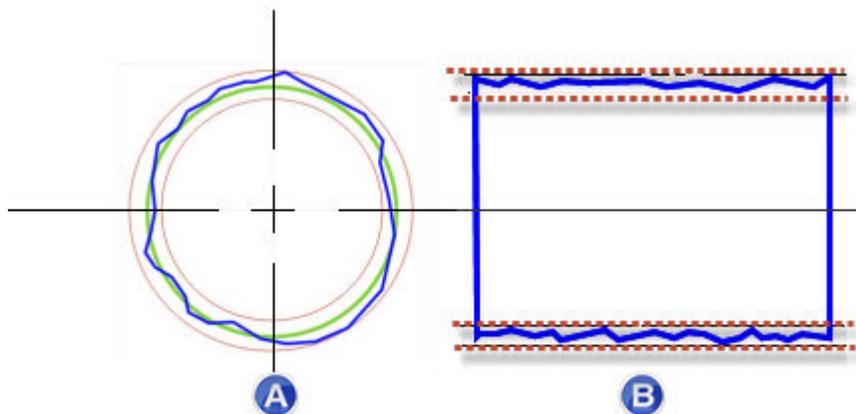
The tolerance value for circularity provides a tolerance band around the feature as shown here:



In this diagram, the green circle indicates the nominal circle, one with a perfect roundness. The brown circles indicate the tolerance band for an acceptable roundness dimension. The blue line shows the actual measured points connected by straight lines.

### Cylindricity

Similar to [Circularity](#), this shows how cylindrical a cylinder feature is. The tolerance creates a tolerance band around the entire surface of the feature. You cannot specify a nominal value for Cylindricity because it's a Form dimension. A measured value of 0 means the feature is perfectly cylindrical.

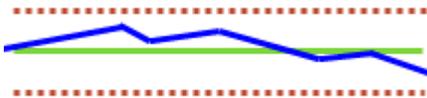


In this diagram of a cylinder, A is the top view and B is the side view. The solid brown lines in A and the dotted lines in B represent the tolerance bands. The blue line represents the measured cylinder and illustrates cylindricity.

### Straightness

This dimension tests if a linear feature is straight enough. Valid features include lines, cylinders, and cones. You cannot specify a nominal value for Straightness because it's a Form dimension. A measured value of 0 means the feature is perfectly straight.

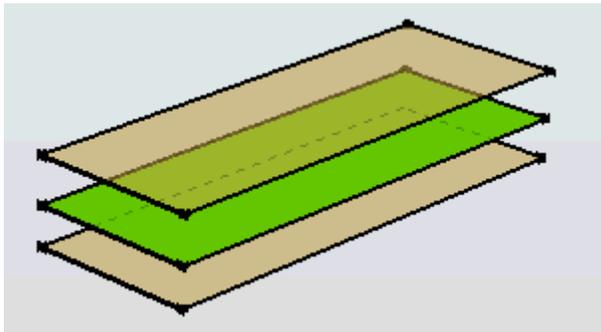
The tolerance value creates a tolerance zone around the line feature or around the center line of a cylinder or cone feature.



In this diagram, the green line represents a perfectly straight line. The dotted lines represent the tolerance zone established from the tolerance value. The blue line represents what a non-perfect measured line might look like.

### Flatness

This shows how flat a plane feature is. You cannot specify a nominal value for Flatness because it's a Form dimension. A measured value of 0 means the plane feature is perfectly flat. The tolerance value creates a tolerance zone above and below the plane feature.



In this diagram, the green plane represents a perfectly flat plane. The tolerance zone is represented by the translucent planes above and below it.

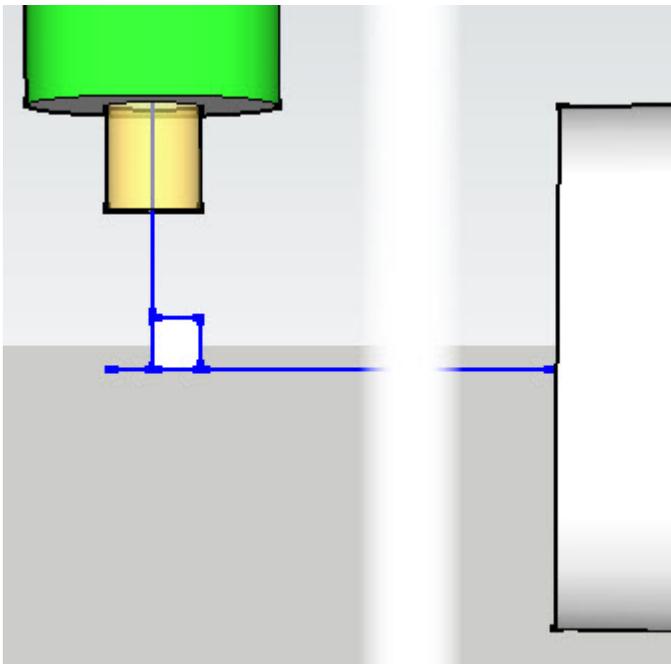
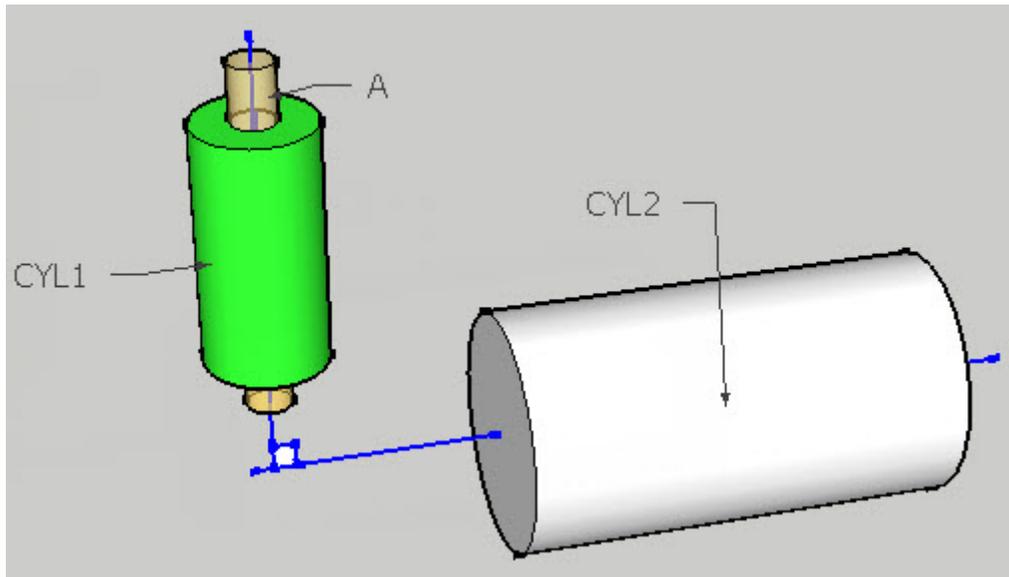
### Perpendicularity

This dimension tests if the angle between two features is 90 degrees. Valid features include lines, planes, cylinders, and cones. The first feature is the feature being evaluated. The second feature is the datum feature. You cannot

## PC-DMIS TOUCH CMM

specify a nominal value for Perpendicularity. A measured value of 0 means the feature is perfectly perpendicular to the datum feature.

The tolerance value creates a radius tolerance zone around the line, center line, or vector of the first feature, perpendicular to the datum feature.

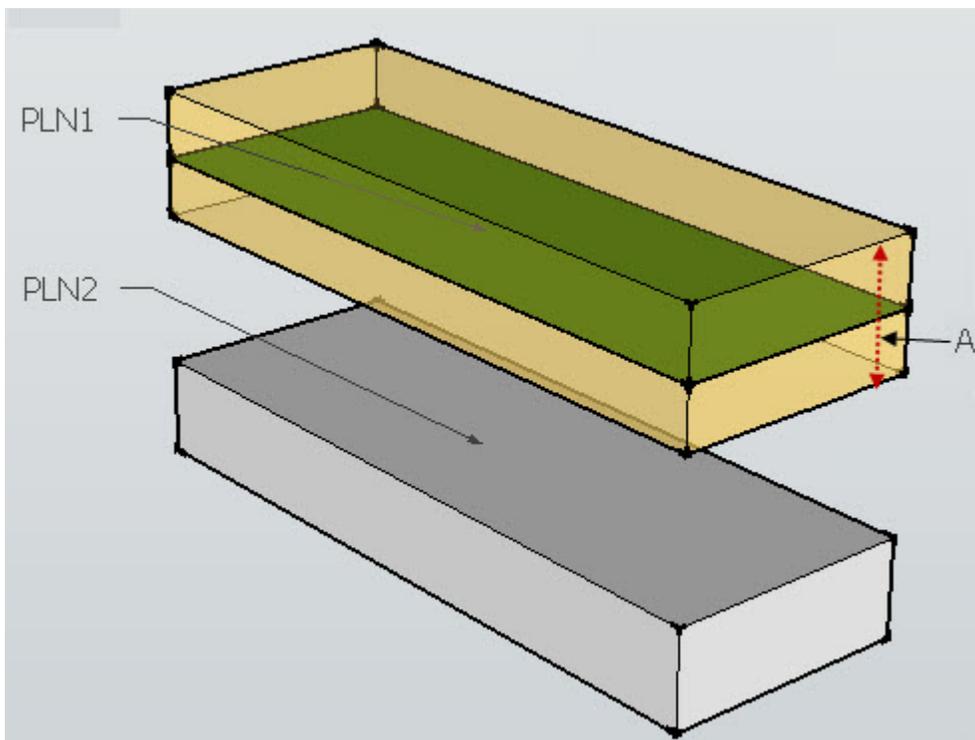


**These diagrams illustrate perpendicularity between two cylinder features. CYL1 is the first input cylinder and CYL2 is the second cylinder. The angle between the two center lines is compared against a perpendicular angle. The tolerance zone around the center line of CYL1 (A). The cylinder's centerline must be contained within this zone for the perpendicularity to pass.**

## Parallelism

This dimension tests if two features are parallel to each other. Valid feature types include lines, planes, cylinders, and cones. The first feature is the feature being evaluated. The second feature is the datum feature. You cannot specify a nominal value for Parallelism. A measured value of 0 means the feature is perfectly parallel to the datum feature.

The tolerance value creates a planar tolerance zone a specified distance away from the line, center line, or plane of the first feature parallel to the datum feature. The feature's axis or center plane must lie within this zone.



This diagram illustrates parallelism between two planes, PLN1 and PLN2. PLN1 is the first selected feature. PLN2 is the datum feature. The tolerance zone around PLN1, indicated by the translucent yellow box (A), shows the area that the plane being evaluated must lie within for the parallelism to pass.

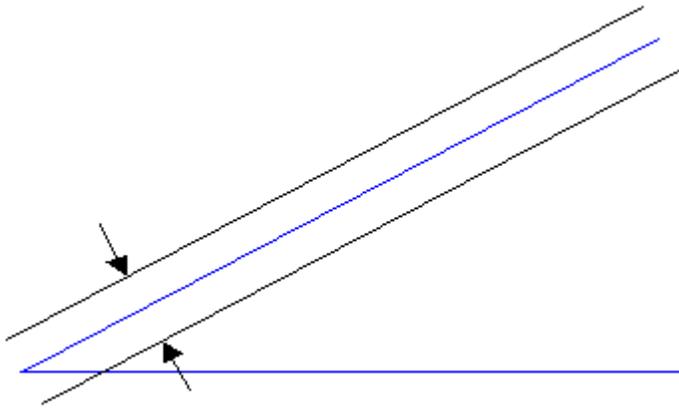
## Angularity

This dimension computes the angularity error of the first feature (a plane or line) to a second reference feature (a plane or line). The first feature is the feature being evaluated. The second feature is the datum feature. If only one feature is used, the software uses the work plane as the reference feature. This lets you

specify an angle that is not 90 degrees (that would be perpendicularity) or 0 degrees (that would be parallelism).

### Example

If you specify a 45-degree angle, the software creates a tolerance band at 45 degrees, and it checks that the hits are within that tolerance band.



The measured value is an angle (blue). The angularity's tolerance band is two parallel lines or planes (black) that never meet.

---

## Constructing Features

### About Constructed Features

Constructed features are constructed from existing learned features. Generally, constructed features don't naturally exist on the part itself or they are difficult to reach or measure.

You may need constructed features if you need to measure a feature that you cannot easily probe on the part, or the feature simply doesn't exist (such as an intersection point between a sphere and a line); or you may need it to set your alignment at a certain location (for example, to set the origin of the alignment at an intersection of two lines).

Constructed features require that you specify a type of feature construction (point, line, plane, circle, slot, or cylinder), a construction method type for the selected feature, and finally the actual input features needed for the construction. For example, to create an intersection point between two lines, first choose the point construction type, then the intersection point method, and then select or measure the two line features.

**Related Topics:** [The New Construction Screen](#), [How to Construct a Feature from Other Features](#)

## How To

### How to Construct a Feature from Other Features

**Objective:** To construct features from other features.

**Concepts:** [About Constructed Features](#), [The New Construction Screen](#)

**Prerequisites:** You must have some [learned features](#) in your part file. Generally, you should [define an alignment](#) first too, although you can use some constructed features as alignment inputs.

#### Follow this procedure:

1. Ensure that the insertion point is where you want it in the Commands list by tapping on the desired command. For more information on the insertion point, see "[The Commands List](#)" topic.
2. Access the **New Construction** screen. You can do this by doing one of the following:
  - Create a brand new part file to open the screen. From the **Home** screen, on the **New** tile, tap **Construction**:



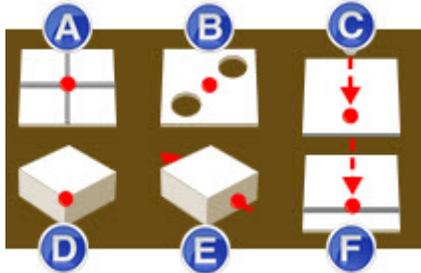
- Edit an existing part file and then from the bottom toolbar tap **New Construction**:



From the left pane, follow the instructions for each step. From the right pane, make all your selections. After you make the proper selection, the software automatically advances to the next step. On methods that allow you to select an undetermined number of features for the construction, you can tap on a feature again to deselect it.

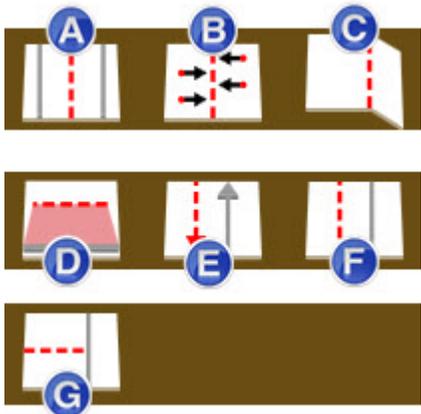
3. For **Type**, from the right pane, tap the type of feature to construct.
4. For **Method**, from the right pane, tap the method to use for the feature construction. If only one method is available for a type, it is automatically selected.

### Point Construction Methods



- A. Intersection Point
- B. Middle Point
- C. Projection Point
- D. Corner Point
- E. Pierce Point
- F. Drop Point

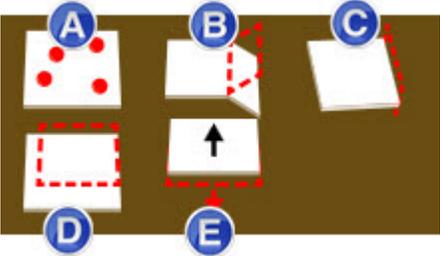
### Line Construction Methods



- A. Middle Line
- B. Bestfit Line
- C. Intersection Line
- D. Projection Line
- E. Reverse Line
- F. Parallel Line
- G.

G. Perpendicular Line

Plane Construction Methods



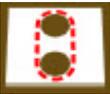
- A. Bestfit Plane
- B. Middle Plane
- C. Perpendicular Plane
- D. Parallel Plane
- E. Reverse Plane

Circle Construction Methods



- A. Intersection Circle
- B. Bestfit Circle
- C. Projection Circle

Round Slot Construction Methods



Two Circle Round Slot

Cylinder Construction Methods



Two Circle Cylinder

5. For **Select/Measure**, on the left pane, the software indicates the types of features to select or measure.

- a. From the filtered list on the right or from the Graphics window, tap a valid feature to select it.
  - b. If the needed feature does not exist, measure it now. To measure, take hits on the part with your measurement device. See "[How to Learn Features to Measure](#)" for more information. Once the feature is measured, it becomes selected.
  - c. Continue selecting input features until you have all your features for your selected construction. If you are constructing a best fit feature, tap **Next** once you're done selecting the input features.
6. Once the needed features are selected or measured, the screen changes to show the constructed feature's name.
  7. For **Complete**, if you need to change the feature's name, tap on the name, and type a new value.
  8. After the features are selected for the construction, tap either **Next** or **Done**.
    - Tap **Done** to automatically create the feature, accepting the default nominals and tolerances. A feature created in this way does not appear in the report. This helps speed up the creation of commands when you don't need to display a feature in the report.
    - Tap **Next** to proceed to edit the feature's nominals and tolerances.
  9. For **Adjust**, this step is optional. If you tapped **Next** instead of **Done**, you can modify tolerances and nominals and choose what axes to send to the report. For more information, see "[Nominals, Tolerances, and Axes](#)".
    - a. In each box you want to edit, tap the nominal or tolerance box, and then use the on-screen keypad to type the desired value. The available axes and boxes vary depending on the type of feature being constructed. Continue doing this until you make the changes you need.
    - b. Tap the colored axis row that has a small dot to its left to show or hide that axis. This way you can control what information ends up in the report. You cannot hide axes without dots. Colored axes appear in the report. Empty axes do not.
  10. At any time you can correct or cancel the construction as needed.
    - To correct it, use **Previous** to go back to an earlier step, or from the left pane, tap on the desired step.
    - To cancel it entirely and close the screen, tap **Cancel**.

11. If you're satisfied with the construction, tap **Done** to accept the construction and close the screen. Or, press DONE  on your jog box.
12. If you want to accept the construction but keep the screen open to create another constructed feature, tap  instead.

The constructed feature is then visible in the part file.

## Reference Topics

### The New Construction Screen

You should access this screen when you want to create a feature from other features, such as the intersection point between two lines. This screen appears if you do one of the following:

- Create a brand new part file and open the screen by tapping the **Construction** button from the **New** tile in the **Home** screen:



- Edit an existing part file and then tap the **New Construction** button from the bottom toolbar:



The screen contains these items:

- **Header** - Initially this shows "New construction". After you define the input features for a constructed feature, this text changes to display the constructed feature's name.
- **Left Pane** - The left pane shows the required steps you need to take to create the constructed feature. The highlighted heading shows the current action you must take.
- **Graphics window** - The Graphics window between the left and right panes shows a drawing of all of the learned features in your part file in relation to each other. You can use this window to select the input features used in the construction, or as a reference when selecting input features from the right pane. When you select an item from the file list, the software

highlights the selected feature in yellow and centers and orients the view. See "[The Graphics Window](#)" for more information.

- **XYZ Information** - Beneath the Graphics window, you can see **X, Y, Z** fields. This is a digital readout of the probe's current position.
- **Right Pane** - During each phase of the construction process, the right pane changes to contain the construction types, methods, and input features, you can select.

Once you select the features for the construction, and the left pane highlights **Complete**, you can tap **Next** to have the right pane show a grid of actual, nominal, deviation, and tolerance values. You can use this pane to change the nominals and tolerances, and to show or hide axes from the report. For more information, see "[Nominals, Tolerances, and Axes](#)".

When measuring or constructing a feature, if both the **Next** and **Done** buttons are available, and you choose **Done**, the feature is created with the default nominal and tolerance values, but it is excluded from the report so you can create the command more quickly. If you want to later display it in the report, you can edit the feature and tap the dot to the left of the feature name. For information on editing a feature in a file, see "[How to Edit a Part File](#)".



The create another button, , creates the constructed feature but keep the screen open to create another constructed feature.

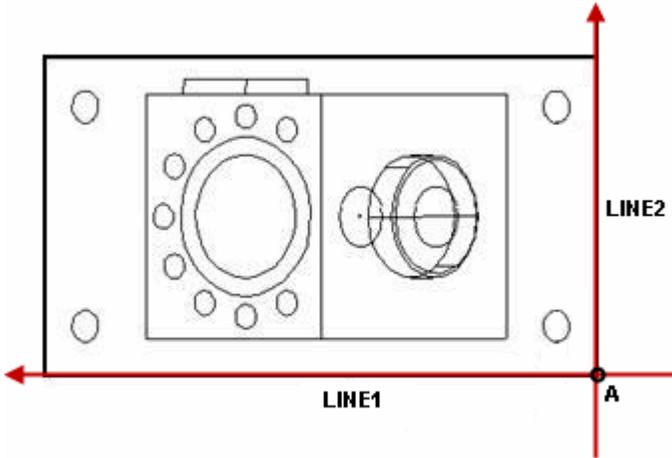
**Related Topics:** [Constructing Features to Measure](#), [Constructed Features](#)

## Point Construction Methods

### Intersection Point

This creates a point at the intersection of two reference features.

### Example

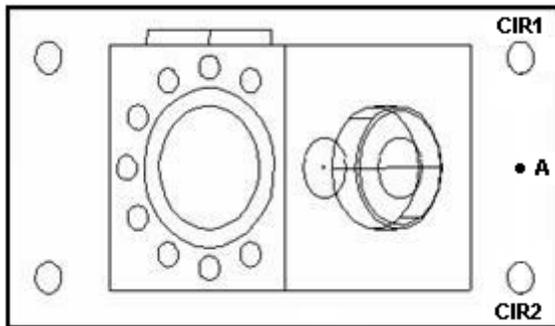


A - Point constructed from two lines (LINE1 and LINE2).

### Middle Point

This creates a point midway between two reference features.

### Example



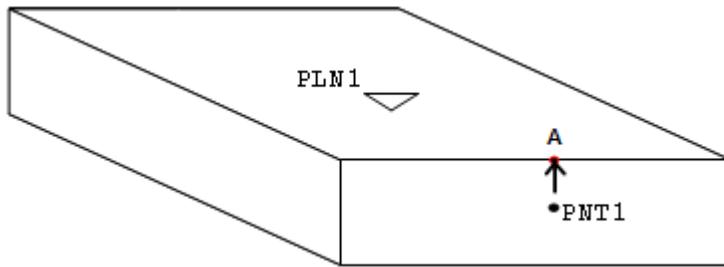
A - Point constructed midway between two circles, CIR1 and CIR2.

### Projection Point

This projects a point onto the selected plane feature.

### Example

## PC-DMIS TOUCH CMM

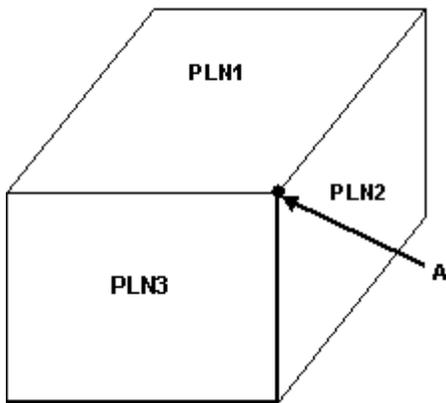


**A** - Projected point constructed from a point, PNT1, and a plane, PLN1.

### Corner Point

This creates a point at the intersection of three plane features.

### Example

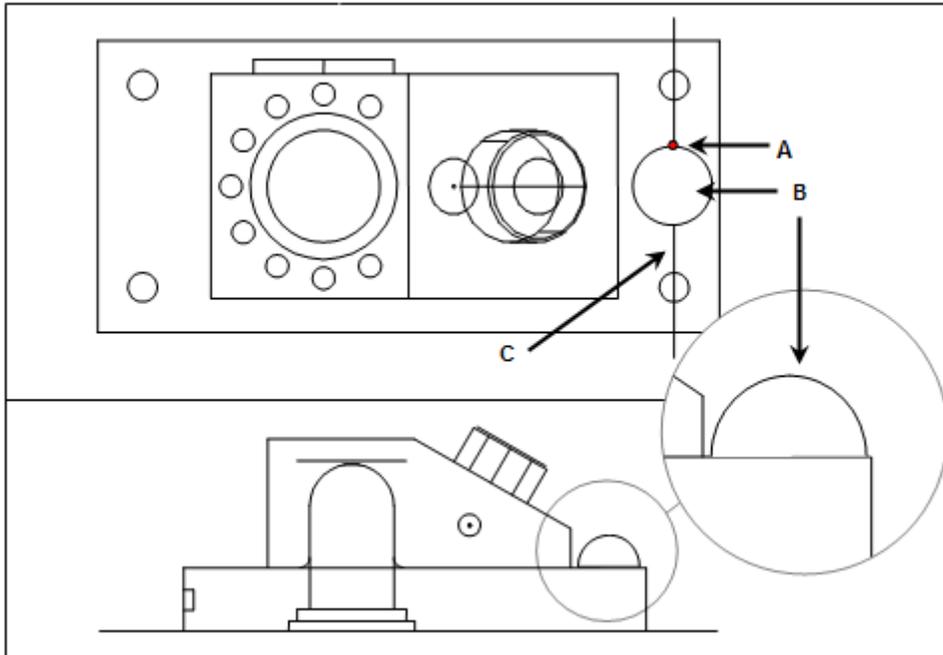


**A** - Point constructed at the intersection of three planes (PLN1, PLN2, and PLN3).

### Pierce Point

This creates a point at the location where one feature pierces the surface or intersects with another feature.

### Example



**A** - Point constructed at the intersection of a line and sphere.

**B** - Sphere feature.

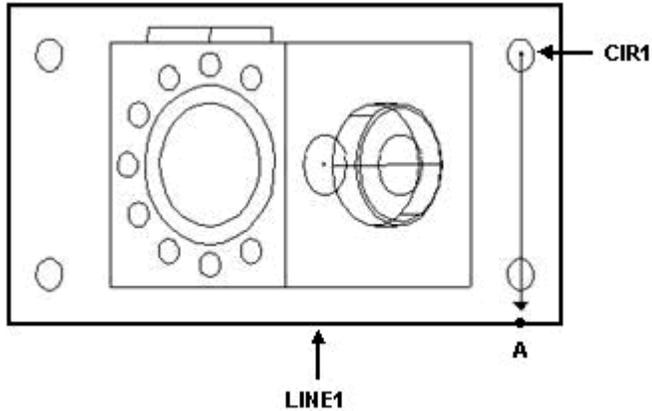
**C** - Line feature. Note that in this case, the direction of the line feature determines which intersection point with the sphere to use.

### Drop Point

This creates a point by dropping the centroid of the first feature onto the second feature. The dropped point is dropped on a line perpendicular to the line, centerline, or plane. If you select two lines, the software drops the centroid of the first line feature onto the second line feature.

### Example

## PC-DMIS TOUCH CMM



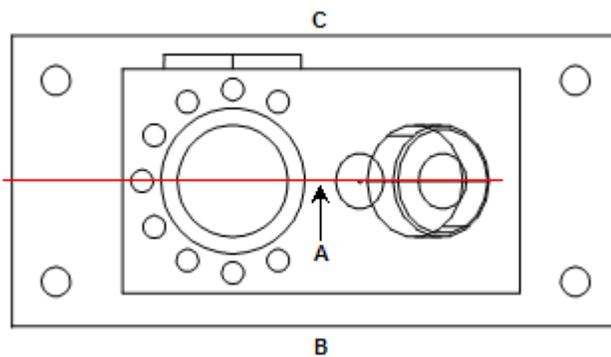
**A** - Point constructed by normally projecting circle, CIR1 (the designated point), to line, LINE1.

## Line Construction Methods

### Middle Line

This creates a line midway between two reference features.

### Example



**A** - Constructed middle line, equally spaced between two lines, B and C (from the front and back of the part).

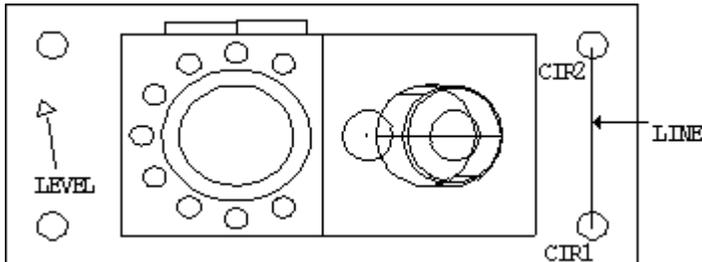
**B** - Front of part

**C** - Back for Part

### Bestfit Line

This creates a line that fits as best it can between multiple reference features. The best fit algorithm computes the feature with the least amount of error possible from each feature.

### Example



If you had two input features, the line is drawn between those two features. However, suppose you construct a line from three points. The line cannot obviously pass through all of the points, but it attempts to minimize the distance of the points from the constructed line.

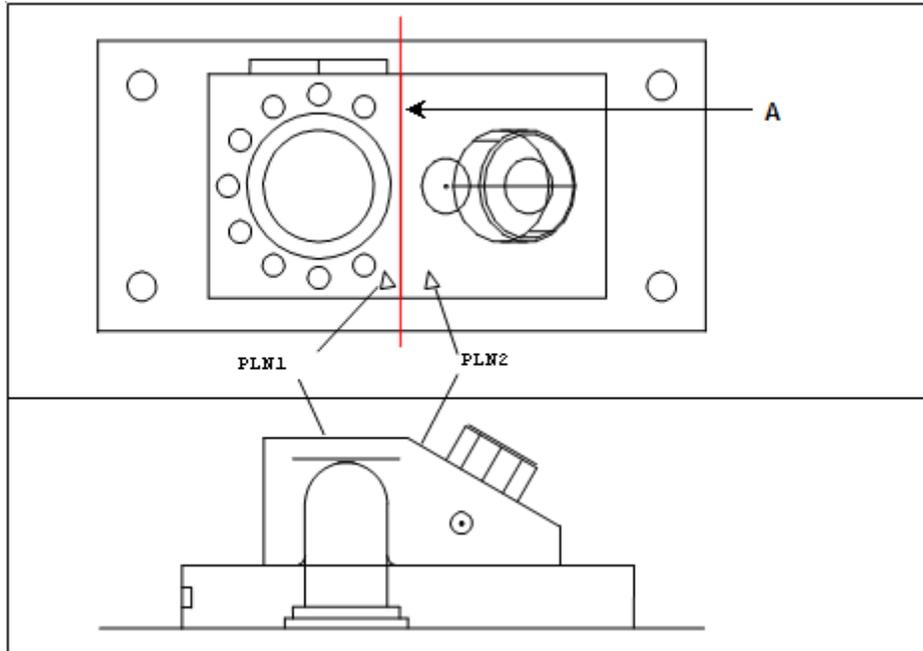
**Note:** If you only use point features as your input features in a best fit construction, the software compensates the constructed feature by the probe tip's radius. This means the construction behaves as if each point had been registered at the very center of the probe tip.

If you use anything other than only point features in a best fit construction, the software does not compensate the constructed feature by the probe tip's radius.

### Intersection Line

This creates a line at the intersection of two plane features.

### Example

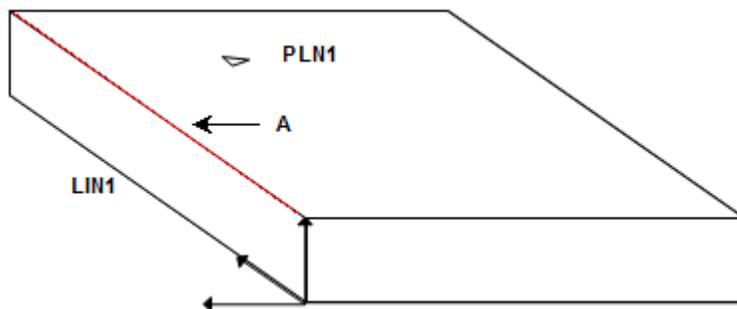


**A** - Line constructed at the intersection of two planes (PLN1 and PLN2).

### Projection Line

This constructs a line feature by projecting the first feature to where it intersects with the second (plane) feature.

### Example



**A** - Constructed line projected from a line feature (LIN1) onto a plane feature (PLN1).

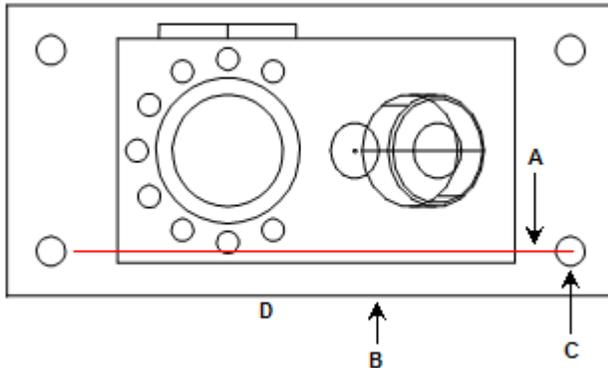
### Reverse Line

This creates a line with a reverse vector from the selected line feature.

## Parallel Line

This creates a line parallel to the first feature and passing through the center of the second feature.

### Example



**A** - Line constructed parallel to the first feature and through the center of the second feature.

**B** - First feature

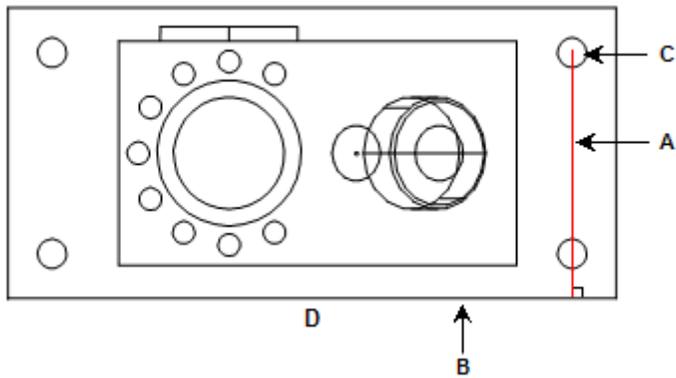
**C** - Second feature

**D** - Front of part

## Perpendicular Line

This creates a line perpendicular to the first feature and passing through the center of the second feature.

### Example



**A** - Line constructed perpendicular to the first feature and through the center of the second feature.

**B** - First feature

**C** - Second feature

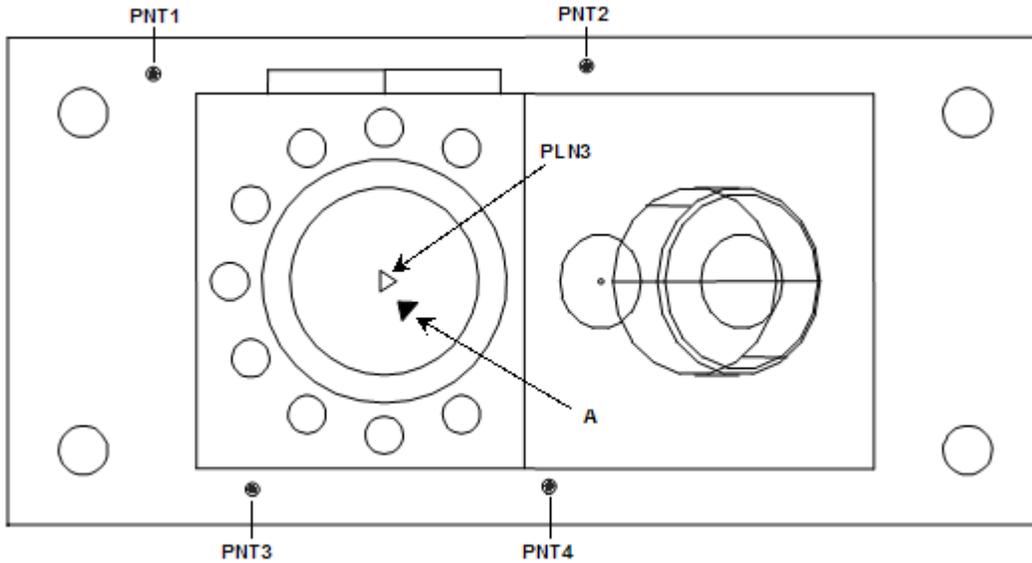
**D** - Front of part

## Plane Construction Methods

### Bestfit Plane

This creates a plane that fits as best it can between multiple reference features. The best fit algorithm computes the feature with the least amount of error possible from each feature.

### Example



**A** - Best fit plane constructed from four points (PNT1, PNT2, PNT3, and PNT4) and a plane (PLN3).

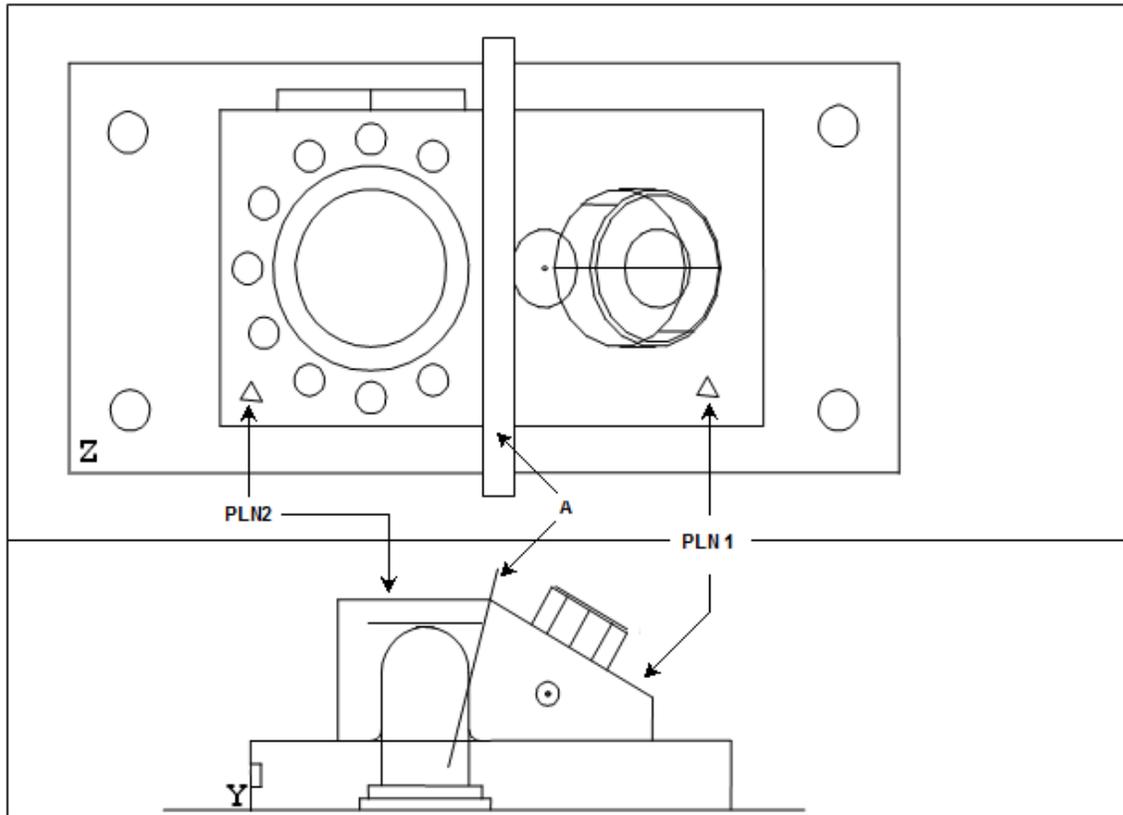
**Note:** If you only use point features as your input features in a best fit construction, the software compensates the constructed feature by the probe tip's radius. This means the construction behaves as if each point had been registered at the very center of the probe tip.

If you use anything other than only point features in a best fit construction, the software does not compensate the constructed feature by the probe tip's radius.

### Middle Plane

This creates a plane midway between two other features.

### Example

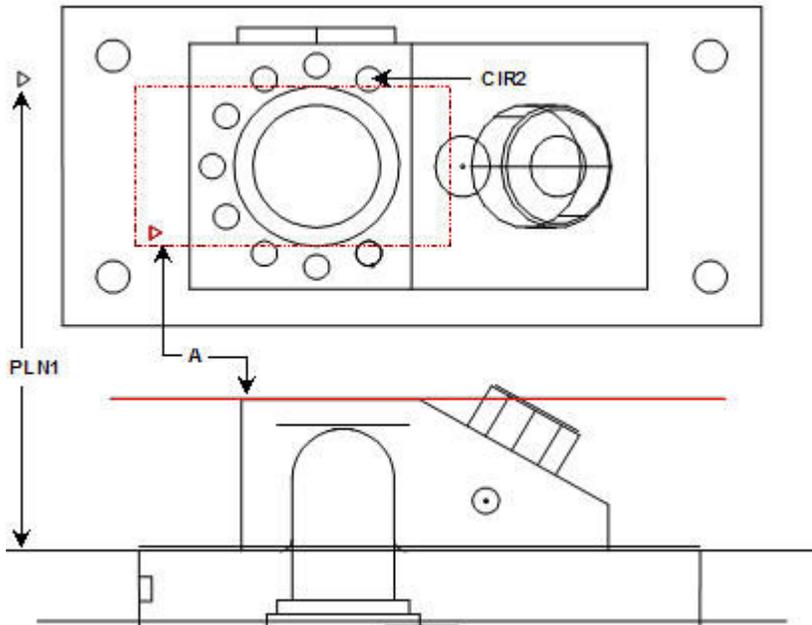


**A** - Constructed plane equally spaced between two planes (PLN1 and PLN2).

### **Parallel Plane**

This creates a plane parallel to the first feature and passing through the centroid of the second feature.

### **Example**

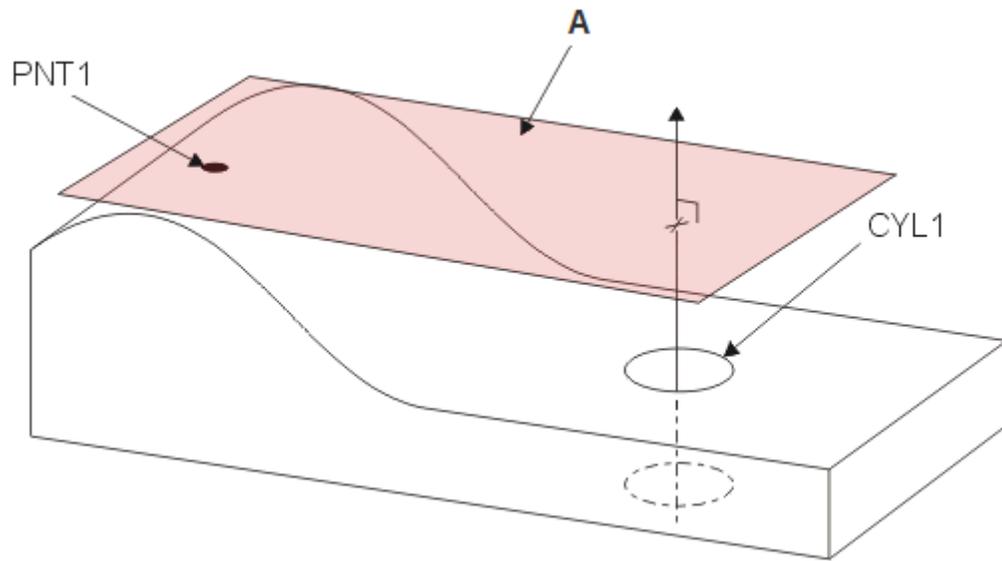


**A** - Plane constructed parallel to the first feature, a plane shown as PLN1, and through the second feature, a circle shown as CIR2.

### Perpendicular Plane

This creates a plane perpendicular to the first feature and passing through the center of the second feature.

### Example



**A** - Plane constructed perpendicular to a cylinder feature (CYL1) and through a High Point feature (PNT1). Note that CYL1 contains an axis element.

**CYL1** - The first feature is a cylinder.

**PNT1** - The second feature is a High Point feature.

### **Reverse Plane**

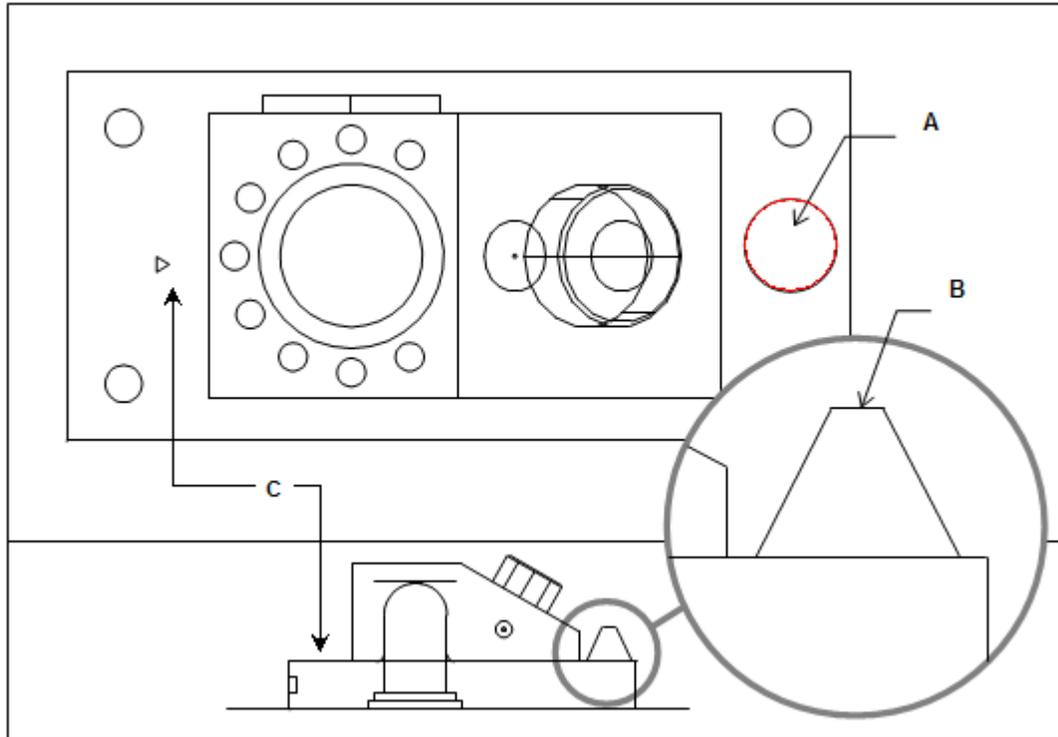
This creates a plane with a reverse vector from the selected plane feature.

## **Circle Construction Methods**

### **Intersection Circle**

This creates a circle at the intersection of a circular feature with a plane, cone, or cylinder.

### **Example**



**A** - Circle constructed from the intersection of a cone and a plane.

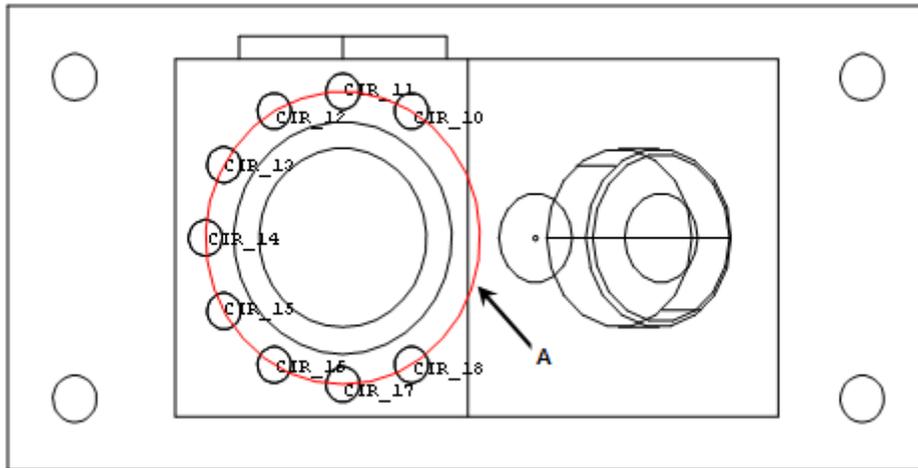
**B** - Cone feature

**C** - Plane feature

### **Bestfit Circle**

This creates a circle as best it can between multiple reference features. The best fit algorithm computes the feature with the least amount of error possible from each feature.

### **Example**



**A** - Best fit circle constructed from three or more features (in this example, from three or more circle features).

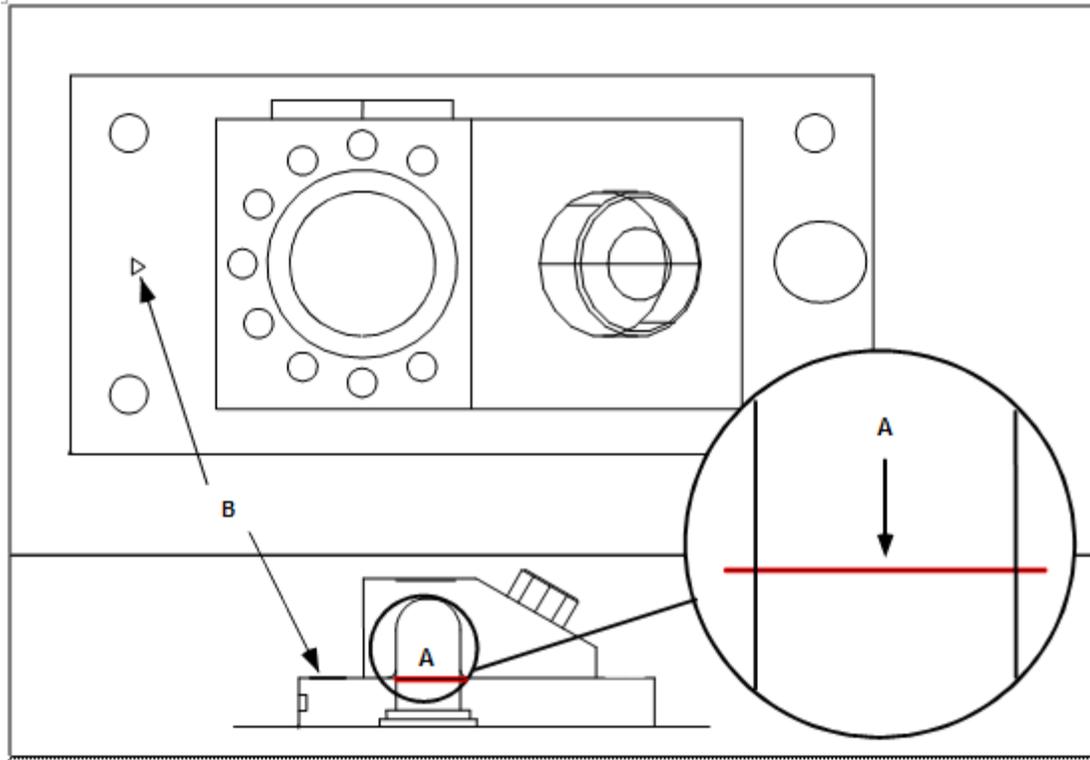
**Note:** If you only use point features as your input features in a best fit construction, the software compensates the constructed feature by the probe tip's radius. This means the construction behaves as if each point had been registered at the very center of the probe tip.

If you use anything other than only point features in a best fit construction, the software does not compensate the constructed feature by the probe tip's radius.

### Projection Circle

This creates a circle from the projection of the first feature onto the second plane feature.

### Example



**A** - Circle constructed from the projection of a circle and a plane.

**B** - Plane feature

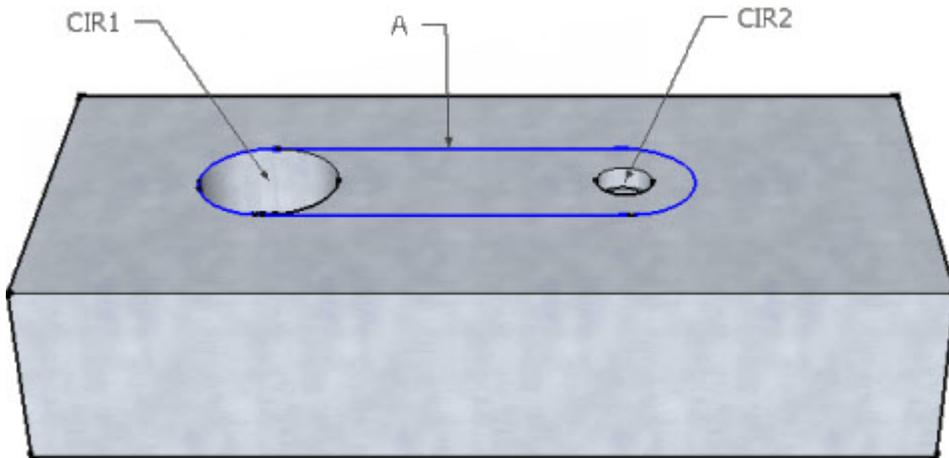
## Round Slot Construction Methods

### Two-Circle Round Slot

This creates a round slot constructed from two circles. The round slot is mostly defined by the first circle feature. It is constructed in the same plane as the first circle. The width of the slot is also determined by the diameter of the first circle. The second circle is only used to determine the length of the slot. The length is the distance from the center of the first circle to the center of the second circle, plus the diameter of the first circle.

If the two input circles are not in the same plane, the center of the second circle is projected perpendicularly onto the plane of the first circle. The distance is then calculated from the center of the first circle to the projected center of the second.

### Example



**A** - Round slot created from two circle features.

**CIR1** - The first circle feature.

**CIR2** - The second circle feature.

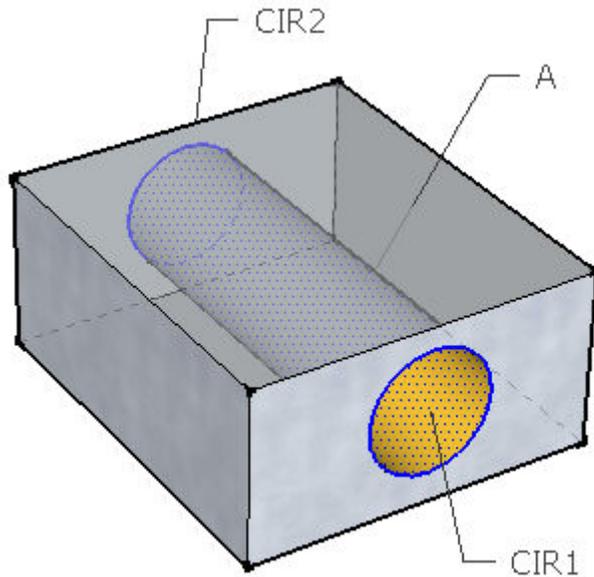
## Cylinder Construction Methods

### Two Circle Cylinder

This creates a best fit cylinder from two circle features. The software constructs the cylinder using the same best fit algorithm as the Bestfit cylinder for six points (three points for each circle). The two circles can be any circle feature of any size.

- If the two circles exist on the same plane, a cylinder with a height of zero is constructed.
- If the two circles exist on different planes, the cylinder is constructed with a height/depth of the distance between the two centroids.

### Example



A - Cylinder constructed from two circle features (CIR1) and (CIR2).

---

## Using DCC Mode

### About DCC Mode and Manual Mode

PC-DMIS Touch supports the following basic probe modes through a single **DCC** (Direct Computer Control) **Mode** command that can be turned **On** or **Off** inside the Commands list:

You can add the DCC Mode command into the Commands list from **Edit** screen on the bottom level of screens by tapping the **DCC**  toolbar button. If an alignment or ClearanceCube do not exist, the software prompts you to create them. The initial **On** or **Off** state of any newly added **DCC Mode** command is the opposite of the bottom-most DCC Mode command in the Commands list.

If you set the **DCC Mode** command to **Off**, the command changes to read "DCC Mode - Manual". In this mode, when you run the part file, the software expects you to manually control the machine's movements and measurements for any features in the Commands list after this mode command. You should use this mode during the manual alignment portions

of a part file that you intend to run on an automatic CMM or for any other features in that file that you want to manually measure. This is the default mode when even if there's explicitly no DCC Mode command in the Commands list.

If you set the **DCC Mode** command to **On**, the command changes to read "DCC Mode - Dcc". In this mode, when you run the part file, the software take over and drives the CMM to automatically run any commands or features listed after this mode command.

**Related Topics:** [Edit Screen](#), [How to Enable DCC or Manual Mode](#), [The ClearanceCube](#), [How to Define a Clearance Cube](#), [About Move Points](#), [How to Use a Move Point Command](#)

## How To

### How to Enable DCC or Manual Mode

**Objective:** To add a DCC (Direct Computer Control) Mode command into the Commands list and set it to On or Off so that machine motion to measure features is done respectively by either the computer or by you.

**Concepts:** [About DCC Mode and Manual Mode](#), [The ClearanceCube](#)

**Prerequisites:** You must have an existing part file created and open. You must have a valid alignment defined. You must have a clearance cube defined.

#### Follow this procedure:

1. Access the **Edit** screen on the bottom level of screens.
2. Tap the **DCC**  toolbar button to add a **DCC Mode** into the Commands list. If a ClearanceCube has not yet been defined for this program, the [Define ClearanceCube](#) screen appears.

The initial **On** or **Off** state of any newly added **DCC Mode** command is the opposite of the bottom-most DCC Mode command in the Commands list.

3. To change it from the default initial state, tap the newly added **DCC Mode** command.
4. Tap **Edit**  to place the command in Edit mode.

5. Tap **On** or **Off** as needed to change that command's state.
6. Tap **Accept** .
  - If you set it to **On**, the software is in DCC mode, and the command changes to read "DCC Mode - Dcc".
  - If you set it to **Off**, the software is in manual mode, and the command changes to read "DCC Mode - Manual".

All subsequent commands use the selected mode until the next **DCC Mode** command.

Similarly, you can tap on any DCC Mode command in the Commands list to change its state.

## How to Define a ClearanceCube

**Objective:** To create a box-like protective envelope, or ClearanceCube, around the part that the probe moves during automatic moves to between features in order to prevent collisions with the part.

**Concepts:** [The ClearanceCube](#), [The Define ClearanceCube Screen](#)

**Prerequisites:** From the **Home** screen, access the **Files** screen. Ensure you have a probe attached and [calibrated](#). Ensure you have an [alignment](#) defined.

### Follow this procedure:

1. If you haven't defined a ClearanceCube yet, from the bottom toolbar, tap the **DCC Mode**  button to open the **Define ClearanceCube** screen. If you've already defined a ClearanceCube, tapping this button only adds in a **DCC Mode** command. If you need to edit an existing ClearanceCube, above the Commands list tap the status icon for the cube .
2. Define the two points that make up the protective box by doing one of the following:
  - Use your jog box and position your probe at the desired location, and then on the jog box, press the DONE button  to read that position.

Or

- In the right pane of the **Define ClearanceCube** screen, tap on the **X**, **Y**, or **Z** boxes, and use the numeric keypad to manually type the values.
3. Tap **Next** and in the right-pane review the cube's final values. You can also adjust the offsets faces from this screen. On the right pane, tap on the boxes and change their values as needed.
  4. Once everything is done, if you don't like the defined box, you have these options:
    - In the left pane, tap on the **Define Point 1** text to restart the process from the beginning.
    - Tap **Previous** (or press ERASE HIT  on your jog box) to delete the points one at a time and then make corrections.
    - Cancel the definition entirely and close the screen by tapping **Cancel** or tapping the white X button in the upper-right corner of the screen.
  5. If you're satisfied with the cube definition, tap **Done** to store the definition with the file or press DONE  on your jog box. The software then adds the **DCC Mode** command into the Commands list.

When a ClearanceCube is defined, a [status icon](#) displaying a cube appears above the Commands list. You can tap on this status icon to edit the ClearanceCube.

By default, when you create a new feature, the software determines what ClearanceCube face it uses based on the probe shank vector. You can tell it to use a different ClearanceCube face when you [create](#) or [edit](#) a feature command.

## How to Use a Move Point Command

**Objective:** To insert a command that tells the machine to move the probe to a specific XYZ location.

**Concepts:** [About Move Points](#)

**Prerequisites:** From the **Home** screen, access the **Files** screen. Ensure you have a probe attached and [calibrated](#). Ensure you have an [alignment](#) defined.

**Follow this procedure to add a Move Point command:**

1. From the Commands list, tap the command where you want to insert the **Move Point** command. The move point is inserted after this command.
2. Move the probe to the desired location and then from the T-Gage CMM's jog box, press the move point button .
3. The CMM sends the XYZ information to the software, and the software inserts the **Move Point** after the selected command. When the software runs the file and it encounters this command, it immediately moves to the defined XYZ location.

**Follow this procedure to Edit a New Move Point command:**

1. From the Commands list, tap the **Move Point** command.
2. Tap **Edit** .
3. To specify new coordinates to move to, tap the **X**, **Y**, or **Z** boxes, and type in the new values.

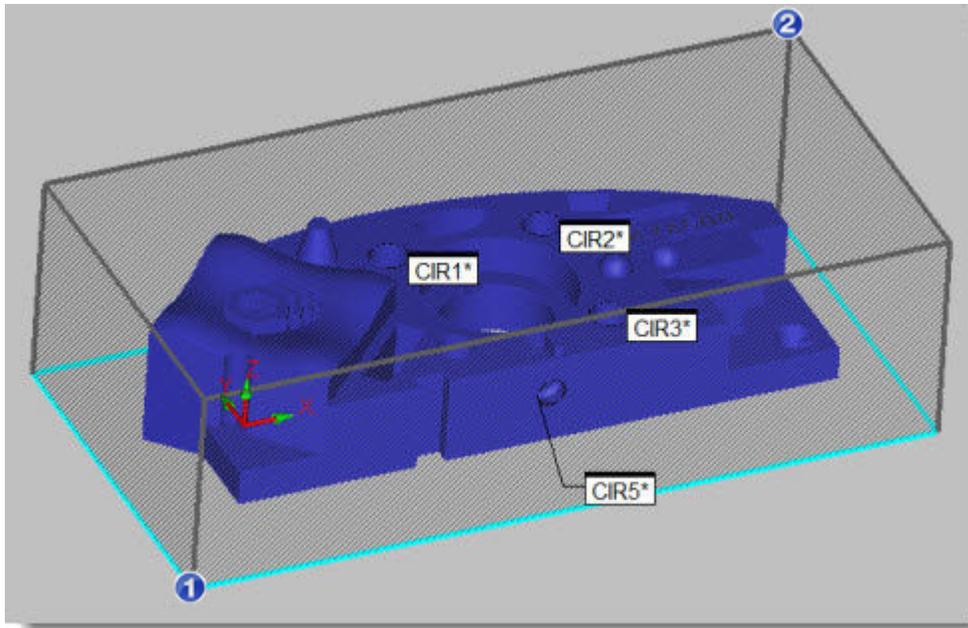
Or

4. To refresh the **X**, **Y**, or **Z** boxes to take the current probe location, tap **Read Position** .

## Reference Topics

### The ClearanceCube

The ClearanceCube in PC-DMIS Touch is a three-dimensional box that represents a protective envelope that surrounds the CAD model. When you do any motion in DCC mode, the probe moves to the offset distance of the box to avoid colliding with the part. The box is defined inside the Define ClearanceCube screen by specifying two points as indicated below:



**This example illustrates an imagined protective box surrounding a part and a set of possible point locations, indicated by 1 and 2, to define the box's dimensions.**

The two points define the three-dimensional boundaries of the cube. For example, you might define a point some distance away from the lower-left corner of a part and the second in space above and beyond the upper-right corner of the part, as shown in the figure above.

When a ClearanceCube is defined, a [status icon](#) displaying a cube appears above the Commands list. You can tap on this status icon to edit the ClearanceCube.

By default, when you create a new feature, the software determines what ClearanceCube face it uses based on the probe shank vector. You can tell it to use a different ClearanceCube face when you [create](#) or [edit](#) a feature command.

**Related Topics:** [The Define ClearanceCube Screen](#), [How to Define a ClearanceCube](#)

## The Define ClearanceCube Screen

This screen appears automatically if you do not have a ClearanceCube defined, and you tap **DCC Mode**  from the bottom-level **Edit** screen to tell your computer to directly control the CMM's motions.

Use the **Define ClearanceCube** screen to define the cube by measuring two points with your probe or by typing the values directly into the **XYZ** boxes on the right pane. The two points should represent two points in space at opposite corners of the cube. The defined cube should exceed the part's volume.

When a ClearanceCube is defined, a [status icon](#) displaying a cube appears above the Commands list. You can tap on this status icon to edit the ClearanceCube.

By default, when you create a new feature, the software determines what ClearanceCube face it uses based on the probe shank vector. You can tell it to use a different ClearanceCube face when you [create](#) or [edit](#) a feature command.

### ClearanceCube Errors:

You may see these errors from time to time in the right-pane of this screen:

**DCC Mode requires a ClearanceCube-** This message appears if you try to add a DCC Mode command into the Commands list and a clearance cube is not defined.

To fix this, you must first [define the ClearanceCube](#). Once defined, the DCC Mode command is added into the Commands list.

**Invalid cube definition** - This message appears if you try to create the ClearanceCube, but the points used are not valid. For example, if all the values that define the two points are still at zero.

To fix this, [redefine the ClearanceCube](#) with proper values.

**Related Topics:** [The ClearanceCube](#), [How to Define a ClearanceCube](#)

## About Move Points

A **Move Point** command moves the probe to a specific XYZ location within your machine volume. You can insert a **Move Point** command by pressing the move point button on your jog box. The button can vary depending on the machine or jog box used. Consult your jog box documentation for the exact button to press.

From the jog box on the T-Gage CMM, you can press this button to add the **Move Point** command:



When you run the file and the software encounters the **Move Point** command, it moves the probe to that XYZ location. If you have a ClearanceCube enabled, a **Move Point** command takes precedence, and after the software runs the Move Point command, it returns to using the ClearanceCube.

To edit a **Move Point** command with the current probe position, select the command, tap **Edit**  and then edit the values or tap **Read Position** .

**Related Topics:** [How to Use a Move Point Command](#)

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## Running Your Part File

### About Run Mode and Running Your Part File

Run mode is also known as Execution mode or Measure mode. In this mode, the software "runs" learned features and other commands that are stored in part files to measure them. Running a part is the final step needed before you can view measurement results in a report.

The software enters Run mode whenever you access the **Run** screen and tap **Run**. In Run mode, important messages for completing the measurement process appear in the status bar and inside the Commands list of the **Run** screen.

**Related Topics:** [How to Run a Part File](#), [The Run Screen](#)

## How To

### How to Run a Part File

**Objective:** To run or execute a part file containing learned feature commands in order to generate a report of measurement results.

**Concepts:** [About Run Mode and Running Your Part File](#), [The Run Screen](#)

**Prerequisites:** Any probes used in the file must be attached to the machine or to the [probe changer](#). You must successfully [calibrate](#) any probe tips you are going to use.

You also need a part file available with the following:

- [A command that loads a calibrated probe](#)
- [A properly defined alignment](#)
- [Learned features](#)
- [Defined dimensions](#)

#### Follow this procedure:

1. From the **Home** screen, under the **Files** tile, tap the file to run. If it's not visible, from the **Files** tile, tap **More Files**.
2. From the list of files, tap the file that you want the software to load.
3. Swipe horizontally until you see the **Run** screen. This is the main interface for running a part file.
4. Start running the part file by doing one of the following:
  - *Run only the selected command* - Tap a command to select it, and then next to the command, tap **Run** .
  - *Run from the selected command to the end* - Tap a command to select it, and then from the bottom toolbar, tap **Run** . When the message that appears asks if you want to run from the top of the Commands list, tap **No**.
  - *Run the entire part file* - To run the entire part file, do not select any commands. Then from the bottom toolbar, tap **Run** .

5. As the file runs, the software does the following:
  - The software enters [Run mode](#).
  - If you're not running the entire file, but are running from the selected command to the end, and if you're using temperature compensation in the file, a prompt appears asking if you want to execute the temperature compensation command stored in the file.
  - This occurs if you choose to execute the temperature compensation command:

If temperature compensation is enabled for the file, and at least two hours has passed since the last temperature compensation, a prompt with a 20 second countdown timer appears asking you if you've already applied the part temperature sensor. If you haven't, apply it now. Once the timer expires, or if you tap **OK**, the current process continues and compensates for temperature. If you tap **Cancel**, the process continues, but it does not compensate for temperature. For information on temperature compensation, see "[About Temperature Compensation](#)".

- Other prompts or other instructions that require your interaction appear in the bottom toolbar and at the current command in the Command list. To proceed, follow them as they appear.
  - The numbers on the DRO change dynamically to show the probe's current position as the probe is moved either manually by the jog box or automatically by the measuring machine.
  - If a feature command is in DCC mode, the probe moves and takes hits on that feature to measure it.
  - A report is generated from the measurement results.
  -
6. Once execution finishes, swipe to the right to view the **Report** screen to save or print the report.

## Reference Topics

### The Run Screen

The **Run** screen appears when you swipe to it on the bottom level of screens. It contains a Graphics window, the Commands list, and a bottom toolbar with a **Run** button.

If you tap **Run**, the software enters Run mode and begins running the commands stored in the part file. If you select a command first, PC-DMIS Touch asks if you want to start the run at the currently selected command. If no command is selected, it begins running the top-most command in the Commands list.

### Graphics Window

This window displays all of the learned features graphically. For detailed information on this window, see "[The Graphics Window](#)".

The **X**, **Y**, and **Z** values under the Graphics window show the probe's current location.

### Commands List

To the right of the Graphics window, the Commands list shows all of the commands that the software runs through. You can use it as an aid to help you follow the instructions provided when each command is processed and to know the progress of the current part file run.

You can also select a command and start execution at that command. Or, you can execute a single command in this list by tapping the small **Run**  button on the command itself.

### Bottom Toolbar

Initially, the bottom toolbar contains only this button:



**Run** - Enters Run mode and starts the run process for all the commands stored in the part file.

Once the software enters Run mode, the bottom toolbar displays instructions on what to measure. It also changes to hold these buttons:



**Stop** - Stops the run entirely.



**Pause** - Pauses the run at its current location.

**Related Topics:** [About Run Mode and Running Your Part File](#), [How to Run a Part File](#), [The Graphics Window](#)

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## Working with Reports

### About Reports

Reports communicate the status of the features in your part program, and they display the number of dimensions in tolerance and the number of dimensions out of tolerance. Getting a measurement report is the end goal behind having a part file. In PC-DMIS Touch, reports are generated automatically after each file run, and if you save a report, it is stored in report folders named after the part file name.

- For information on how to generate, preview, print, save, export, or clear a report, see "[How to Work with Reports](#)".
- For information on how to view all of the reports for a part file in a report folder, see "[How to Open a Report Folder](#)".

**Related Topics:** [The Reports Tile](#), [The Reports Screen](#), [The Print Preview Screen](#), [The Report Screen](#)

## How To

### How to Open a Report Folder

**Objective:** To open a report folder in order to manage reports.

**Concepts:** [The Report Screen](#), [The Print Preview Screen](#), [The Reports Screen](#)

**Prerequisites:** You must first run a part file to generate a report and then save that report.

#### Follow this procedure:

1. From the **Home** screen, on the **Reports** tile, tap **More Reports** to access the [Reports screen](#).
2. In the **Reports** screen, filter the list to find the folder you want.

3. Tap its name to open that folder and see all of the reports for that part file. You can then print or delete a report.

## How to Work With Reports

**Objective:** To perform various report operations such as previewing the report, sending the output to your default printer, exporting the report as a PDF file, saving the report, and clearing report data.

**Concepts:** [The Report Screen](#), [The Print Preview Screen](#), [The Reports Screen](#), [Reports Tile](#)

**Prerequisites:** You must have one of the following:

- A part file with an alignment, learned features, and defined dimensions
- A saved report

### To Generate and View a Report:

1. Open the **Report** screen by running the part file as defined in the "[How to Run a Part File](#)" topic.
2. In the **Report** screen, on the bottom toolbar, tap **Preview** to open the **Print Preview** screen.

### To Modify a Report Header's Logo:

1. Open the **Report** screen by running the part file as defined in the "[How to Run a Part File](#)" topic.
2. In the **Report** screen, on the bottom toolbar, tap **Edit Logo** to display an **Open** dialog box so you can choose a custom image to use for the header.
3. Select an image and click **Open** to use that image in the report. Supported images are: .jpg, .png, and .gif

### To Print or Export a Report:

1. Do steps 1 and 2 of the above procedure.
2. In the **Print Preview** screen, on the bottom toolbar, do one of the following:

- Tap **Print** to send the report's output to the default printer.
- Tap **Export** to save the report as a .pdf file. PC-DMIS Touch saves the report to this folder using the same name as your part file:  
C:\Users\

### To Save a Report:

1. Generate a report.
2. In the **Report** screen, on the bottom toolbar, tap **Save**. The report is saved to the report folder for that part file.

### To Clear a Report:

1. Generate a report.
2. In the **Report** screen, on the bottom toolbar, tap **Clear**.

### To View and Print a Saved Report:

If you want to print one of the four most recently saved reports, on the **Reports** tile, tap the report name. Then in the **Print Preview** screen, on the bottom toolbar, tap **Print**. If you don't see your saved report in the **Reports** tile, follow these steps:

1. To open the **Reports** screen, from the **Home** screen, on the **Reports** tile, tap **More Reports**.
2. Find the desired folder name.
3. Tap its name to open that folder and show the stored reports.
4. Tap the report name or tap **Print Preview** .
5. In the **Print Preview** screen, on the bottom toolbar, tap **Print**. The software sends the report's output to the default printer.

## Reference Topics

### The Reports Tile

The **Reports** tile shows the four most recently generated measurement reports for the most recently opened part files. Also, a number is listed next to each measurement report. This number shows how many total reports are available for the associated part file. You can tap the report name to open the selected report in the [Print Preview screen](#). You can also tap the number to the right of

the report name to open the [Reports screen](#) in the detailed report view to show all of the reports in that folder.

If you want to view a different report, access the **Reports** screen by tapping **More Reports**.

Use the **Reports** screen to see all of the report folders and choose the desired one to open.

**Related Topics:** [The Reports Screen](#), [How to Generate, View, Print, or Save a Report](#), [How to Open a Report Folder](#), [The Print Preview Screen](#)

## The Reports Screen

When you run a file, a report is generated and is shown temporarily in the **Report** screen. If you tap **Save**, the software creates a report file for that report in a folder with the same name as that file. The **Reports** screen contains all of the report folders and reports stored by the software and sorted in reverse date order.

**Note:** Do not confuse this **Reports** (plural) screen with the bottom-level **Report** (singular) screen. They are two different screens.

To access this screen, from the **Home** screen, on the **Reports** tile, tap **More Reports**.

The **Reports** screen contains these two view states:

- First, a folder view. The **Reports** screen initially displays a list of all of the report folders in reverse date order. You can use this screen to filter the folders as needed.
- Second, a detailed report view. If you tap on a folder in the folder view, the screen changes to show a detailed view of all of the reports in that folder.

You should use this screen to print and manage reports and manage report folders. You should also use it to print or delete a report that isn't one of the recent ones listed in the **Reports** tile.

### Search Feature

In the folder view state, because you haven't typed anything, the screen initially shows all the folders with saved reports. They are sorted in reverse date order.

As you type characters in the search box, the software automatically filters the folders to only show folders with names that contain the set of characters typed.

### Folder and Report Information

In the folder state, on the left side of the list, you can view each part file's associated picture. Tap a picture to show a full screen view. You can also see the folder name and the date and time the folder was last modified.

### Operations

- To open a folder, from the folder view, tap a folder name to open it and see all of the saved reports in that folder.
- To delete a folder and all its contents, from the folder view, tap **Delete**  next to the folder.
- To show the report in the [Print Preview screen](#), from the folder view, tap the report name. Then from the detailed view, tap **Print** .
- To delete a specific report: In the detailed view, tap **Delete**  next to the report.
- To return to the folder view, tap .

**Related Topics:** [The Report Screen](#), [How to Open a Report Folder](#), [How to Generate, View, Print, or Save a Report](#), [The Print Preview Overlay Screen](#)

### The Print Preview Screen

Use the **Print Preview** screen to see how a report will look if it is sent to a printer or other output device. The **Print Preview** screen differs from the normal [Report screen](#) because it shows a full print preview, including margins, page breaks, page numbers, and so on.

To access this screen, do one of the following:

- Access the [Report screen](#) and then tap **Print Preview** .
- Access the [Home screen](#). From the **Reports** tile, tap a specific report.
- Access the **Home** screen. From the Reports tile, tap **More Reports** to open the [Reports screen](#). Tap the part file for the report you need to print.

Then next to the report you want to print, tap **Print Preview**.



The **Print Preview** screen contains these buttons:



**XPS** - Saves the report as an .xps file to the My Documents directory with the same file name as your part file.



**PDF** - Saves the report as a .pdf file to the My Documents directory with the same file name as your part file.



**Print** - Sends the report to your default printer.

**Related Topics:** [How to Generate, View, Print, or Save a Report](#), [The Report Screen](#), [The Reports Screen](#)

## The Report Screen

The **Report** screen shows the most recently executed or opened report. It shows the status of each dimension defined in the part file. Out-of-tolerance dimensions appear with a red background in the dimension's header. In-tolerance dimensions appear with a green dimension header. Each dimension also contains axes that define a specific characteristic reported.

### Available Axes

- **XYZ** - The location of the point or center point of a feature or dimension.



- - Displays the Length value.



- - Displays the Width value of a slot.



- - Displays the Straightness error.



- - Displays the Flatness error.

-  - Displays the Diameter value.
-  - Displays the Radius value.
-  - Displays the Roundness error.
-  - Displays the Angle value.
- **CY** - Displays the cylindricity error.
- **TP** - Position Tolerance. This is used in Position dimensions.
- **DF** - Diameter of the feature being dimensioned.
- **LF** - Length of a slot feature being dimensioned.
- **WF** - Width of a slot feature being dimensioned.
- **M** - A generic measurement performed by the dimension. For example, for the Distance dimension, this axis represents the distance measured between two features.

As an axis moves away from its nominal position, in the graph to the right of the axis's data, a colored arrow shows the amount of deviation and the deviation direction.

The number of decimal places displayed is based on your software configuration and the current measurement units. For more information on decimal places, see "[About Decimal Places](#)".

**Note:** Do not confuse this **Report** screen with the middle-level [Reports screen](#). They are two different screens.

The bottom toolbar contains these buttons.



- Edit Logo** - This shows an **Open** dialog box so you can choose a custom logo image to use in the report's header. For more information, see "[How to Work with Reports](#)".
- Print Preview** - This shows the report in the [Print Preview screen](#).
- Clear** - This clears the current report data from the screen.
- Save** - This saves the report with the name you give it.

**Related Topics:** [The Print Preview Screen](#)



# General Reference Topics

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## About Touch Gestures

While PC-DMIS Touch functions with a standard keyboard and mouse setup, its primary audience is the touch screen user. This topic describes the supported touch gestures recognized by the software:



**Tapping** - Tapping an item with your finger accesses the touched item or performs the functionality associated with that item. Or, if the item is a box that can take letters or numbers, it displays either a keypad or a keyboard. For more information, see "[About Keyboard and Keypad Input](#)".



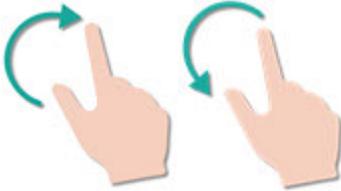
**Swiping** - Dragging your finger or fingers across your touch screen performs a "swipe". The user interface screens in this product typically have multiple options laid out horizontally on the screen. Swiping horizontally, left or right, brings new screens or functionality into view. On a vertical list swiping up and down scrolls through the list of items.



**Single Finger Drag** (Graphics window) - Dragging left, right, up, or down does a 3D rotate of the model in that direction.



**Two Finger Drag** (Graphics Window) - Dragging two fingers pans the view and moves the model in that direction.



**Two Finger Rotate** (Graphics window) - Rotating two fingers on the screen does a 2D rotate of the model in the direction twisted.



**Pinching** (Graphics window) - Pinching your fingers together zooms out on the model.



**Stretching** (Graphics window) - Stretching the your fingers apart zooms in on the model.

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## About Nominals, Tolerances, and Axes

### The One-Click Nominals and Tolerances Display Button

 The small dot to the left of a feature or dimension name acts as both an access button and an indicator. Tapping this button provides one-click access to the nominals and tolerances information, so you can edit the nominals or tolerances and enable or disable information on your report.

- If the dot is filled, at least some information from the feature or dimension is enabled to show in the report.
- If the dot is clear, information is hidden from the report.

On the **New Measurement** and **New Construction** screens, you must explicitly tap on the dot to access the nominals and tolerances information for the feature. This way you can measure features (such as alignment features) without needing to turn off the axes all the time (see "The Nominals, Tolerances, and Information" heading below).

On the **New Dimension** screen, the dot is always filled, meaning all axes, by default, are displayed when the report is generated.

Once the nominals information appears in the right pane, you can tap this button to show or hide all axes associated with that feature or dimension.

### The Nominals, Tolerances, and Axes Information

On the **New Measurement**, **New Construction**, **New Dimension**, or **Edit** screens, if you tap **Next** before tapping **Done**, or if you tap on the dot next to the feature or dimension name in the right pane, the right pane changes to show a grid of actual, nominal, deviation, and tolerance values. (Note, however, that for bestfit constructions, you must tap **Done** once and then **Next** to get to this grid). You can use this grid to edit the nominal or tolerance values or to control what axes appear on the report.

- You can tap on a value and use the on-screen [keypad](#) to change it to a new value.
- You can tap on any axis with a dot to turn it on or off. This shows or hides that axis from the report. An axis that does not have a dot cannot be turned on or off. A turned-off axis, like this one, is empty except for the nominal value and is hidden from the report:



This example shows the different components making up an axis row:

	A	B	C	D
		-84.6650		
E	X	-84.6645	-0.1000	0.1000
F		-0.0005		
		11.9200		
	Y	11.9003	-0.0500	0.0500
		0.0197		
		-11.9200		
	Z	-11.8240	-0.0500	0.0500
		-0.0960		

- A. **Axis** - In-tolerance values display in green. Out-of-tolerance values display in red. The actual axes that appear vary based on the selected dimension, feature inputs in a dimension, or measured feature.

**Available Axes:**

- **XYZ** - The location of the point or center point of a feature or dimension.
-  - Displays the Length value.
-  - Displays the Width value of a slot.
-  - Displays the Straightness error.
-  - Displays the Flatness error.

-  - Displays the Diameter value.
-  - Displays the Radius value.
-  - Displays the Roundness error.
-  - Displays the Angle value.
- **CY** - Displays the cylindricity error.
- **TP** - Position Tolerance. This is used in Position dimensions.
- **DF** - Diameter of the feature being dimensioned.
- **LF** - Length of a slot feature being dimensioned.
- **WF** - Width of a slot feature being dimensioned.
- **M** - A generic measurement performed by the dimension. For example, for the Distance dimension, this axis represents the distance measured between two features.

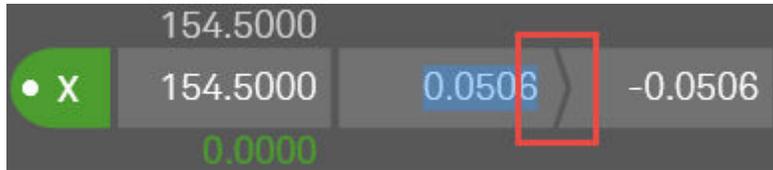
- B. **Actual/Measured** - You can edit this value.
- C. **Upper Tolerance** - You can edit this value. For information on bilateral tolerances, see the "[Bilateral Tolerances](#)" subheading below. You can specify a negative upper tolerance by tapping the - button on the keypad.
- D. **Lower Tolerance** - You can edit this value. For information on bilateral tolerances, see the "[Bilateral Tolerances](#)" subheading below. You can specify a positive lower tolerance by tapping the + button on the keypad.
- E. **Nominal** - You can edit this value.
- F. **Deviation** - You cannot edit this value. In-tolerance values display in green. Out-of-tolerance values display in red.

The number of decimal places displayed is based on your software configuration and the current measurement units. For more information on decimal places, see "[About Decimal Places](#)". Modified values are also rounded according to those decimal places.

### Bilateral Tolerances

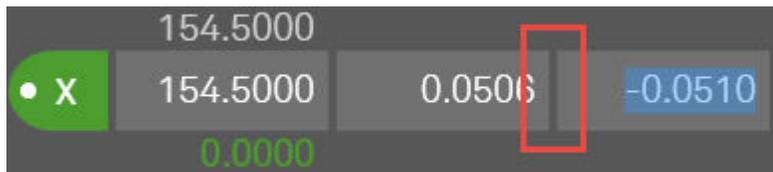
Because most bilateral tolerances have the same upper and lower tolerances, the software acts intelligently when it comes to applying bilateral tolerance values. First, once the feature is created, the software applies a default equal bilateral tolerance.

- Whenever a bilateral tolerance value is the same for the upper and the lower tolerances, or whenever there's just an upper tolerance, a small gray chevron symbol (the equal bilateral tolerance indicator) appears between the upper and lower tolerances, as indicated here:



If the equal bilateral tolerance indicator exists and you tap to change the upper tolerance value for an axis row, the software assumes you want the same lower tolerance value. It then uses the same value for both.

- If you then later tap the lower tolerance value and change it, the software assumes you don't want to have an equal bilateral tolerance. A vertical bar appears between the upper and lower tolerances to indicate that the tolerances are no longer the same:



You can re-establish the equal bilateral tolerance values by making the values the same.

**Related Topics:** [How to Learn Features to Measure](#), [How to Define a Dimension](#), [How to Construct a Feature from Other Features](#), [How to Edit a Part File](#)

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## The Commands List

The Commands list, located to the right of the Graphics window, contains all of the commands in your part file. It is sometimes referred to as a relational view because related commands are linked inside the current selected command and are shown with a smaller icon. You can tap on a related command to instantly jump to it in the list.

If you select a feature command, it is also highlighted in the Graphics window.

You can scroll through the Commands list by swiping up and down on the list. You can expand or collapse items in this list by tapping on the white triangle to the left of expandable and collapsible commands.

### Grouped Commands

Features that reside on the same surface, called a "Face" in the software, are grouped under an expandable or collapsible label for that face. By default, features that are grouped under a Face command in the Commands list are only drawn in the Graphics window when that Face command is expanded. If a Face command is collapsed, features under that Face are drawn fainter, with translucency, and without feature labels.

For information on Faces, see "[About Faces](#)".

### Relational Commands

You can use the Commands list to edit or delete features and any related references (such as dimensions or alignments). See "[How to Edit a Part File](#)". To edit or delete a command, you must first select it from the list. When you tap a command, it becomes selected and displays a detailed view of the command in a boxed area. This detailed view lists all commands that reference the selected command immediately below it. You can tap a referenced command to easily jump to it in the list.

### Filtered Commands

The software automatically filters the commands in the list when using certain functionality, such as when creating new dimensions or constructed features.

### Child Commands

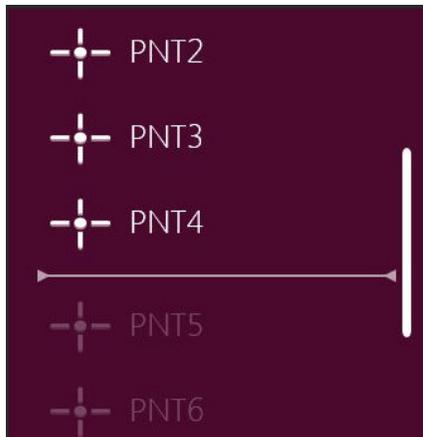
Child commands, the commands under the top-level Face command, must be deleted before a delete button becomes enabled on the parent command.

### The Insertion Point

In the **Edit** screen, the selected command indicates your current position in the part file. New commands are inserted after this selected command. This is called the insertion point. To change the insertion point, simply swipe up or down in the Commands list to the desired position and tap on an existing command. All

commands above the selected command are then "visible" for selection in other screens and operations.

When creating [Measured Features](#), [Constructed Features](#), [Alignments](#), or [Dimensions](#), you can select existing features from your Commands list or the Graphics window. Prior to accessing these screens, your insertion point must be beneath any features for them to show up for selection. On these screens, the insertion point appears as a horizontal line, like this:



You cannot change the insertion point in these screens. You must change it on the **Edit** screen.

**Related Topics:** [\[The Edit Screen \(for Editing Commands\)\]](#)

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## About the Status Icons

These status icons above the Commands list help you to quickly understand the status of your part file, the software's connection with a CMM, probe files, and so on:

### Probe Status Icons

A probe tip status icon displays the active probe tip for the selected command and the probe's current status. A green tip indicates the active probe tip.



A probe tip status icon with grayed out shanks and tips indicate they are calibrated, but not active. A white shank with a green tip indicates that the probe tip is calibrated, active, and ready for use for the current command.



A probe tip status icon with a red shank indicates an uncalibrated probe tip or a failed tip calibration.



A probe tip status icon with a yellow shank and tip indicates that the tip calibration is stale.



A probe status icon with a circle around it means that the probe command does not match the probe that is currently loaded on the CMM. The command either uses a different probe, a different angle, or both. You can tap this icon to cause the CMM to load that probe or to rotate to that angle so that it matches the active probe command in the Commands list. Any new hardware to load must already reside on the probe changer. A prompt appears, and if you click to confirm the probe change, the CMM makes the change. If it's a different probe, it loads the active probe from the probe changer. If it's a different angle, it rotates to that angle.

### Clearance Cube Status Icon



This icon appears when the ClearanceCube has been properly defined for your file. If the ClearanceCube is undefined, no icon appears. You can tap this icon to edit the existing ClearanceCube.

### DCC Mode Status Icon



This icon appears whenever you select or run a feature command that is in DCC (Direct Computer Control) motion. This occurs whenever a **DCC Mode On** command precedes a feature in the Commands list. This means the machine's movements is controlled by the computer, not by you.

**Related Topics:** [About Probe Calibration](#), [The ClearanceCube](#)

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## About Decimal Places

The number of decimal places displayed in the software's user interface and in reports is based on your current configuration of PC-DMIS Touch and the measurement units in your part file.

PC-DMIS Touch Configuration	Measurement Units	Decimal Places
Portable	Inch	Four (0.0000)
Portable	Metric	Three (0.000)
CMM	Inch	Five (0.00000)
CMM	Metric	Four (0.0000)

Angularity dimensions are the exception to the above table and always report a total of three digits after the decimal (0.000).

**Related Topics:** [Nominals, Tolerances, and Axes](#), [Report Screen](#)

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## The Settings Tile

The **Settings** tile shows the most recently accessed or commonly used settings. Tap on one of the buttons on this tile to open the selected setting or perform the action.



**Settings** - Accesses the **Settings** screen.



**Shutdown** - Exits PC-DMIS Touch.

**Related Topics:** [The Settings Screen](#)

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## The Settings Screen

The **Settings** screen contains a list settings organized by category.

### Reports

**Report Export Path** - Defines an alternate export path for the **Export** button on the **Print Preview** screen. If this is blank, this default export path is used:

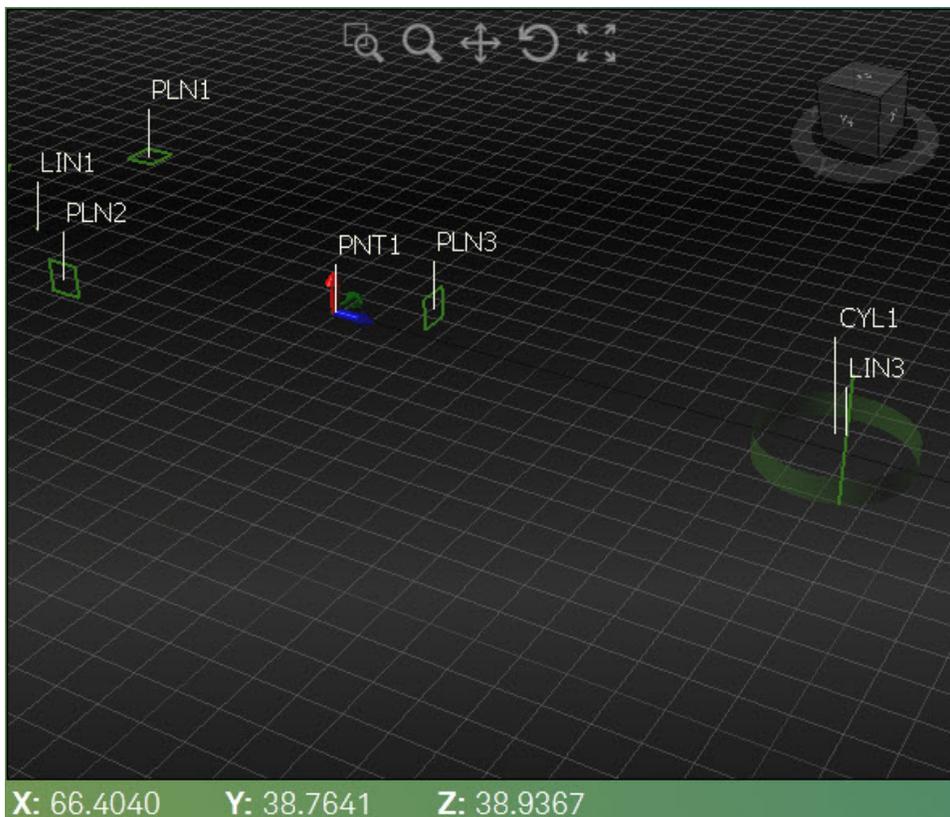
C:\users\\Documents\Touch Reports

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## The Graphics Window

When the software learns features, those features and their labels are drawn in the Graphics window. This window appears in many screens. It provides a relational view between learned features.

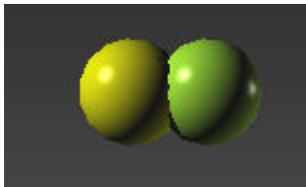
The Graphics window can draw these feature types: points, lines, circles, planes, cylinders, cones, round slots, square slots, and spheres.



### Graphics window

This window gives you an easy way to visualize, identify, and even select previous learned features. This is especially helpful when you need to choose learned features when creating alignments, defining dimensions, or constructing features.

- When you touch an item from the Commands list to select it, the software highlights the selected feature in the Graphics window. The measured results are shown in the yellow selection color. For example, in this close-up view of a single point, because it is the current selected feature, it shows the nominal (in green) and the measured (in yellow):



It then orients the view to the vector of the Face (or reference plane) associated with that feature. If you select a feature that doesn't have a Face, it orients the view so that it looks straight down that feature's axis. In other words, the view is normal to the feature's axis.

- When you touch a feature in the Graphics window to select it, or when you execute a feature, the software highlights the selected feature and automatically orients the view as discussed above to show the Face for the selected feature. It then zooms to show the Face and all other features on that Face. It also selects the feature in the Commands list.
- By default, when a Face command in the Commands list is collapsed, features grouped with that Face are hidden.

When you run a part file, the Graphics window is also useful, as it provides a visual representation of what to measure next.

### Animated Probe

The software also shows an animated probe in the Graphics window in relation to the current alignment. The probe moves in real time as you physically move the probe when learning features or during file execution.

### Square Grid

The main information portrayed by this window are any learned features drawn graphically on top of a square grid. The grid indicates an XY plane at the Z=0 height.

### Origin Symbol

The blue, green, and red origin symbol (or trihedron) shows the origin of the coordinate system and the current orientation. When you [create an alignment](#), this symbol is drawn at the alignment's origin. The colored arrows correspond to the X (blue), Y (green), and Z (red) axes. If you zoom in and out on the Graphics window, the origin symbol remains a static size.

### Orientation Cube

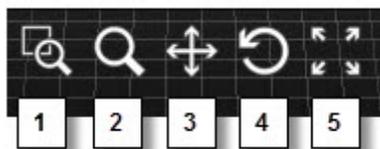
The cube in the upper-right of the window provides an alternate way of orienting the view. On the cube, or on the circular band surrounding it, tap on any face, edge, corner, or axis letter. It turns red and then rotates until that face, edge, corner, or axis is facing you. For example, tapping on the Z+ plane orients the view to show just that face.



You can also touch and drag the cube to place the window in Rotation mode. When you rotate the window, any drawn features are rotated three-dimensionally in the direction you drag your finger.

### Pan, Zoom, and Rotate Buttons

These buttons pan, zoom, and rotate the drawn features in the Graphics window:



1. **Zoom Window** - This lets you drag a small blue-shaded window over any portion of the Graphics window. The Graphics window zooms and centers any information in the window when you lift your finger.

2. **Zoom** - This places the window in Zoom mode so that when you drag the window up and down, the Graphics window zooms in and out. (You can also zoom in and out by using a mouse wheel.)
3. **Pan** - This places the window in Pan mode so that when you drag the window, the drawn features are also moved. This lets you see learned features that reside beyond of the window's current boundaries.
4. **Rotate** - This places the window in Rotate mode so that when you drag the window, any drawn features are rotated three-dimensionally.
5. **Zoom Fit** - This zooms in and out on the part as needed to fit all learned features in the window's current boundaries.

You can also perform some of these pan, zoom, rotate, operations by using your touch screen. For more information, see "[About Touch Gestures](#)".

### Digital Readout

Located beneath the Graphics window, the probe digital readout (DRO) dynamically shows the XYZ coordinate location of the probe tip as it moves through 3D space. If your probe supports angles, such as an indexable probe, the DRO also shows the probe tip command's rotated position, combined from the A axis angle and the B axis angle. For information on probe angles, see "[About Probes, Tips, and Angles](#)".

The DRO helps you know the following:

- The probe tip's current position.
- The current AB rotation angle used
- When the machine is connected and moving, since the numbers change during probe motion.

For example, you may need to use the DRO to manually move the probe to an exact XYZ position to take a hit, or you might use the DRO while manually moving the probe to verify if a created alignment is correct.

**Related Topics:** [About Learn Mode and Measuring a Feature](#), [About Run Mode and Running Your Part File](#), [About Alignments](#)

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## About Advanced Properties

This topic provides a reference to all the advanced properties on the various screens:

### ***Advanced Properties for Measured Features***

Since the initial points taken to locate and learn some features are probably insufficient for actual measurement, you may want to adjust the number of points, rows, levels and so on, and then remeasure to get better data. You can do this with the advanced properties.

In the **New Feature Measurement** screen or the **Edit** screen (for Editing Commands), you can edit feature-specific advanced properties. To show these items, from the buttons at the bottom of the screens, tap **Next** until the **Advanced** text in the left pane is highlighted.

The left pane, under the **Advanced** text, shows this information:

The **Scan** and **Touch** items determine the method used by the probe as it makes contact with the part to measure the feature. This area only becomes active if you have an alignment and have inserted a DCC command:

**Scan** - This measurement method tells the software to measure the feature without raising the probe from the surface. If this option appears, it means that both the feature and the current probe support the scan method. This uses an internal strategy to scan the feature.

**Touch** - This measurement method tells the software to measure the feature by probing discrete points. This is the default method. You can define how many points to take during its measurement. The probe raises up and moves to the new point location before making contact.

**Remeasure** - This button immediately moves the probe and remeasures the feature with the current advanced properties. During measurement, this changes to a **Stop** button that you can tap at any time to immediately stop the current measurement. Once remeasuring finishes, the software displays the measurement results on the right pane.

The right pane also changes to display one or more of these properties for supported features:

**Note:** When you first probe points to learn a feature, except for square slot features, the software remembers where you probed those points. If you don't make any adjustments to the items below, the CMM probes the feature again at those exact points. However, with the exception of the **ClearanceCube** option, if you modify any other advanced property on the right pane, the software permanently replaces your probed points with recalculated points.

For square slot features, the software *a/ways* recalculates the original probed points to avoid colliding with the part.

**ClearanceCube** - This icon appears if you have a ClearanceCube defined. It shows the current ClearanceCube face the software intends to use for the feature. By default, when you create a new feature, the software selects **Use Tip Vector**. This means when that feature runs, the software approaches and backs away from that feature using the probe shank vector used to learn the feature. You can override this and have it approach and retract using a specific alignment face, or you can turn off the ClearanceCube functionality altogether for the feature.

**Points** - This defines the number of points to measure on the feature. For circular features, the points are equally spaced between the starting and ending angles. This is only used with the **Touch** option on the left pane.

**Depth** - This defines the distance away from the surface to which the probe moves in order to probe the points for the feature. This uses the same measurement units as that of your part file.

**Points Per Level** - This defines the number of points to measure on each level of the feature. This is only used with the **Touch** option on the left pane.

**Levels** - Used with cylinders and cones, this defines how many levels of points to measure on the feature. If you see a value of 1 here, it means PC-DMIS is using your original probed point locations for the feature.

**Points (Total)** - Used by the sphere, this defines the total number of points to measure among the number of rows on the sphere.

**Points Per Row** - Used by planes, this defines the number of points to measure on each row of the feature.

**Rows** - This defines the number of rows on the feature.

- For a plane, this is only used with the **Scan** option on the left pane. It defines the number of rows to scan on the plane.
- For a sphere, there are always at least two rows of points, and all rows are positioned between the angles defined by **Start Angle 2** and the **End Angle 2**. The points on a sphere feature are not split equally among the rows.

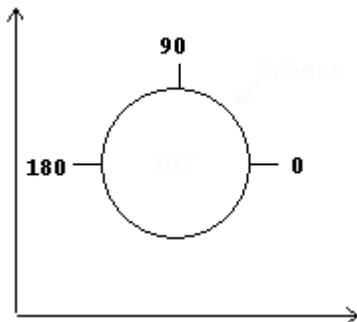
**Thread Pitch or Number of Threads** - This value appears for circle or cylinder features. It improves the measurement accuracy if you're measuring a threaded hole or stud. The label for this value changes based on the file's units of measure.

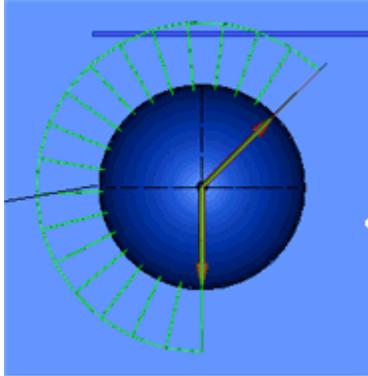
- **Thread Pitch** appears if your file uses millimeters. This is the distance between threads.
- **Threads Per Inch** appears if your file uses inches. This is number of threads in one inch of material.

If the value is anything other than zero, the software assumes a threaded hole or stud, and it staggers the feature's hits along the feature's theoretical axis, spacing them around the feature, using the **Start Angle** and **End Angle** values.

- To follow a standard (clockwise) thread pattern, reverse the starting and ending angles (for example, 720 - 0).
- To cause the measurement to reverse from a rising pitch to a falling pitch (up/down), change the value to a negative number.

**Start Angle** - This defines the angle where probing begins around a circular feature from the start point. If you rotate the view so that you look down at the feature's center, the start point of 0 degrees is on the feature's right side. Probed points are equally spaced between the start and end angles in a counterclockwise direction.

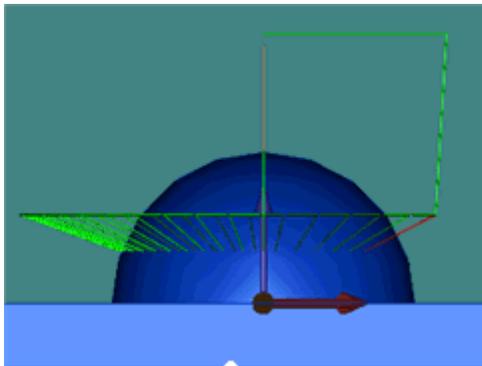




In this example sphere feature, it shows where 20 points contact the part if a Start Angle at 45 degrees and an End Angle of 270 degrees are used.

**End Angle** - This defines the angle where probing ends around a circular feature.

**Start Angle 2** - This defines the secondary starting angle on a sphere feature. If looking at the side view of a sphere, an angle of 0 degrees starts at the sphere's equator and then moves towards the top pole as the angle increases to 90 degrees and then back down to the equator on the opposite side of the sphere at 180 degrees.



Side View of the same sphere feature, showing a Start Angle 2 of 20 degrees and an End Angle 2 of 90 degrees with two rows. In this case, since the End Angle 2 is 90 degrees, it positions the second row directly on the sphere's top, providing space for only one point on the second row.

**End Angle 2** - This defines the secondary ending angle on a sphere feature.

### ***Advanced Properties for Alignments***

You can use these properties to adjust an alignment command:



- Shows the alignment's name. Tap the name to modify the name. The small dot to the left of the alignment is currently disabled.

**Level feature** - Shows the current level feature. The list next to it determines the axis tied to the level feature. The level feature establishes the orientation of the normal axis of the current working plane. The part is considered leveled to that feature.

**Rotate feature(s)** - Shows the chosen rotation feature or features. The chosen axis is rotated to this feature, or if two circles are chosen for the rotate features, to the imaginary line between the two circles.

**Rotate** - Defines the axis to rotate to the feature. The rotated axis is moved parallel to the vector of the selected feature or the line between two circles.

**Rotate about** - Defines the axis about which the rotation takes place.

**Rotation Offset** - Defines a rotational offset about the specified axis. For example, if you type 45, the alignment rotates by forty-five degrees.

**Origin offsets** - Moves the alignment origin by specific manual offset distances. For example, if you type 5 in **X**, the origin moves five units along the X axis.

**Related Topics:** [How to Learn Features to Measure](#), [The Edit Screen \(for Editing Commands\)](#), [How to Learn Features to Measure](#), [How to Edit a Part File](#), [How to Create a Part Alignment](#), [New Alignment Screen](#)

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## About Temperature Compensation

Temperature compensation compensates measurements based on room and part temperature, thus increasing accuracy of the inspection process. You can enable temperature compensation for a new part file if you create the part file using the **New File** screen and select the part material there. For more information on creating a part file, see "[How to Create a New Part File](#)".

## PC-DMIS TOUCH CMM

The software automatically attempts to compensate for temperature during these processes:

- When you calibrate a probe tip
- When you run a part file or learn a feature for a part feature that has temperature compensation enabled

### The Countdown Timer

A 20 second countdown timer appears during these times to remind you to attach the part's temperature sensor:

- Whenever you open a part file that has temperature compensation enabled
- Whenever you run a part file that has temperature compensation enabled, and at least two hours has elapsed since the last temperature compensation

If you tap **Cancel**, the timer closes and execution continues without compensating for temperature.

If you tap **OK**, the timer closes and execution continues, and compensates for temperature.

If the countdown expires, the timer closes and execution continues and compensates for temperature.

**Related Topics:** [About Probe Calibration](#), [About Run Mode and Running Your Part File](#), [The New File Screen](#)

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## The Tools Screen

The **Tools** screen contains a convenient listing of all available hardware tools and operations. You can choose to access a hardware operation from this screen.

To access this screen, from the [Home screen](#), on the **Tools** tile, tap **Tools**.

This screen is organized into these hardware categories:

### Probe Changer

This lists the current probe changer, the date it was calibrated, as well as buttons to calibrate probes; define and calibrate probe changers; and to make port assignments. For information on the buttons and their operations, see the "[The Tools Tile](#)" where these buttons are already discussed.

### Probes

This provides a sortable and filterable list of available configured probes. For information on using this list, see the "[The Probe Assignment Screen](#)" where identical controls are already discussed.

### Calibration Tools

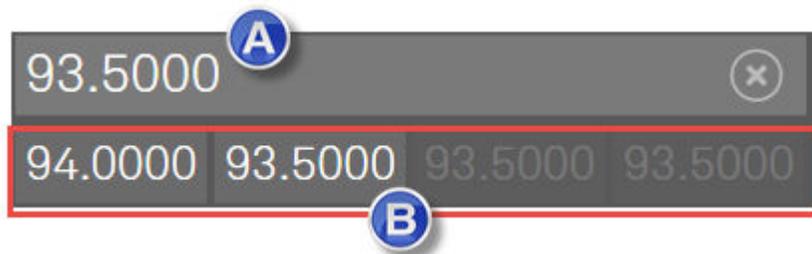
Provides a list of all available calibration tools.

---

## About Keyboard and Keypad Input

If you tap a box that accepts user input, depending on the type of field, either a complete virtual keyboard or a smaller number keypad appears as part of the user interface.

- For the full virtual keyboard, it closes once it loses focus or once you tap X on the keyboard's title bar.
- For the keypad, you can type a new value, or beneath the **New Value** box, the software provides some preset rounded values taken from the current value. You can tap on any of these rounded values to have the software use that value.



- A. The new value
- B. Some possible rounded preset values

After defining the new value, to accept a new value and close the keypad, tap the green check mark on the keypad.

### Note:

- If you tap the decimal button first, PC-DMIS Touch assumes that you want to just edit the decimal portion of the number and replaces just the value after the decimal with the newly entered numbers. You may find this useful when you need to edit a large number.
- If you tap the backspace button, PC-DMIS Touch deletes the right-most character in the **New Value** box, and if you then tap any new numbers, it appends them to the end of the value.
- If tap the + or - button, the sign of the value changes accordingly. This means you can specify positive lower tolerances and negative upper tolerances if needed.

You can also use PC-DMIS Touch with a standard desktop configuration with a keyboard and mouse.

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## Common Jog Box Buttons





**DONE** - Completes the current screen or to close a user prompt message, just like tapping **Done** on that screen. You can also use **DONE** to read the current position or to proceed through the screens (like tapping **Next**).



**ERASE HIT** - Deletes a probed point stored in memory and to go back to a previous step. If no points are stored in memory, this button closes the current screen, just like tapping **Cancel** on that screen.



**MOVE POINT** - Sends the CMM's ruby tip's XYZ information to the software as a **Move Point** command. When the software runs the file and it encounters this command, it immediately moves to the defined XYZ location.



# Glossary

## 5

**5-Way Star Cluster Probe:** A probe with five stylii arranged in a star-shaped cluster.

## A

**Active Tip:** The currently used probe tip.

**Alignment:** A command that establishes your coordinate system's location by aligning major axes with measured features. An alignment fixes your part in 3D space, aligning it with your machine.

## C

**Calibration:** Also known as qualification. This is the process used to calibrate a probe tip so that the software knows where the center of the probe tip is located in 3D space when contact is made with the part.

**Calibration Tool:** Typically this is a spherical piece of hardware on a post that you screw into the CMM's faceplate. You measure this sphere to calibrate probe tips. See Tool.

**Centerline:** An imaginary line that travels down the center of a linear feature, such as a cylinder. This is sometimes termed an axis line.

**Centroid:** The very center of a feature, usually the center of a circular feature.

**ClearanceCube:** A three-dimensional cube-like box that represents a protective envelope that surrounds the CAD model. When you do any motion in DCC mode, the probe moves to the offset distance of the box to avoid colliding with the part.

**CMM:** See Coordinate Measuring Machine.

**Controller:** This is a control box that the underlying PC-DMIS software communicates with to drive the CMM.

**Coordinate Measuring Machine:** Typically, a CMM has a base plate onto which the part is placed for inspection. Either the vertical ram moves the probe, or the table moves underneath the probe. Some CMMs have horizontal or gantry configurations measuring larger parts.

## D

**Datum Feature:** A feature on the part identified by a datum reference symbol on the print.

**DCC Mode:** A software mode that runs commands under Direct Computer Control. See Direct Computer Control.

**Digital Readout:** A portion of the user interface showing the current XYZ location of the probe tip.

**Direct Computer Control:** If commands are in this mode, the CMM's motion is automatically controlled by the computer without user intervention.

**DRO:** See Digital Readout.

## E

**Edit mode:** A command placed in this mode can be modified. You must tap the small pencil icon next to a command to place it in Edit mode.

**Execution Mode:** See Run Mode

## F

**Face:** A surface, defined as a plane or an alignment face, onto which certain two-dimensional features are projected once measured. A Face command is required in the part file before you can measure those features.

**Feature of Size:** A surface or set of opposed parallel surfaces that are associated with a size dimension, such as a sphere, a cylinder, a notch, and so on.

**File:** See Part File.

**Form Dimension:** Any dimension that reports on a feature's shape or form. These include: Straightness, Flatness, Circularity, and Cylindricity

## H

**Hit:** See Point.

## I

**Insertion Point:** This is where new commands are inserted into the Commands list.

## J

**Jog Box:** The jog box is the hand held box with a joy stick that moves the CMM.

## L

**Learn Mode:** A mode where the software can learn what features you want it to measure. You determine the features to learn by probing hits on different

features when in learn mode. Learned features are stored in files called part programs.

**LMC:** Least Material Condition. The condition of a feature where it uses the least amount of material. A hole at LMC will be at its largest, whereas a stud at LMC will be at its smallest.

## M

**Manual Mode:** A software mode that waits for you to manually drive the machine with the jog box to perform the motion or measurement.

**MMC:** Maximum Material Condition. The condition of a feature where it uses the most amount of material. A hole at MMC will be at its smallest, whereas a stud at MMC will be at its largest.

## P

**Part File:** A file containing the instructions needed to measure the learned features stored in the file. Also termed a "part program".

**Point:** A recorded XYZ location of the probe tip against some surface. Also known as a "hit".

**Probe Changer:** This is sometimes called a "tool rack" or a "tool changer". It is a rack that is positioned next to the CMM and contains different probe tips for different measurement applications. These probe tips can be swapped in and out with Tip Change command. The LSPX1C is the type of probe changer used by PC-DMIS Touch.

**Probe Tip:** A probe stylus and ruby ball used to take points on a part.

## Q

**Qualification:** See Calibration.

## R

**Ram:** The vertical arm of the T-Gage machine that holds the currently used probe body.

**RFS:** Regardless of Feature Size.

**Run Mode:** In this mode, the software measures learned features.

## S

**Shank:** See Stylus.

**Stylus:** Also termed a "shank". This is a cylindrical metallic body into which the ruby ball is attached.

## T

**Tile:** A grouping of the most recently selected or most popular items. They appear on the Home screen for quick access.

**Tool:** A "tool" is another term for a calibration sphere. See "Calibration Tool".

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