



## **PC-DMIS Touch 2013 - Portable User Manual**

April 25, 2014



(For use with a ROMER Absolute measurement arm)



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## PC-DMIS TOUCH Portable

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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 the ROMER Absolute Arm.

Before you can use PC-DMIS Touch, you need to install it onto your computer. The installation is controlled by License Management Software (LMS).

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

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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", "flick", "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

## PC-DMIS TOUCH Portable

- [Working with Part Files](#)
- [Measuring Features](#)
- [Creating an Alignment](#)
- [Working with Faces](#)
- [Defining Dimensions](#)
- [Constructing Features](#)
- [Running Your Part File](#)
- [Working with Reports](#)

### **General Reference Topics**

- [About Nominals, Tolerances, and Axes](#)
- [The Commands List](#)
- [About Decimal Places](#)
- [About Touch Gestures](#)
- [The Graphics Window](#)
- [About Advanced Properties](#)
- [The Tools Tile](#)
- [The Tools Screen](#)
- [About Keyboard and Keypad Input](#)
- [About Portable Arm Buttons](#)
- [About Remote Mouse](#)

# 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:**

## PC-DMIS TOUCH Portable

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.

## 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 flick 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 [flick](#) (or scroll) 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 flick to the right, 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:

## PC-DMIS TOUCH Portable

- **Top Level** - The top level, known as the **Home** screen, contains these tiles: [New tile](#), [Files tile](#), [Reports tile](#), and [Tools tile](#). Flick 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](#). Flick 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 **Up**  from any middle level screen.
- **Bottom Level** - The bottom level contains these screens: [Edit screen](#), [Report screen](#), and [Run screen](#). Flick left or right as needed.

In addition, some screens lie on top of other screens. These are those screens: [New Dimension screen](#), [New Construction screen](#), [New Alignment screen](#), [New Measurement screen](#), [Print Preview screen](#), [Edit screen](#) (for commands)

### 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](#)

## 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 connects to the device and displays the [Home screen](#) in the currently selected language.

**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.

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

- [New](#) - Creates a new part file
- [Files](#) - Runs one of the four most recently accessed part files
- [Reports](#) - Opens one of the most recently viewed reports
- [Tools](#) - Opens one of the most recently accessed or most popular tools

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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 flick 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 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:* Type the **Revision Number** and **Serial Number**.
      - *Required:* Choose the **Measurement Units** to use, either **Inches** or **Millimeters**.
      - *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. When finished, tap **Done**, or press the yellow button on your measurement arm. 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.

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

## 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:

- **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.
- **Measurement Units** - This determines the type of measurement units to use in the part file (inches or 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 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.

**Related Topics:** [The New Tile](#), [The Files Tile](#), [The Files Screen](#)

## 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 flick 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 in a predefined directory. The software automatically filters the files to show only the files 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 edit a file, tap its name. The [Edit screen](#) opens. From there you can edit the part file.
- 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 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.
-  [Face](#) - 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.
-  [Measurement](#) - Learns a new feature. It opens the **New Measurement** screen.
-  **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](#)

## 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](#)".

**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](#).

#### 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 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".

- 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 **New Face**:



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.

- Select/Measure.** For each point you take, move the probe to the feature, carefully touch the feature with the probe tip, and press the measure button on your arm (see "[Portable Arm Buttons](#)"). 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.
- 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 the white button on your arm.
- If you want to override the guessed feature, in the right pane, tap a valid feature type.
- Once the points are taken, the screen changes to show the feature's name.

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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 the yellow button on your measurement arm 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 the yellow button on your measurement arm to proceed.
15. 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. You can also press the white button on your measurement arm to go back to the previous step.

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

16. 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 white button on your measurement arm.
17. If you're satisfied with the measurement, tap **Done** to accept the measurement and close the screen. Or, press the yellow button on your measurement arm.

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.
- **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 also shows any points taken during the active measurement. When the measurement finishes, the software shows a drawing of the newly learned feature. For more information, see "[The Graphics Window](#)".
- **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](#)".

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](#)

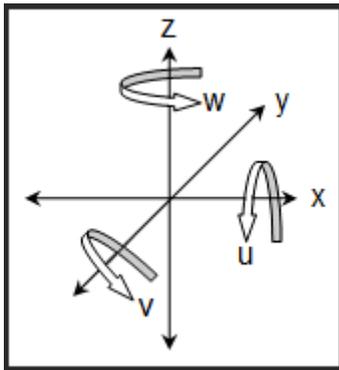
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## Creating an Alignment

### 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).**

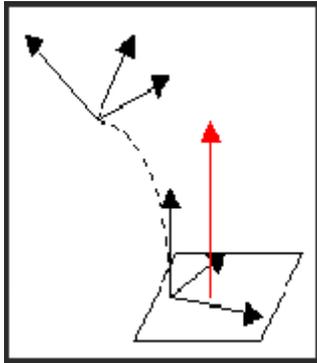
To establish an alignment, you must measure three valid features termed "alignment features" and then use those features to define the X, Y, and Z origin. These alignment features "align" each major axis to a selected feature, thereby constraining the six degrees of freedom, and fixing the part in 3D space.

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

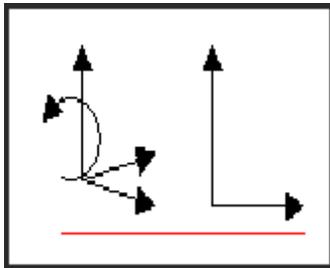
### A Sample Alignment

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

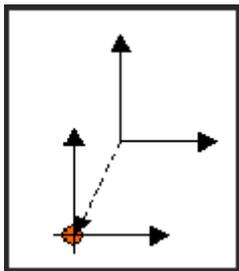
**Plane** - An axis is associated with the normal vector of the plane. This determines the orientation of that axis, and the part is considered leveled to the selected plane. This constrains two degrees of rotation:



**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 the yellow button on your measurement arm 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 to be provided here once implemented>

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

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- 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 white button on your measurement arm.
  6. If you're satisfied with the alignment, tap **Done** to accept the alignment and close the screen. Or, press the yellow button on your measurement arm.

**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-).

- 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.

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- 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.

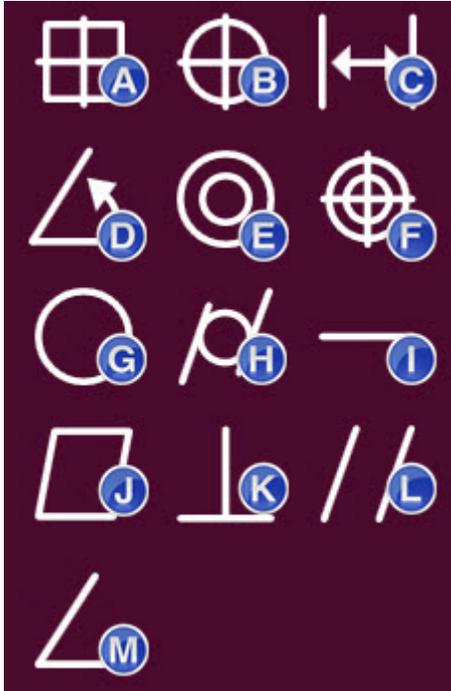
**Note:** Since most users want to measure items on their alignment's level plane, a Face from the plane used to level the part is automatically inserted into the Commands list immediately after any newly created alignment. For example, if you use the **Xplus** plane as your alignment's level axis plane,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

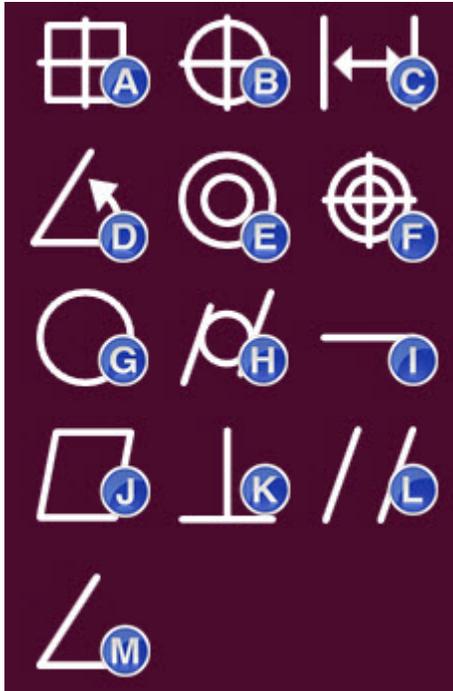
1. Ensure that the insertion point is where you want it in the Commands list by tapping on the desired command. For more information on the insertion point, see "[The Commands List](#)" topic.
2. Access the **New Dimension** screen. You can do this by doing one of the following:
  - Create a brand new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

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- b. If the needed feature does not exist, measure it now. To measure, probe hits on the part with your measurement device. See "[Learning Features to Measure](#)" for more information. Once the feature is measured, it becomes selected.
  - c. Continue selecting input features or options until you have everything you need for your selected dimension.
  - d. 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**.
6. 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
7. 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".

8. 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**.
9. Tap **Done** to proceed.
10. If you're satisfied with the dimension, tap **Done** to accept the dimension and close the screen. Or, press the yellow button on your measurement arm.

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.

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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](#)".

**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<br>D (Diameter based on the face)  |
| Cone     | XYZ<br>A (Angle)   |
| Cylinder | XYZ<br>D (Diameter)<br>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<br>T (for points on curved surfaces, this is the deviation along the approach vector)                                      |
| Slot     | XYZ<br>D (Diameter)<br>R (Radius)<br>L (Length)  |

|        |                     |
|--------|---------------------|
| Sphere | XYZ<br>D (Diameter) |
|--------|---------------------|

Location dimensions are constructed automatically whenever a feature is measured.

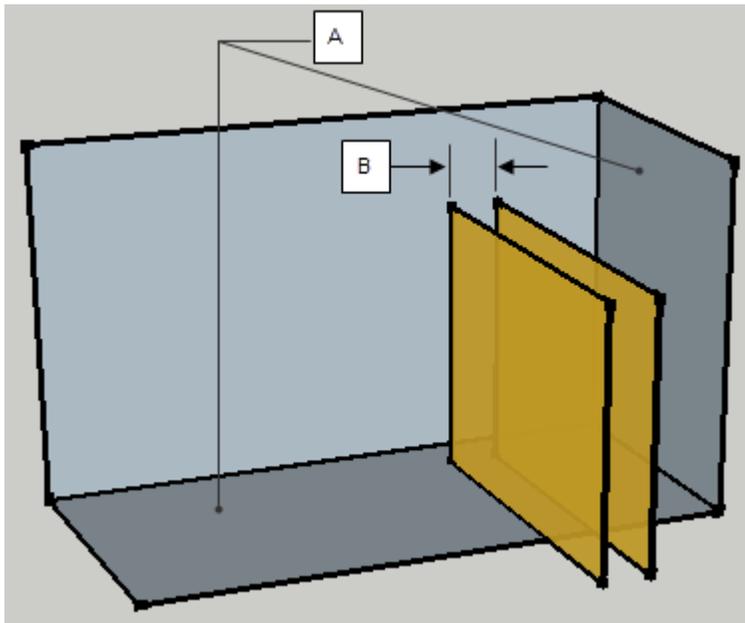
## Position

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.

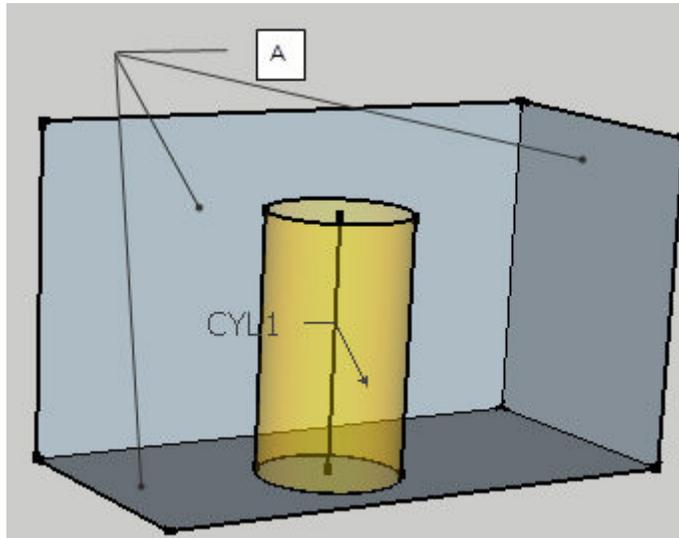
A measured value of 0 for a Position dimension means the feature has the perfect location and position. The tolerance value defines the amount of deviation away from 0 that is allowed for the dimension to still be in tolerance. The tolerance value defines the tolerance zone. Common tolerance zones for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

- The area between two parallel planes:



This diagram shows a tolerance zone from two parallel planes (B) and the datum planes (A).

- A cylindrical area shown in CYL1 below:



**This diagram shows a cylindrical tolerance zone (CYL1) and the datum planes (A).**

The tolerance controls the location. It also indirectly controls the form and orientation.

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

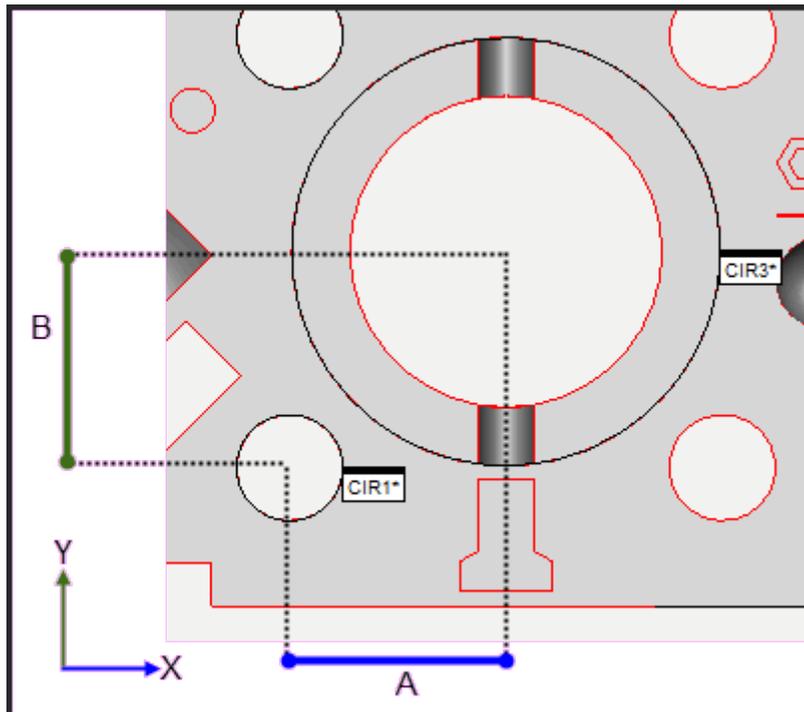
These determine what solution to use for a distance dimension. You can choose a 3D solution by choosing **3D**. Or you can choose a 2D solution by choosing one of the axes options (**X**, **Y**, or **Z**) and calculate a distance parallel to the chosen axis. Since 2D distance projects the features onto 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.

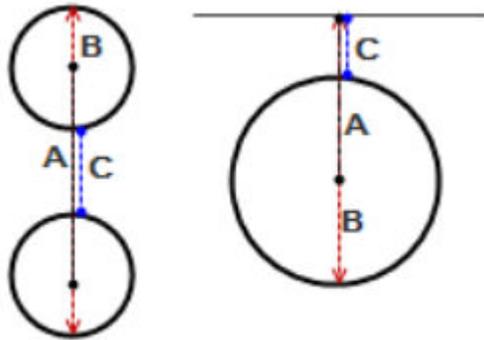


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For example, in the above diagram, the distance labeled A is the distance in X (or parallel to X) between CIR1 and CIR3, and the distance labeled B is the distance in Y (or parallel to Y).

### 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:

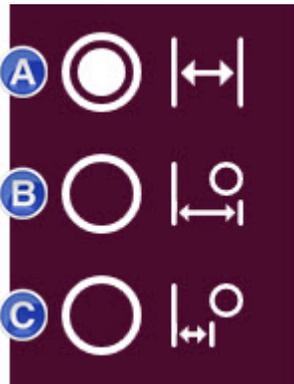


**A** - Normal calculated distance between features (black line). The radius is ignored. This is the default.

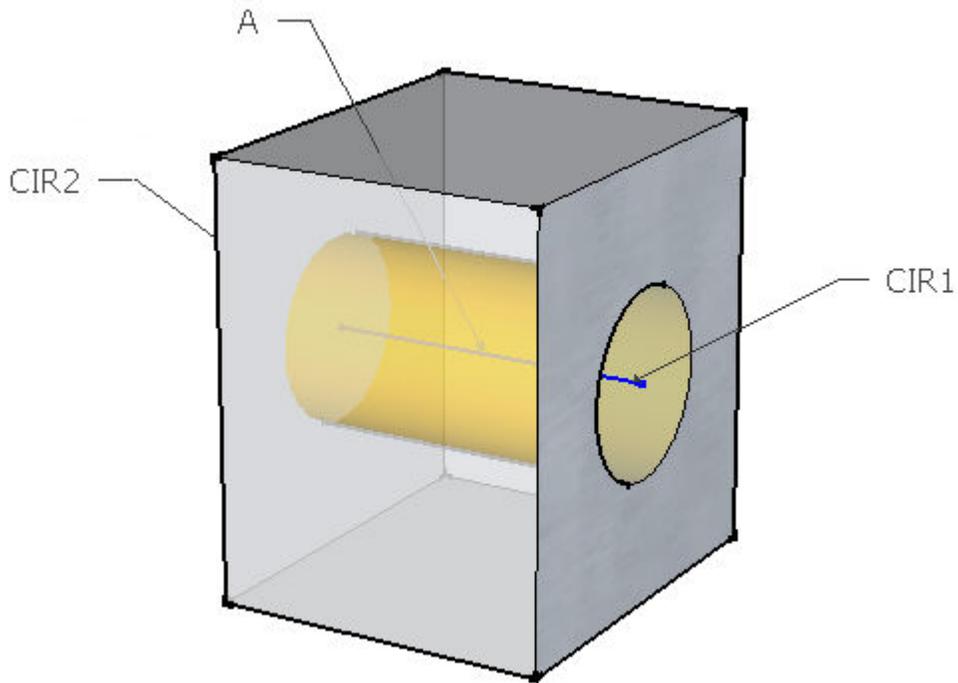
**B** - Add Radius (red line)

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

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



### Example

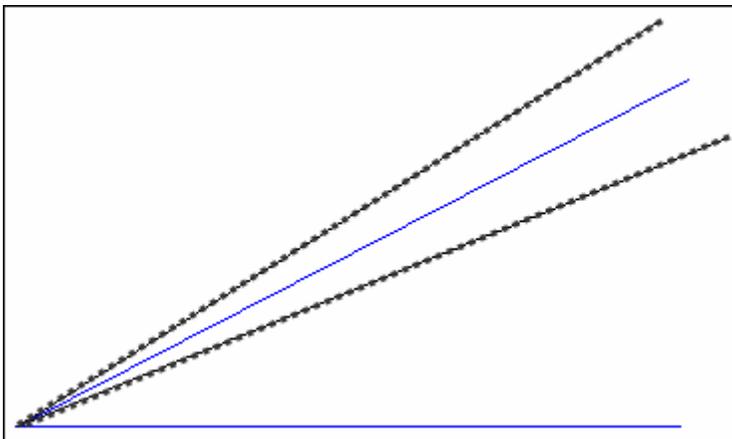


**A** - The reported dimension is the distance between the centers of the two circles (CIR1 and CIR2) at opposite ends of a cylinder.

### 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.

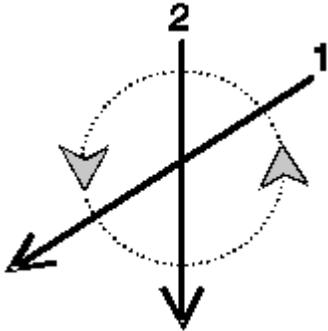


## PC-DMIS TOUCH Portable

**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:



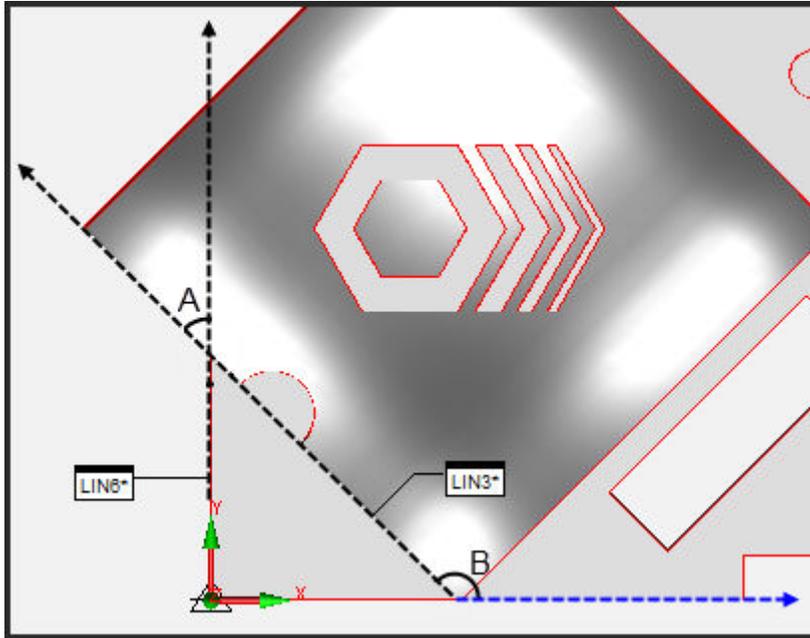
### Angle to an Axis

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

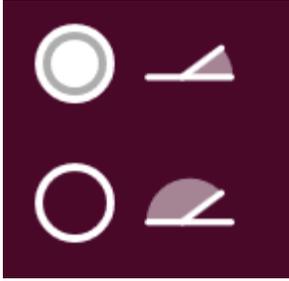
This determines how to calculate the angle between two features. Angle dimension can either be two-dimensional or three-dimensional. If your active face (the selected face or the face for a selected command) is an [axis plane](#), you can choose between either option. If the active face is a measured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

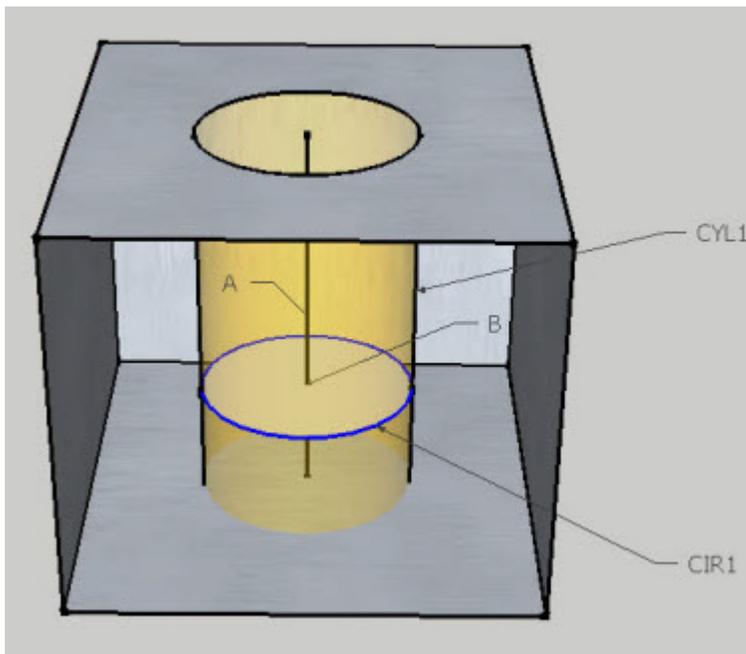
If you need an alternate or complementary angle, under **Angle Solutions**, you can tap the specific angle you want reported:



## 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.

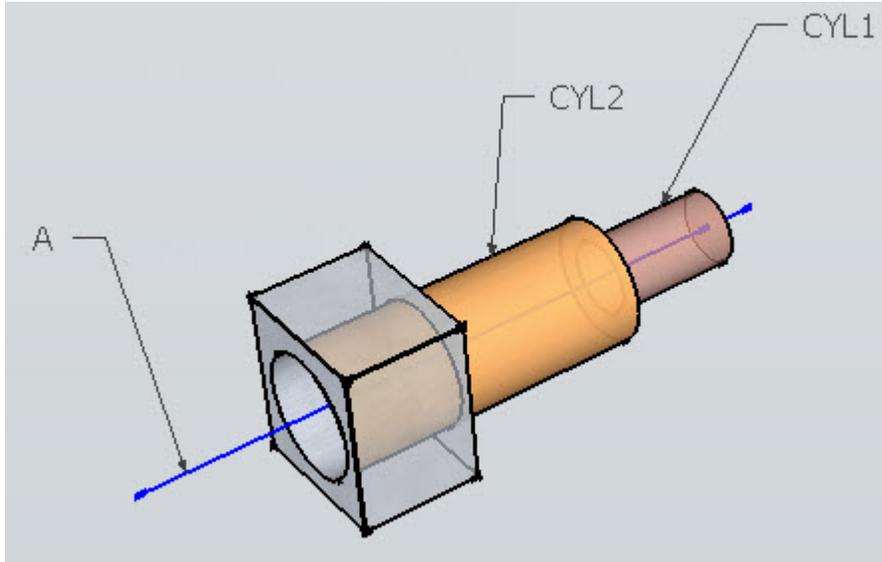
### Example



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.

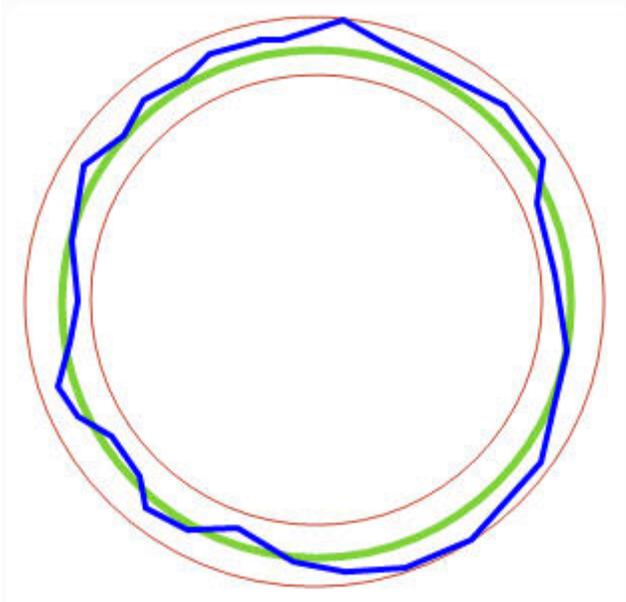


**In this diagram, Coaxiality can test if the axes of the two cylinders, CYL1 and CYL2, share the same center line (A).**

## 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.

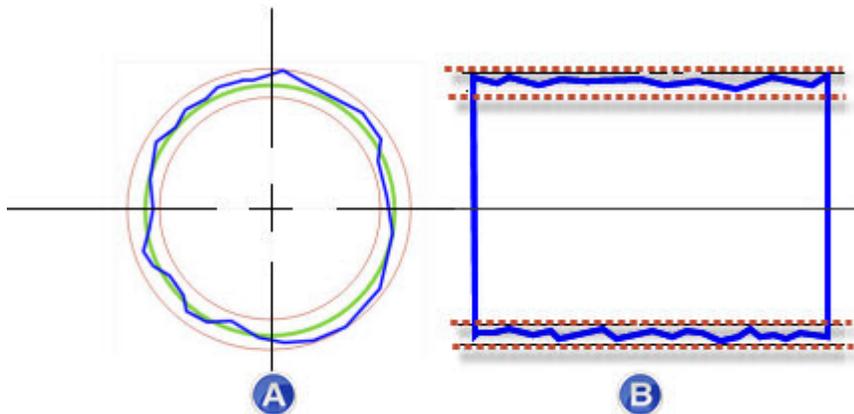
The tolerance value for circularity provides a tolerance band around the feature as shown here:



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.

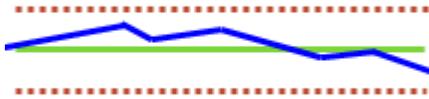


In this diagram of a cylinder, A is the top view and B is the side view. The solid brown lines in A and the dotted lines in B represent the tolerance bands. The blue line represents the measured cylinder and illustrates cylindricity.

## 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.

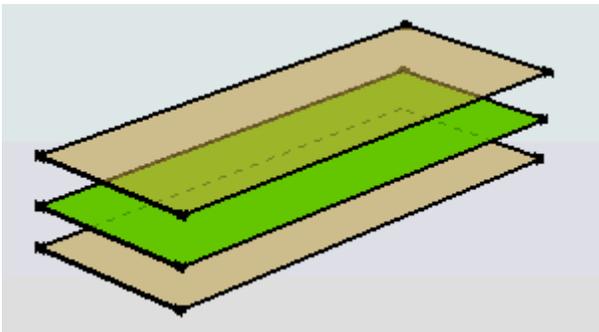
The tolerance value creates a tolerance zone around the line feature or around the center line of a cylinder or cone feature.



**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.

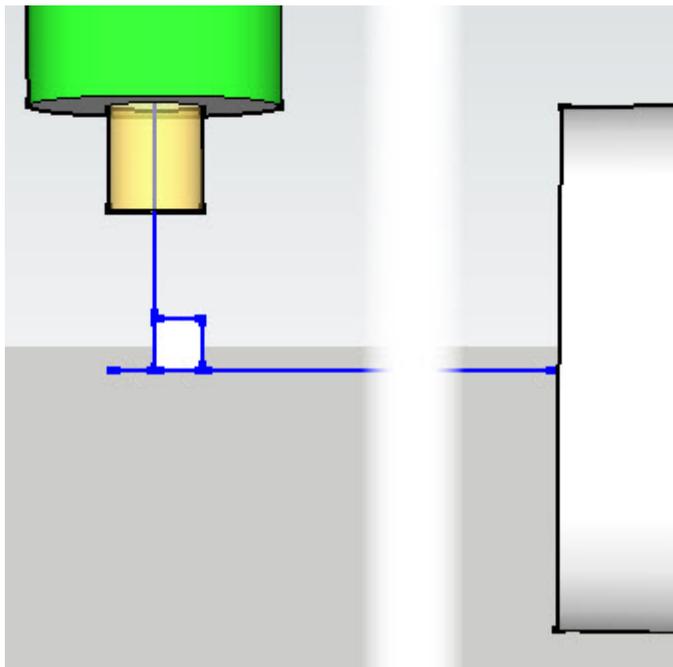
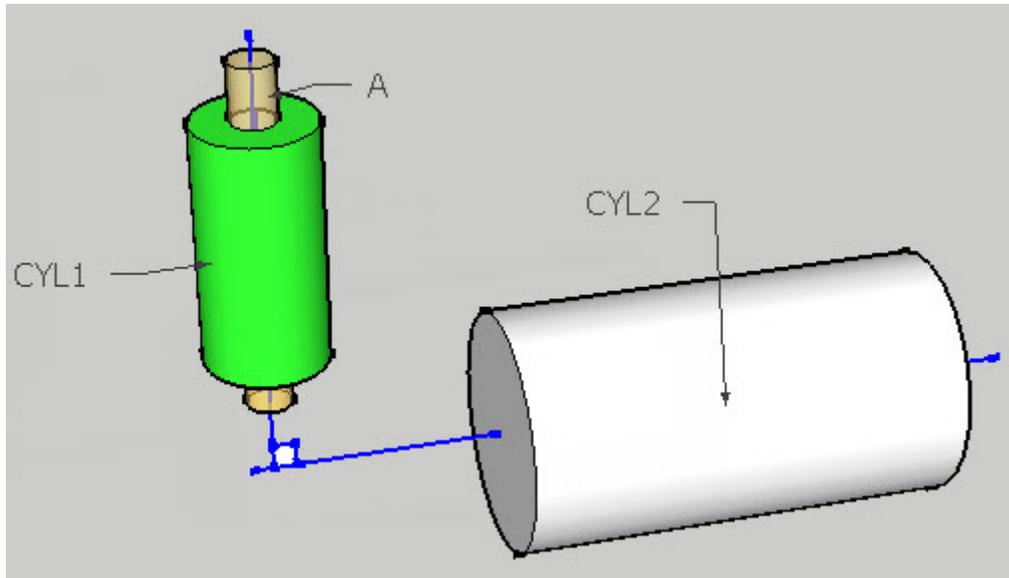


**In this diagram, the green plane represents a perfectly flat plane. The tolerance zone is represented by the translucent planes above and below it.**

### 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 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.

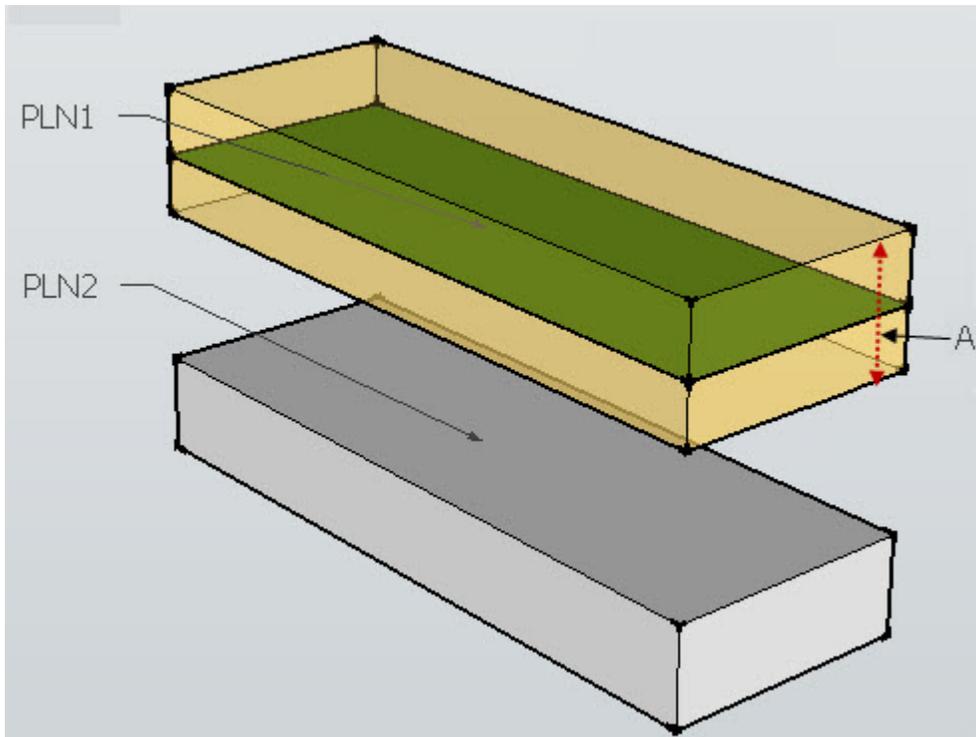


These diagrams illustrate perpendicularity between two cylinder features. CYL1 is the first input cylinder and CYL2 is the second cylinder. The angle between the two center lines is compared against a perpendicular angle. The tolerance zone around the center line of CYL1 (A). The cylinder's centerline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



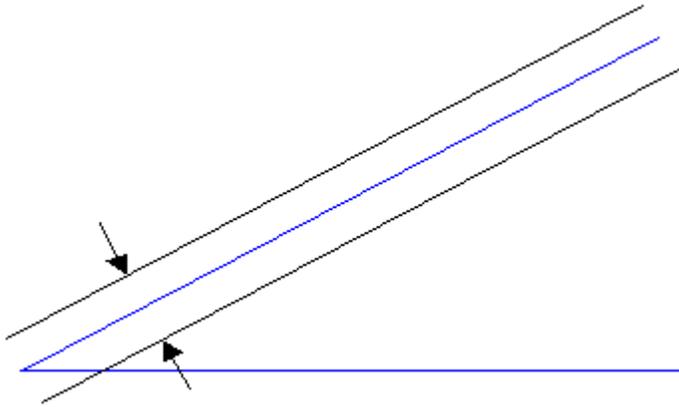
**This diagram illustrates parallelism between two planes, PLN1 and PLN2. PLN1 is the first selected feature. PLN2 is the datum feature. The tolerance zone around PLN1, indicated by the translucent yellow box (A), shows the area that the plane being evaluated must lie within for the parallelism to pass.**

## 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**:



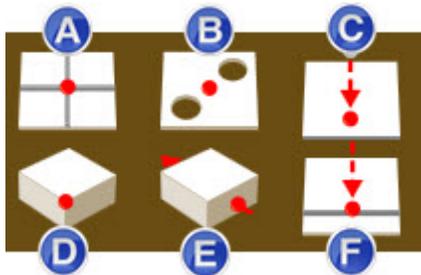
- Edit an existing part file and then from the bottom toolbar tap **New Construction**:



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.

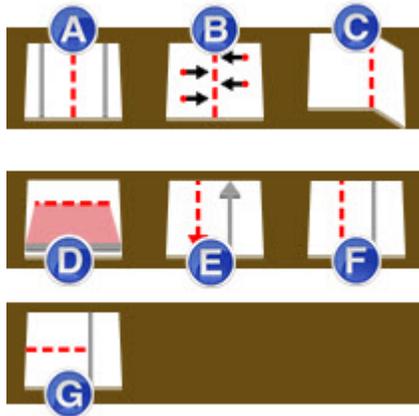
3. For **Type**, from the right pane, tap the type of feature to construct.
4. For **Method**, from the right pane, tap the method to use for the feature construction. If only one method is available for a type, it is automatically selected.

**Point Construction Methods**



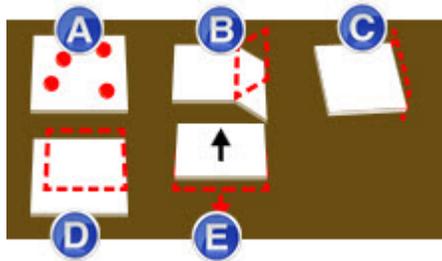
- A. Intersection Point
- B. Middle Point
- C. Projection Point
- D. Corner Point
- E. Pierce Point
- F. Drop Point

**Line Construction Methods**



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

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- 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 the yellow button on your measurement arm.

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](#)".

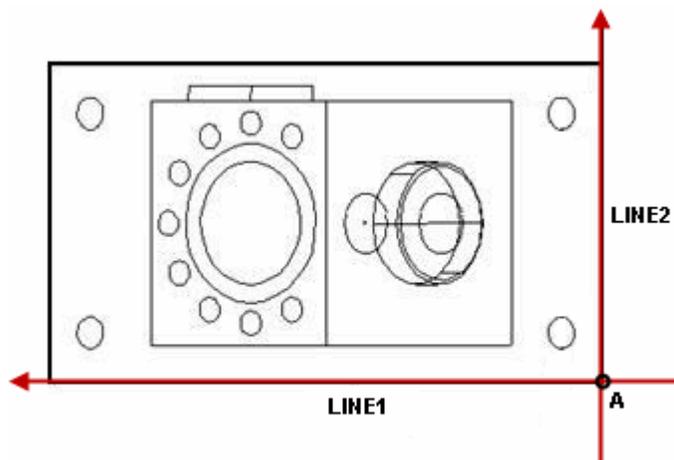
**Related Topics:** [Constructing Features to Measure](#), [Constructed Features](#)

## Point Construction Methods

### Intersection Point

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

#### Example

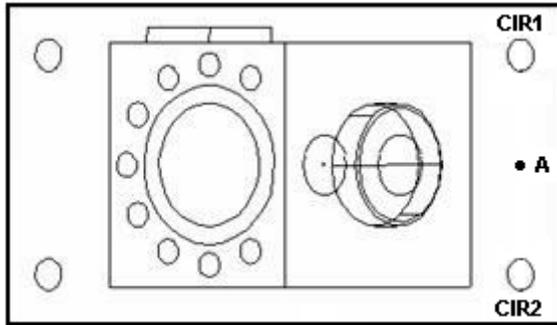


A - Point constructed from two lines (LINE1 and LINE2).

### Middle Point

This creates a point midway between two reference features.

#### Example

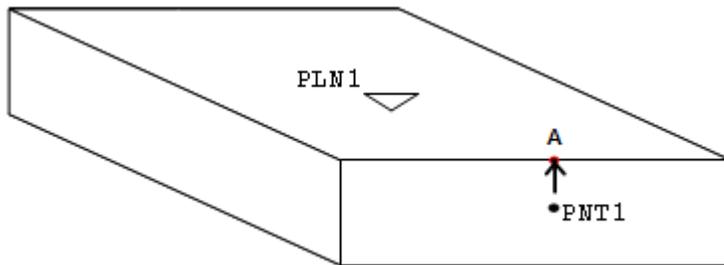


A - Point constructed midway between two circles, CIR1 and CIR2.

### Projection Point

This projects a point onto the selected plane feature.

#### Example

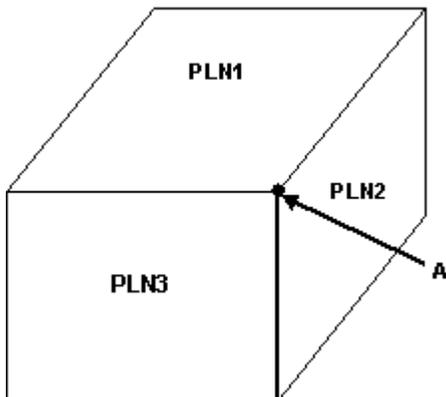


A - Projected point constructed from a point, PNT1, and a plane, PLN1.

### Corner Point

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

#### Example

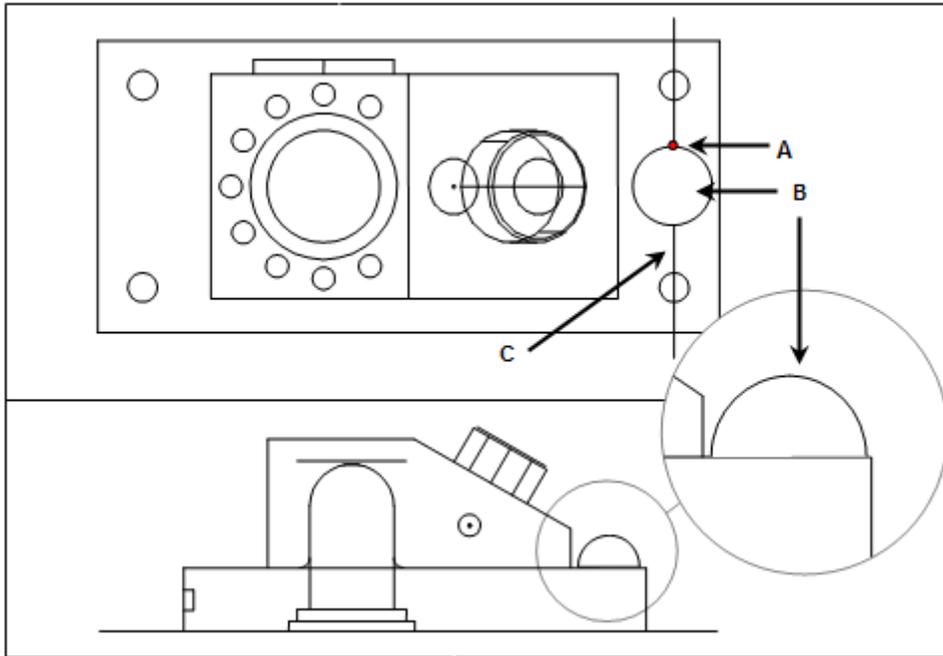


**A** - Point constructed at the intersection of three planes (PLN1, PLN2, and PLN3).

### 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.

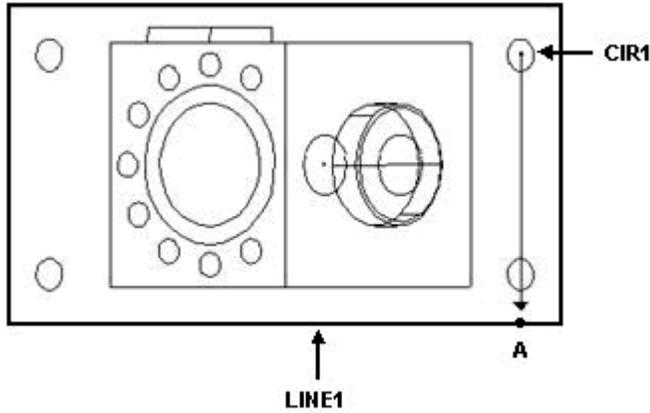
**B** - Sphere feature.

**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



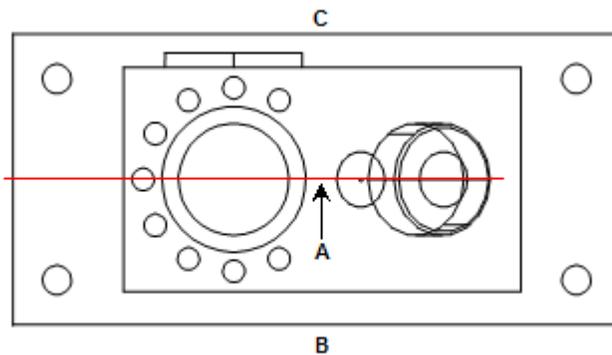
A - Point constructed by normally projecting circle, CIR1 (the designated point), to line, LINE1.

## 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).

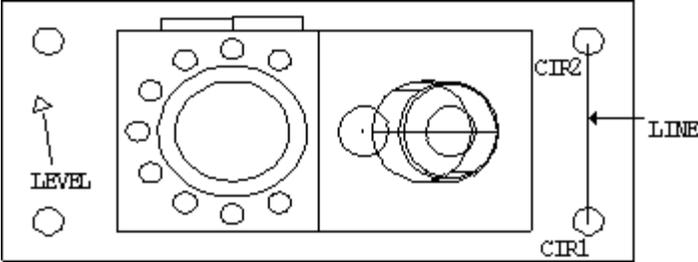
B - Front of part

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.

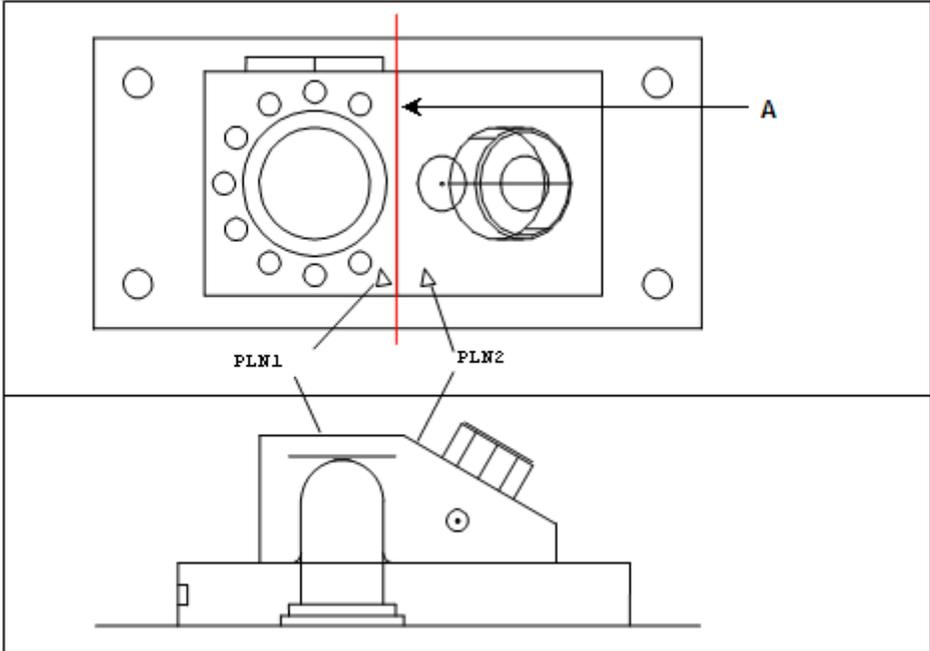
**Note:** If you only use point features as your input features in a best fit construction, the software compensates the constructed feature by the probe tip's radius. This means the construction behaves as if each point had been registered at the very center of the probe tip.

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

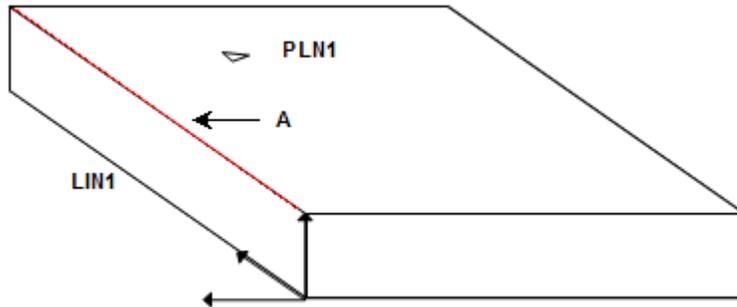


A - Line constructed at the intersection of two planes (PLN1 and PLN2).

## 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).

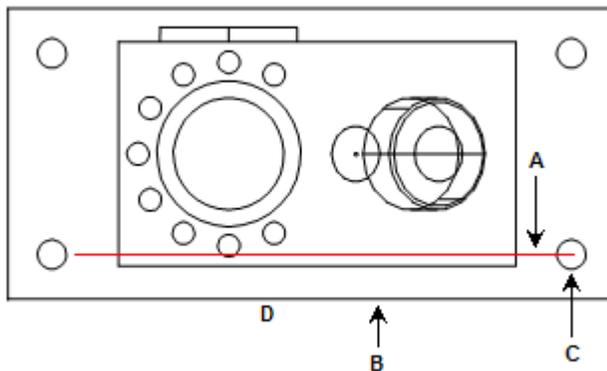
## Reverse Line

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

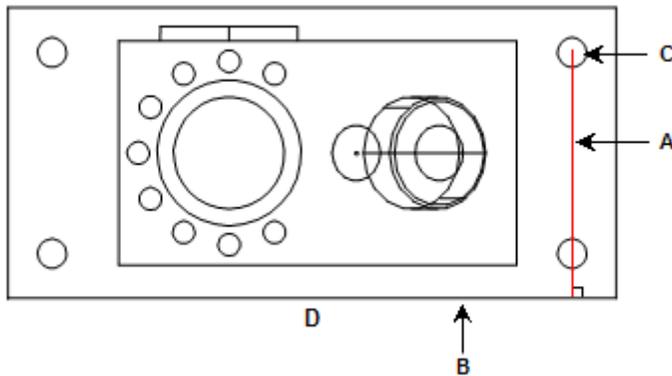
C - Second feature

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

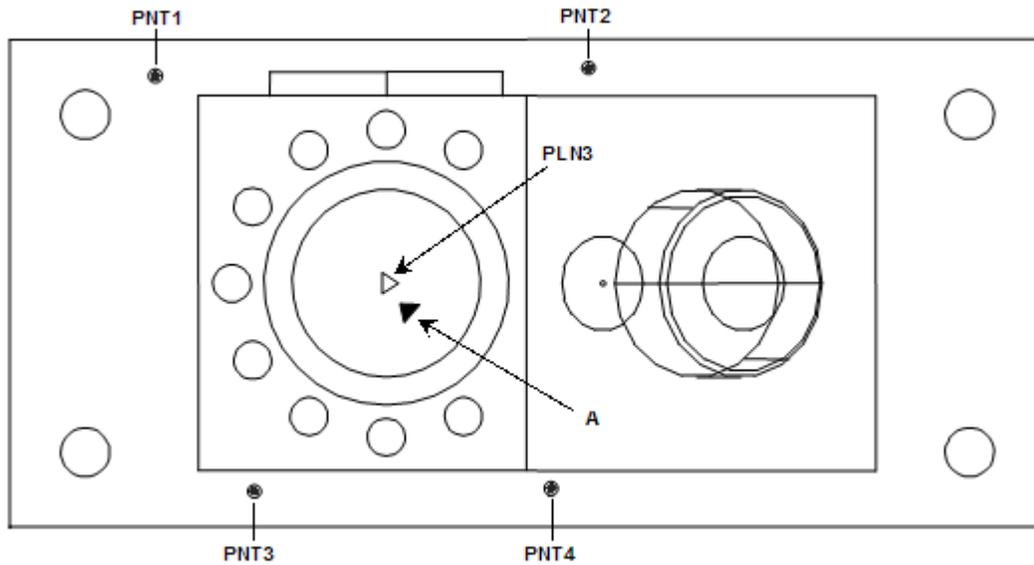
## Plane Construction Methods

### 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

## PC-DMIS TOUCH Portable



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

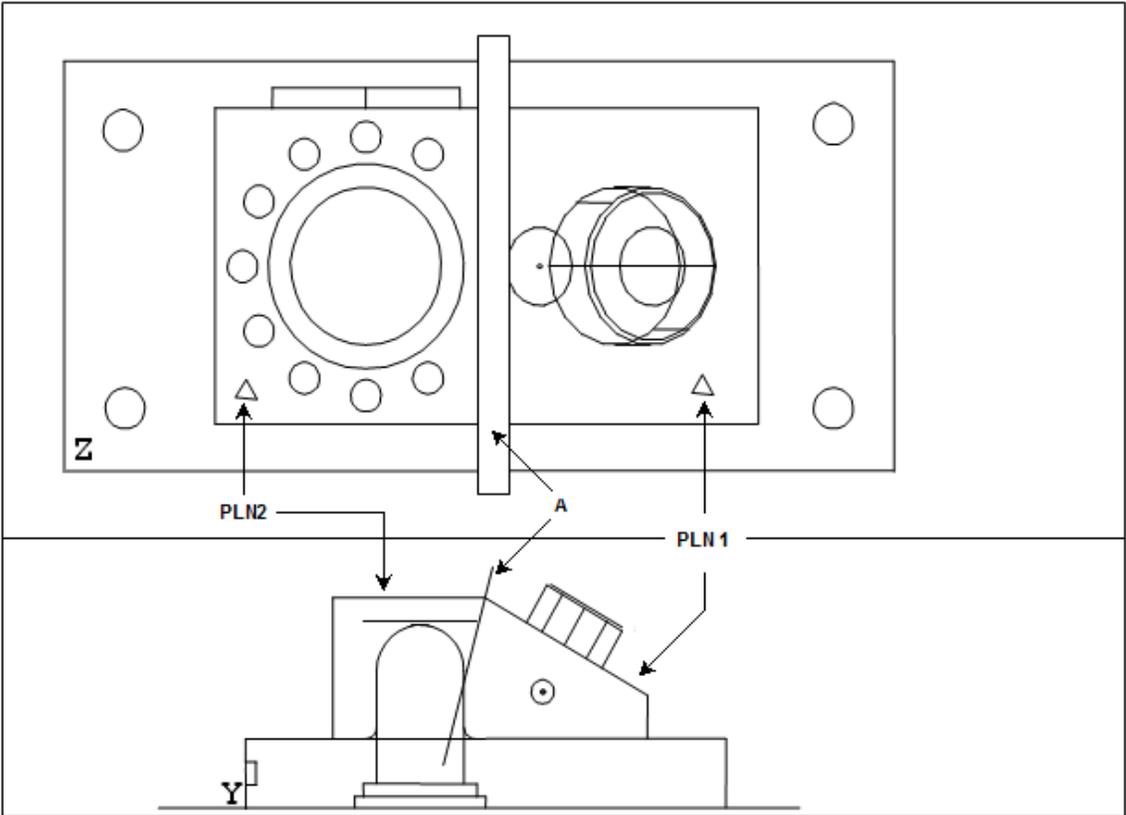
**Note:** If you only use point features as your input features in a best fit construction, the software compensates the constructed feature by the probe tip's radius. This means the construction behaves as if each point had been registered at the very center of the probe tip.

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



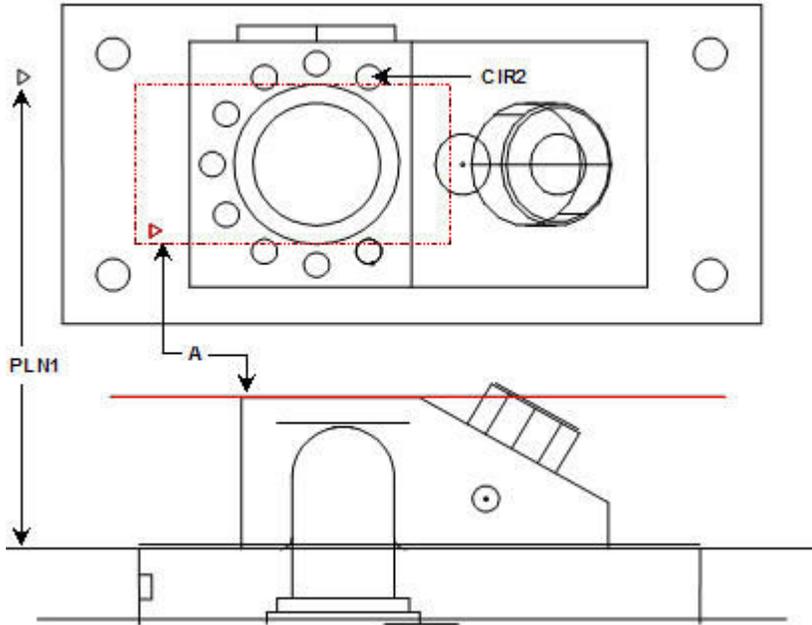
A - Constructed plane equally spaced between two planes (PLN1 and PLN2).

**Parallel Plane**

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

**Example**

## PC-DMIS TOUCH Portable

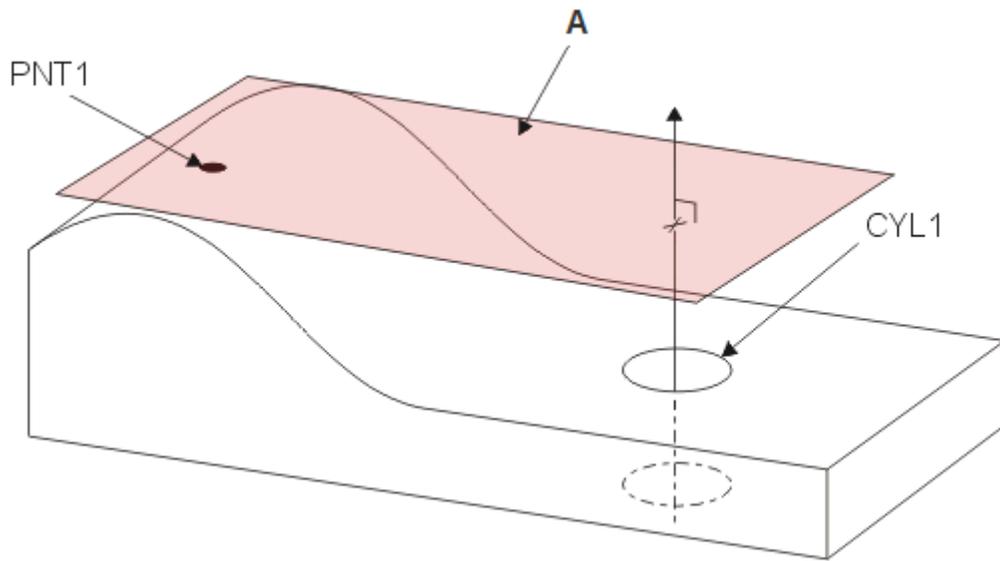


**A** - Plane constructed parallel to the first feature, a plane shown as PLN1, and through the second feature, a circle shown as CIR2.

### 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**

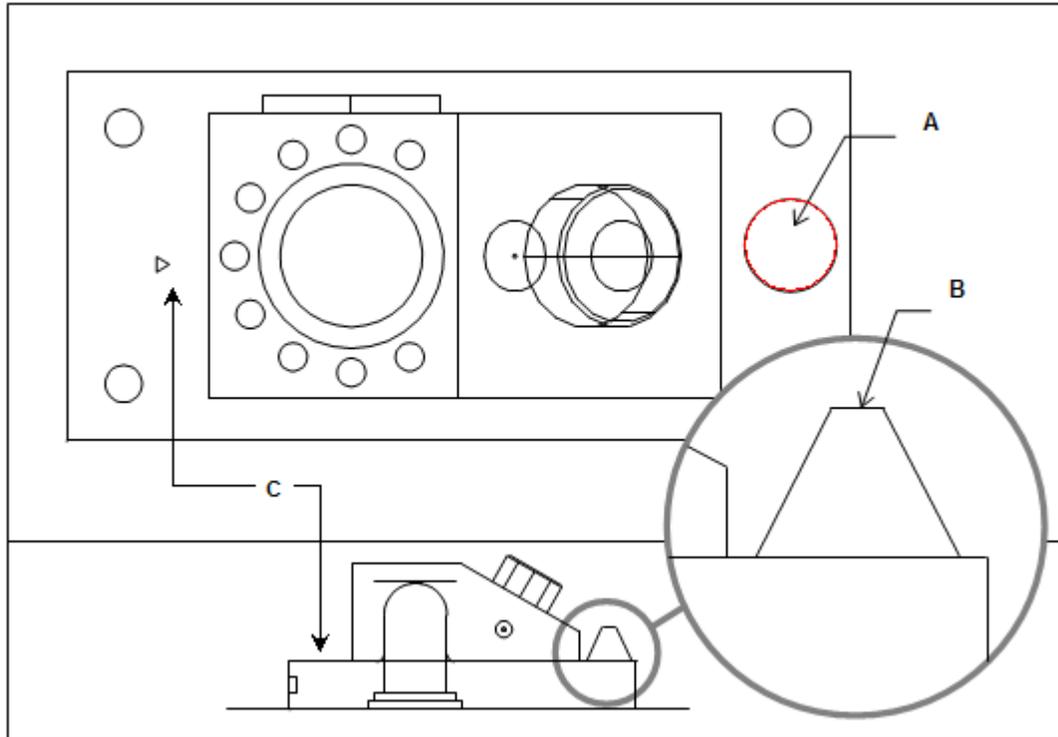
This creates a plane with a reverse vector from the selected plane feature.

## **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.

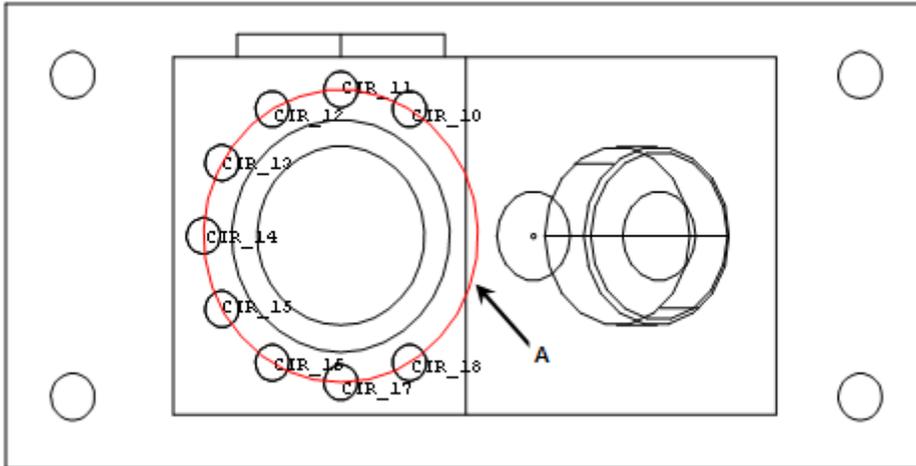
**B** - Cone feature

**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).

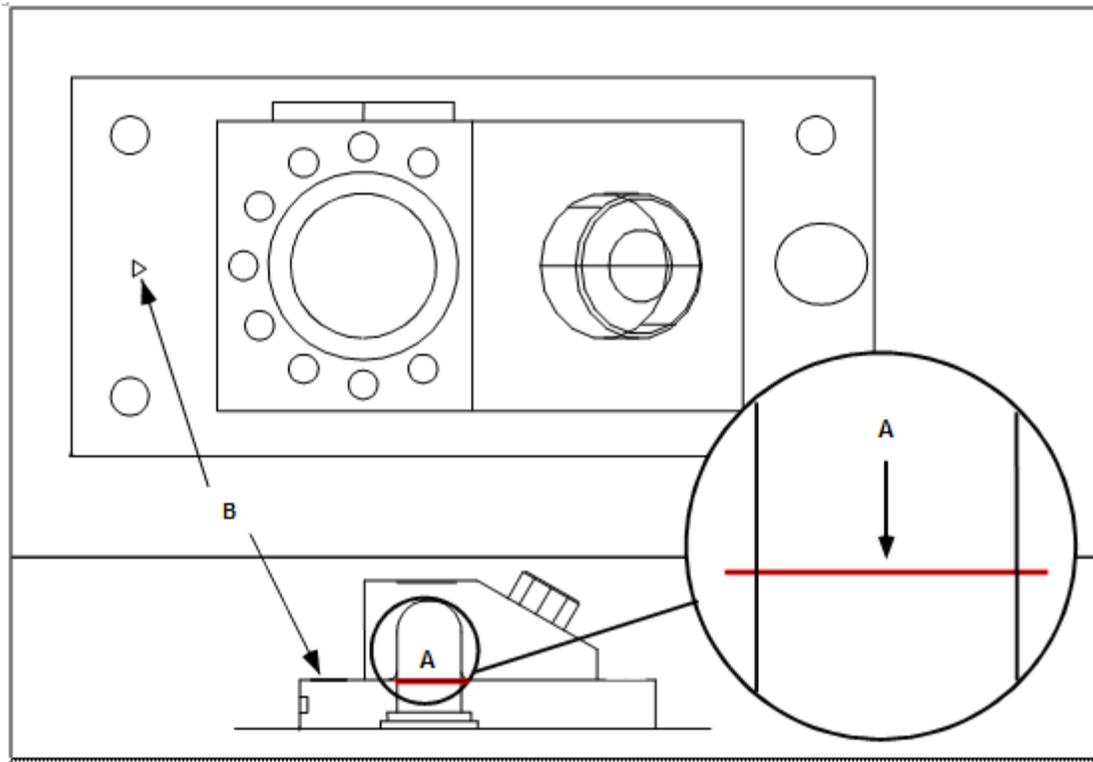
**Note:** If you only use point features as your input features in a best fit construction, the software compensates the constructed feature by the probe tip's radius. This means the construction behaves as if each point had been registered at the very center of the probe tip.

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

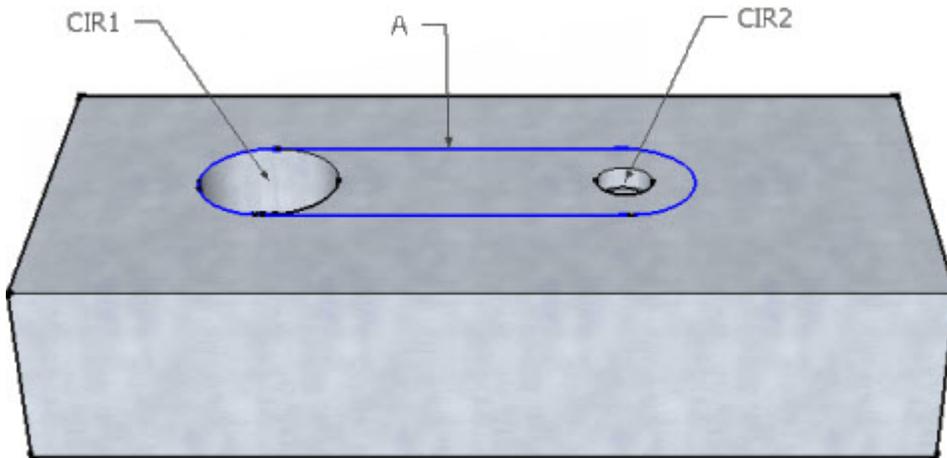
## Round Slot Construction Methods

### 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.

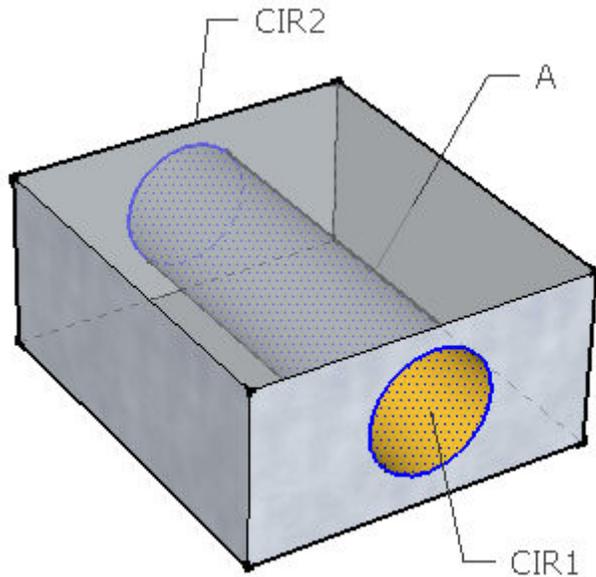
## Cylinder Construction Methods

### 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).

---

## 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:**

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 following occur:
  - The software enters [Run mode](#).
  - 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.

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- The **X**, **Y**, and **Z** values underneath the Graphics window change dynamically to show the probe's current position as you move the probe during measurement.
  - 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 flick 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 a report, see "[How to Generate, Print, Saving, or View a Report](#)".
- 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:**

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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 Generate, View, Print, or Save a Report

**Objective:** To preview the report and then send the output to your default printer.

**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 **Print Preview**  to open the **Print Preview** screen.

### To Print or Save 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 **XPS**  to save the report as an .xps file. PC-DMIS Touch saves the report to the My Documents folder using the same name as your part file.
- Tap **PDF**  to save the report as a .pdf file. PC-DMIS Touch saves the report to the My Documents folder using the same name as your part file.

### 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 **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.

**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:

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- First, a folder view. The **Reports** screen initially displays a list of all of the report folders. 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. As you type characters in the search box, the software automatically filters the folders to only show the ones 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.

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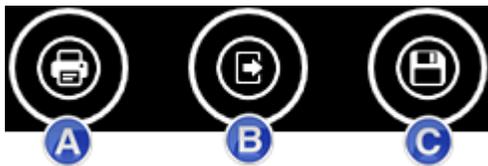
-  - Displays the Diameter value.
-  - Displays the Radius value.
-  - Displays the Roundness error.
-  - Displays the Angle value.
- **CY** - Displays the cylindricity error.
- **TP** - Position Tolerance. This is used in Position dimensions.
- **DF** - Diameter of the feature being dimensioned.
- **LF** - 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 three buttons.



- A. **Print Preview** - This shows the report in the [Print Preview screen](#).
- B. **Clear** - This clears the current report data from the screen.
- C. **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 Nominals, Tolerances, and Axes

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



The small dot to the left of a feature or dimension name, and the background color around the dot, act 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. Its color also acts as a status indicator.

- If you see a background color, at least some information from the feature or dimension is enabled to show in the report.
- If you don't see a background color, it is hidden from the report.

On the **New Measurement** and **New Construction** screens, this button does not show a background color; you must explicitly tap on it 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, this button will always show a background color, 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. If you set tolerances to zero for an axis row, the axis is automatically turned off. Similarly, if no tolerances exist, and you tap an axis, the software chooses default tolerances.
- 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:

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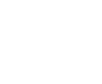
This example shows the different components making up an axis row:



- 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. **Lower Tolerance** - You can edit this value.
- D. **Upper Tolerance** - You can edit this value.
- 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.

**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 flicking 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

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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.

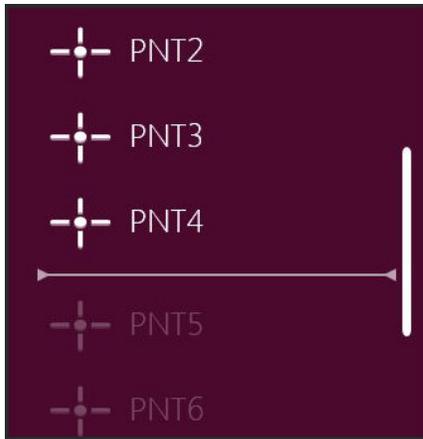
### Child Commands

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 flick 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 \(for Editing Commands\)\]](#)

## 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](#)

## 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:

## PC-DMIS TOUCH Portable

**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](#)".

**Flicking** - Dragging your finger across your touch screen performs a "flick". The user interface screens in this product typically have multiple options laid out horizontally on the screen. Flicking horizontally, left or right, brings new screens or functionality into view.

**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.

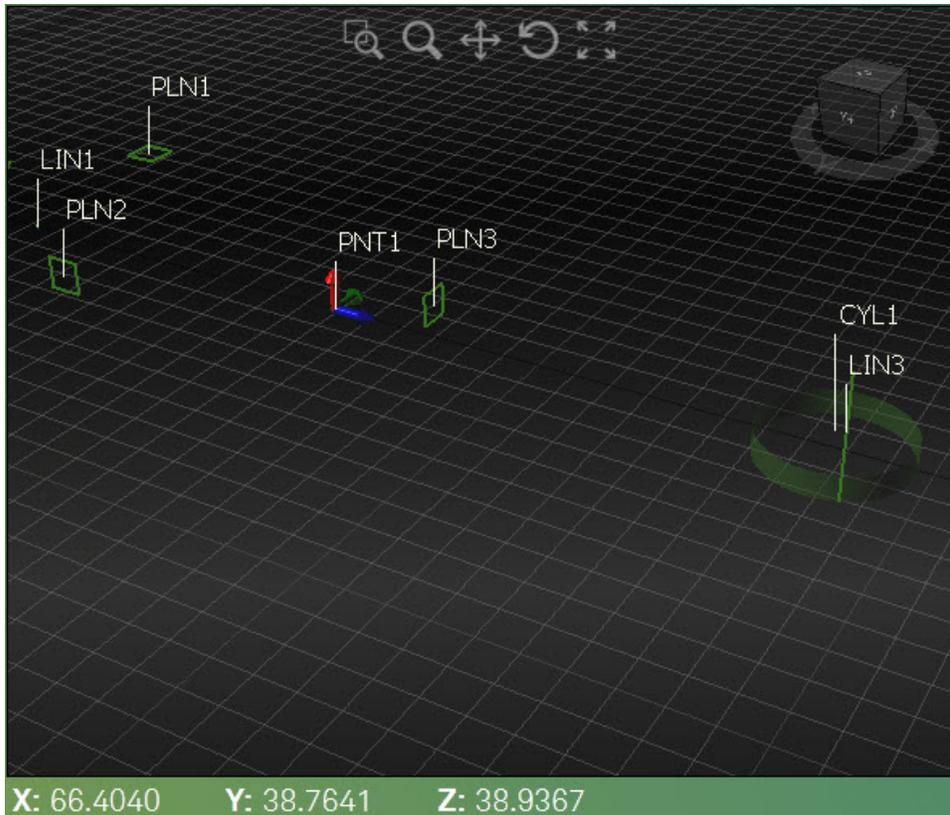
**Stretching** (Graphics window) - Stretching the your fingers apart zooms in on the model.

---

## 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.

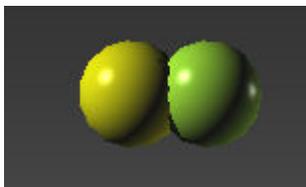
The Graphics window can draw these feature types: points, lines, circles, planes, cylinders, cones, round slots, square slots, and spheres.



### 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

## PC-DMIS TOUCH Portable

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 drawn fainter, with translucency, and without feature labels.

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

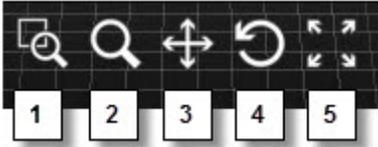
The cube in the upper-right of the window provides an alternate way of orienting the view. On the cube, or on the circular band surrounding it, tap on any face, edge, corner, or axis letter. It turns red and then rotates until that face, edge, corner, or axis is facing you. For example, tapping on the 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](#)".

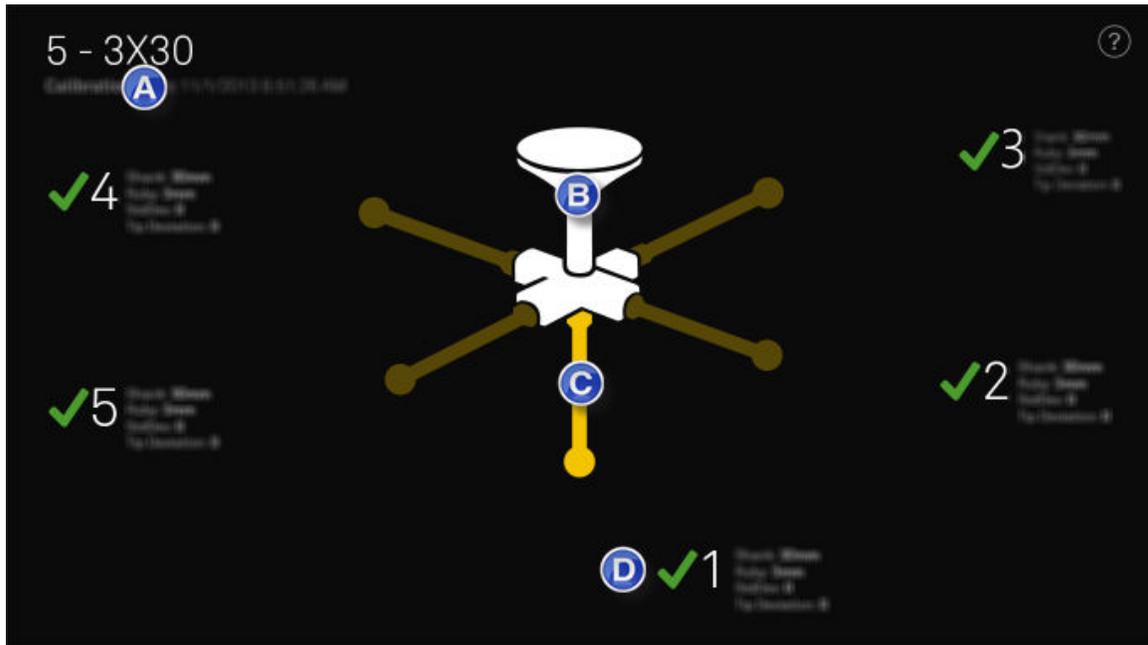
### Digital Readout

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.

The DRO helps you know the probe tip's current position. It also helps you know 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](#)

## 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 and calibration date.
- B. Diagram of the probe body.
- C. Diagram of the probe tip. The active probe tip shows in a brighter hue than other tips.
- D. A number for each tip used on the probe. Next to each tip number, the following information is displayed:
  - **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 indicates a good calibration that is not stale.
- A yellow tip, like that shown in the example image above, 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 Changer Configuration screen
- The Probe command in the [Commands list](#)
- The Probe calibration screen

---

## About Advanced Properties

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

### *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](#)

---

## The Tools Tile

The **Tools** tile shows the most recently accessed or commonly used tools. Tap on one of the buttons on this tile to open the selected tool or perform the action.

**Settings** - Accesses the Tools screen. For information, see "[The Tools Screen](#)".



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

Using this screen, you can then select the desired tool.

---

## The Tools Screen

The **Tools** screen contains a list of all available tools. You can choose to access any available tool from this screen.

To access this screen, from the [Home screen](#), on the **Tools** tile, tap **Settings**.

---

## About Keyboard and Keypad Input

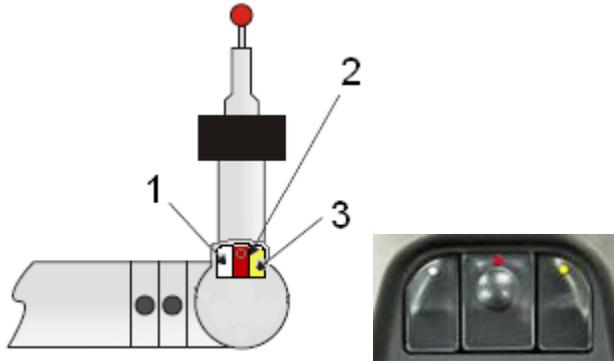
If you tap a box that accepts user input, depending on the type of field, either a complete virtual keyboard or a smaller number keypad appears as part of the user interface.

- For the keypad, after tapping the new value, to accept a new value and close the keypad, tap the green check mark on the keypad.
- For the full virtual keyboard, it closes once it loses focus or once you tap X on the keyboard's title bar.

You can also use PC-DMIS Touch with a standard desktop configuration with a keyboard and mouse.

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## About Portable Arm Buttons



The following Measure Mode functions are available for the buttons indicated above:

- **Measure one point** - Press and release the RED button (item 2 in graphic above).
- **Erase one point** - Press and release the WHITE button (item 1 in the graphic above).
- **Finish a measurement** - Press the YELLOW button to end a measurement (item 3 in the graphic above).

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## About Remote Mouse (Mouse Mode)

Your Absolute Arm supports a mode called Remote Mouse (also sometimes called Mouse mode). This means that when you place your arm hardware in this mode, it performs similarly to a standard computer mouse, allowing your arm's wrist motion and button presses to perform mouse clicks on the computer screen. This provides speed and efficiency when measuring because you do not have to be physically touching your screen to interact with the user interface.

For specific information on how to enable and use mouse mode, in your arm's user manual, consult the "How to use the Remote Mouse" topic.



# Glossary

## A

**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

**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.

## D

**Datum Feature:** A feature on the part identified by a datum reference symbol on the print.

**Digital Readout:** A portion of the user interface showing the current XYZ location of the probe tip.

**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.

**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.

## L

**Learn Mode:** A mode where the software can learn what features you want it to measure. You determine the features to learn by taking 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

**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.

**Mouse Mode:** A hardware mode that allows your measurement arm to act like a mouse and interact with your computer screen.

## P

**Part File:** A file containing the instructions needed to measure the learned features stored in the file.

**PC-DMIS Mode:** A software mode where the PC-DMIS TOUCH user interface is turned off and PC-DMIS runs in its native user interface. This mode is only available if you purchase a Touch with PC-DMIS license.

**Point:** A recorded XYZ location of the probe tip against some surface. Also known as a hit.

## R

**RFS:** Regardless of Feature Size.

**Run Mode:** In this mode, the software measures learned features.

## T

**Tile:** A grouping of the most recently selected or most popular items. They appear on the Home screen for quick access.

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