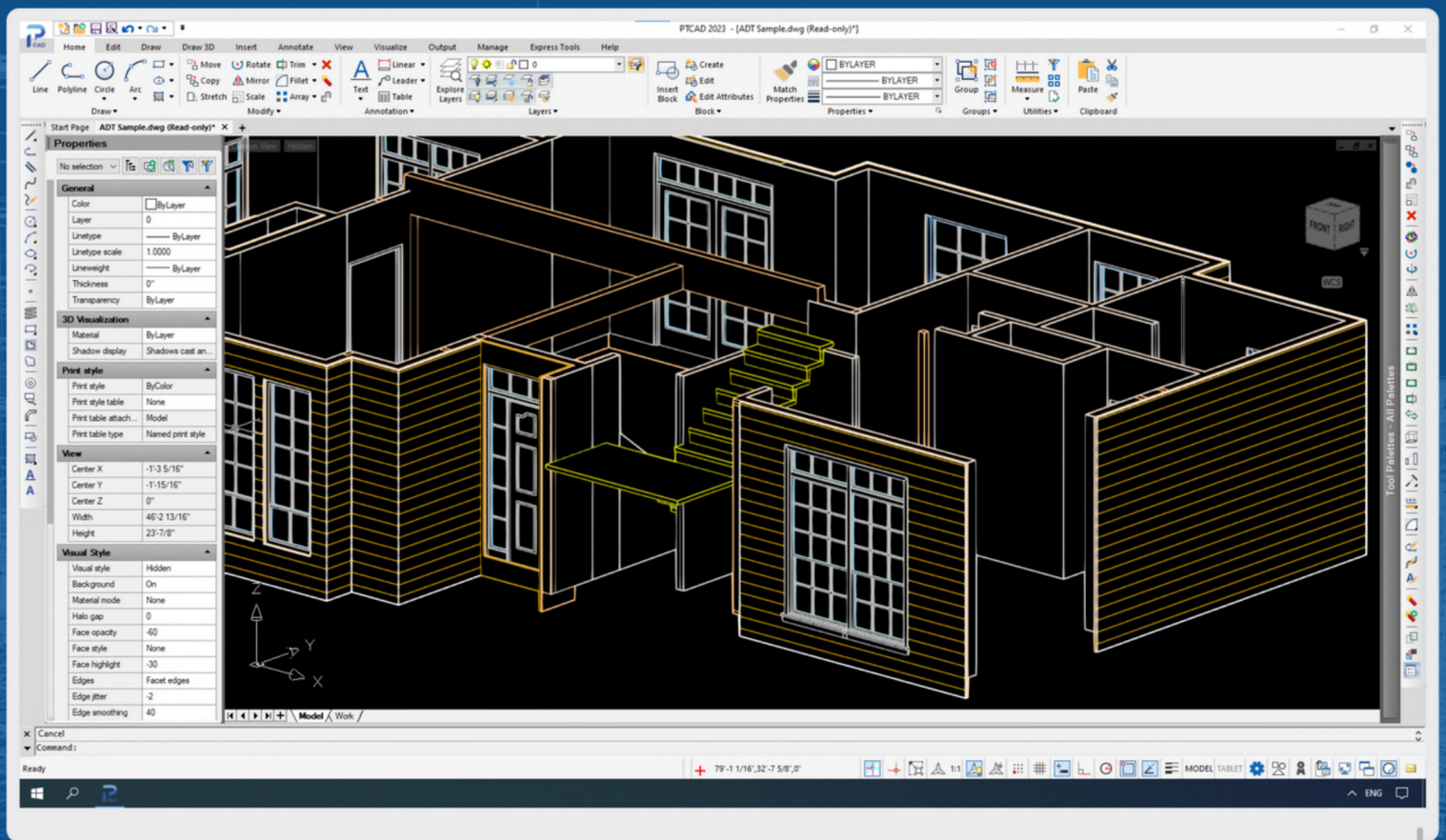


PTCAD

Unbeatable value for a complete CAD solution

User Guide



Natively DWG Compatible | Flexible Licensing | Lisp & Plugin Support



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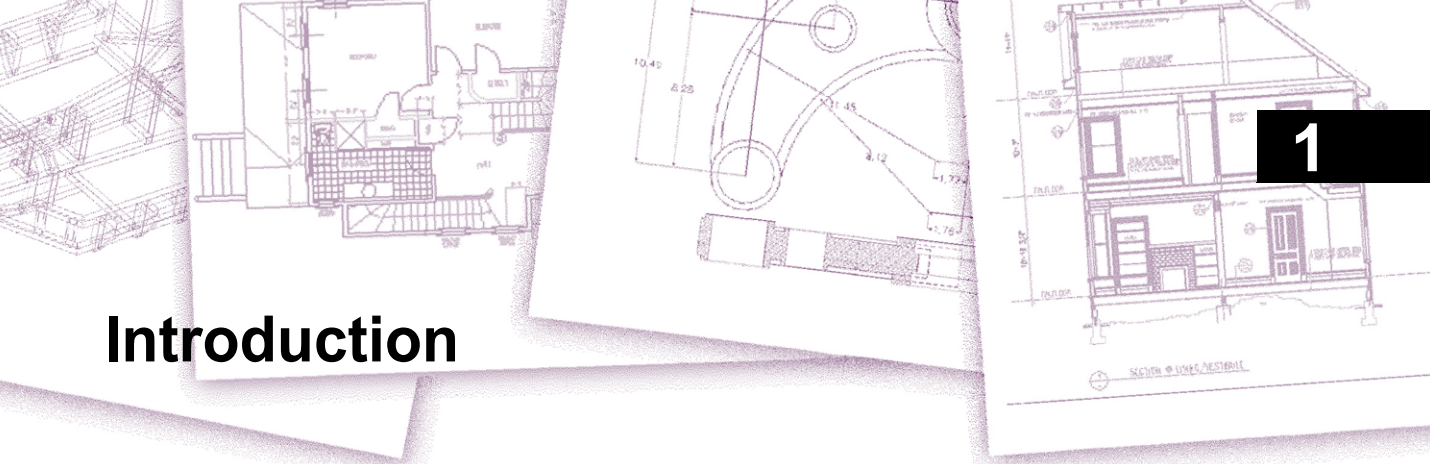
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Introduction

Using PTCAD is part of an integrated documentation set that includes this manual and a comprehensive collection of help resources to give you the information you need to create drawings in PTCAD®.

Topics in this chapter

- About PTCAD and other CAD software 3
- Comparing PTCAD and CAD to manual drafting 5
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This manual is organized into chapters that parallel how you work in PTCAD, according to the tasks you might perform. The tasks are divided and organized into the following work-focused chapters:

Introduction: Chapter 1 An overview of the key features of PTCAD plus basic concepts of computer-aided design (CAD) as they apply to PTCAD.

Getting started: Chapter 2 Installing PTCAD, starting and exiting PTCAD, working with toolbars, and selecting commands.

Working with drawings: Chapter 3 Opening and saving an existing drawing and starting a new drawing. Using drawing settings to establish paper size, scale factors, and text height. Working with colors in your drawings. Using drawing aids such as entity snaps and orthogonal mode to draw accurately.

Creating entities: Chapters 4–5 Working with simple entities such as lines, circles, and arcs and with complex entities such as polygons, spline curves, planes, wipeouts, boundary hatches, and more.

Viewing your drawing: Chapter 6 Moving around in the drawing, changing its magnification, creating multiple views, and saving arrangements of windows of the drawing.

Working with coordinates: Chapter 7 Working with Cartesian coordinate systems, specifying two-dimensional and three-dimensional coordinates, and defining your own user coordinate systems.

Working with the Explorer: Chapter 8 Using PTCAD's Explorer to manage your drawings: controlling layers, layer states, linetypes, multiline styles, text styles, table styles, dimension styles, multileader styles, coordinate systems, views, visual styles, layouts, materials, groups, external references, and blocks and copying information between drawings.

Getting drawing information: Chapter 9 Working with the additional information in PTCAD drawings, calculating areas and distances, and displaying other drawing information.

Modifying entities: Chapter 10 Selecting, copying, rearranging, resizing, and editing entities.

Annotating and dimensioning your drawing: Chapters 11–12 Using text to annotate drawings; using dimensions to annotate the measurement of entities.

Working with blocks, attributes, and external references: Chapter 13 Using blocks and external references to combine entities and data for reuse; creating attribute information to extract for use in other programs.

Formatting and printing drawings: Chapter 14 Combining drawings into finished layouts, customizing print options, printing copies, and also publishing multiple drawings at a single time using sheet sets and sheet lists.

Drawing in three dimensions: Chapter 15 Viewing, creating, and editing three-dimensional entities. And visualizing them using hidden line removal, shading, and photo-realistic rendering.

Working with other programs: Chapter 16 Sharing drawings and data with other users, documents, and programs.

Customizing PTCAD: Chapter 17 Customizing the appearance and operation of the program to suit your needs.

Understanding AutoCAD compatibility: Appendix Describes similarities and differences between PTCAD and AutoCAD.

This section introduces you to PTCAD®, its features, and its comprehensive capabilities for creating drawings of various types.

About PTCAD and other CAD software

PTCAD is designed for anyone who wants a fast and efficient CAD program with all the power and versatility of standard programs such as AutoCAD® by Autodesk®, Inc., or MicroStation® by Bentley Systems, Inc., at an affordable price. Using today's advanced technology, PTCAD integrates the Microsoft® Windows® interface with a powerful CAD engine.

PTCAD provides unparalleled compatibility with AutoCAD, using most of the same file formats including those for drawings (.dwg files), commands, linetypes, hatch patterns, and text styles. You can also use AutoCAD menu files and run Autodesk® AutoLISP® programs. If you have written your own ADS (Autodesk® AutoCAD Development System®) programs, simply recompile them to link with the PTCAD libraries. Many third-party ADS programs already support PTCAD. If you have a program that is not already supported, ask your software vendor to provide an PTCAD-compatible version of the program.

PTCAD is more compatible with the AutoCAD program than any other CAD product, delivers additional tools with advanced CAD features, and has a seamless Microsoft® Windows integration. This powerful program provides a superb combination of features for CAD users like architects, engineers, and designers.

PTCAD incorporates standard features found in other CAD programs, along with features and capabilities you won't find anywhere else. Its multiple-document interface (MDI) lets you open and work with several drawings at the same time. You can easily copy drawing entities between drawings. In addition, the powerful Explorer lets you manage information and settings and quickly copy layers, linetypes, and other information between drawings.

Using AutoCAD legacy drawings

PTCAD fully supports AutoCAD legacy drawings. PTCAD reads and writes .dwg files in their native format without any loss of data, from AutoCAD 2007 back to Version 11, including AutoCAD LT®. Because PTCAD uses Autodesk DWG format as its native file format, no translation is required.

PTCAD provides you with the appropriate tools for your experience level—whether you are a beginner, intermediate, or advanced CAD user. If you are just starting out with CAD, you may want to use the beginner level, which provides toolbars containing the basic tools such as lines, arcs, and circles. As you become more experienced, you can move to the intermediate level and then to the advanced level, which gives you access to more than 300 commands through toolbars, menus, and keyboard entries. You can change the experience level in the Options dialog box on the Tools menu.

PTCAD supports three-dimensional wireframes and surfaces. Three-dimensional drawings can be displayed in wireframe, hidden-line view, and surface shading. Some versions of PTCAD also support creating and editing 3D solids; all versions display 3D solids, along with limited editing features.

Some versions of PTCAD support displaying and working with raster images in your drawings. However, PTCAD does not display images located inside of blocks and externally referenced drawings (xrefs). When a drawing containing proxy entities is loaded into PTCAD, a message displays indicating that some entities will not display, however, the entities reappear when you open the drawing later in AutoCAD.

It's easy to customize PTCAD. You can modify menus and toolbars, create custom menus, dialog boxes, command aliases, and scripts, and add custom programs written in any of several programming languages, including DRX (the program's Autodesk® ARX-compatible language), LISP (the program's Autodesk AutoLISP-compatible language), and SDS™ (Solutions Development System™, the program's Autodesk ADS-compatible language). There is also Microsoft® Visual Basic for Applications (VBA).

You can run existing Autodesk® AutoLISP applications in PTCAD with little or no modification. PTCAD uses the `Apload` command so you can easily load LISP programs. PTCAD reads files that contain dialog control language (DCL) statements as well, which makes PTCAD compatible with dialog boxes created for AutoCAD.

Using AutoCAD commands with PTCAD

Because PTCAD supports hundreds of AutoCAD commands, you use the commands you already know. For example, to draw a circle, use the `Circle` command. To copy a circle, use the `Copy` command.

When you press `Enter` or the spacebar, you activate the command—just like in AutoCAD. PTCAD accepts the special characters used by AutoCAD, such as point filters (for example, `.xy`), relative coordinates (the `@` symbol), and the apostrophe (the `'` prefix) for transparent commands. PTCAD function keys are also similar to those used in AutoCAD.

Because you do not need to learn a new set of commands, you are immediately productive with PTCAD.

Comparing PTCAD and CAD to manual drafting

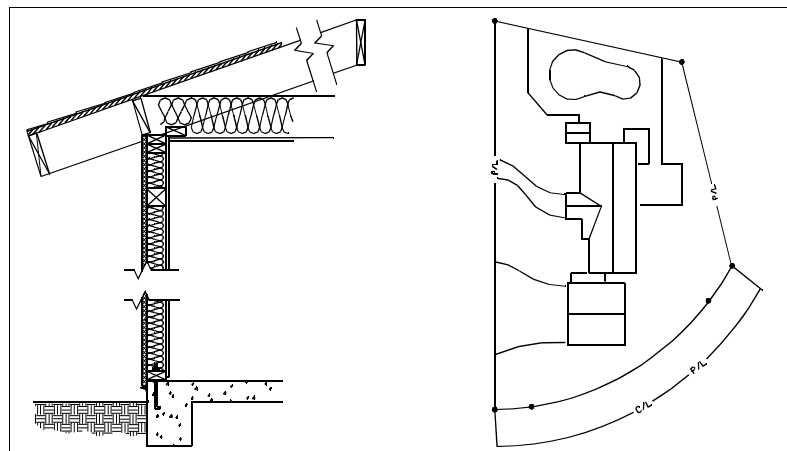
PTCAD greatly reduces the time and effort it takes to create and revise drawings. Not only can you produce accurate drawings faster, you can also reuse the information in your drawings. These are the primary reasons for making the transition to CAD from traditional, manual drafting on paper.

As with any tool, however, to use it effectively, you need to be familiar with some of the special features, functions, and concepts of CAD. If you are familiar with manual drafting, you'll find some conceptual similarities in CAD as well as some differences.

Drawing to scale

In traditional, manual drafting, you usually determine the scale of the drawing before you even start to draw, because you are working with a sheet of paper of a fixed size. You may have to reduce or enlarge the entity you are drawing to fit within the confines of the paper.

When you create a drawing in PTCAD, you draw everything full-size. You determine the type of units in which your drawing is measured. If you are drawing a building, 1 drawing unit might equal 1 inch. If you are drawing a map, 1 drawing unit might equal 1 mile. Your drawing environment and the CAD drawing file itself are not limited to the size of a particular sheet of paper.



As you draw, you can use commands such as Pan and Zoom to work on different portions of the drawing and to magnify the display of the drawing to view small details. These commands have no effect on the actual size of the entities in your drawing; they affect only the way the drawing is displayed on your screen. Only when you print or plot your drawing do you need to set the scale so that the printed drawing fits within a specific paper size.

Using tools

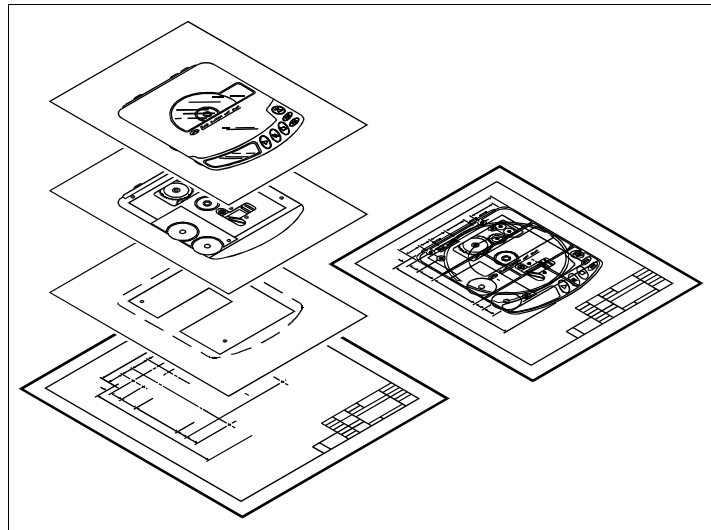
In manual drafting, you use tools such as pencils, rulers, T-squares, templates, erasers, and so on. When you create a drawing in PTCAD, you use a mouse instead of a pencil, and you use the mouse to select other tools—commands you select from a menu or a toolbar.

You use some tools to create basic entities, such as lines, circles, and arcs, and other tools to modify existing entities (for example, to copy or move them or to change properties such as color and linetype).

Organizing information

In traditional drafting, you often separate elements such as walls, dimensions, structural steel members, and electrical plans onto separate, translucent overlays. When you want to print the working drawings, you can create several different drawings by combining different overlays.

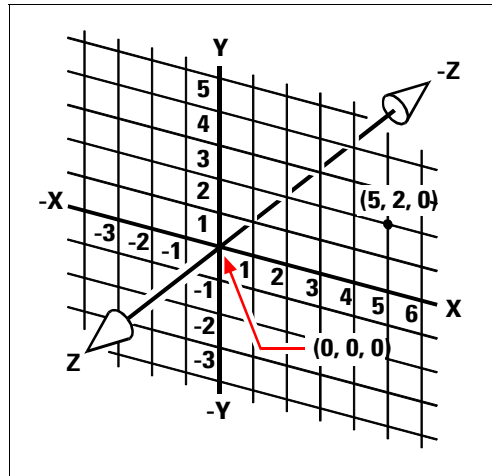
When you create a drawing in PTCAD, you use layers to organize elements in a similar manner. However, the layers feature in CAD offers numerous advantages over physical transparencies. The number of overlays you can combine to print a manually drafted drawing is limited by the printing process. There is no such limitation in CAD. With PTCAD, you can define an unlimited number of layers, any of which can be visible or invisible at any time. You can name each layer and assign each its own color, linetype, lineweight, and print style. You can also lock individual layers to ensure that information on those layers isn't altered accidentally.



Use layers to organize drawing information.

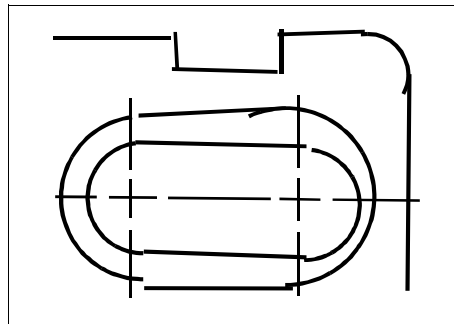
Drawing accurately

When you create a manual drawing, ensuring accuracy typically requires a lot of manual calculations and rechecking. By contrast, PTCAD offers a number of drawing aids that ensure accuracy from the start. For example, you create and modify entities based on an underlying Cartesian coordinate system. Every location in the drawing has its own x,y,z -coordinates. You can also display a grid as a visual reference to your coordinate system.

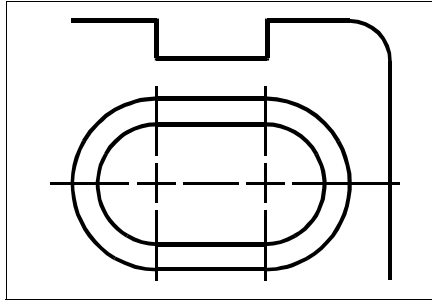


Every location in the drawing has its own x,y,z -coordinates within the underlying Cartesian coordinate system. Point $5,2,0$ is 5 units to the right (along the x -axis), 2 units up (along the y -axis), and 0 units (along the z -axis) from the origin (the $0,0,0$ point).

Settings such as snap and entity snap allow you to draw accurately without specifying coordinates. The snap setting forces the selected points to adhere to the grid increment or to any other increment you set. Entity snaps let you snap to precise geometric points on existing entities—for example, the endpoint of a line or the center of a circle. Another setting, orthogonal, constrains lines so that they are drawn parallel to the vertical and horizontal axes.



Paper-based drawings lack the high degree of accuracy possible when using CAD. Lines often overlap or fail to meet adjacent lines.



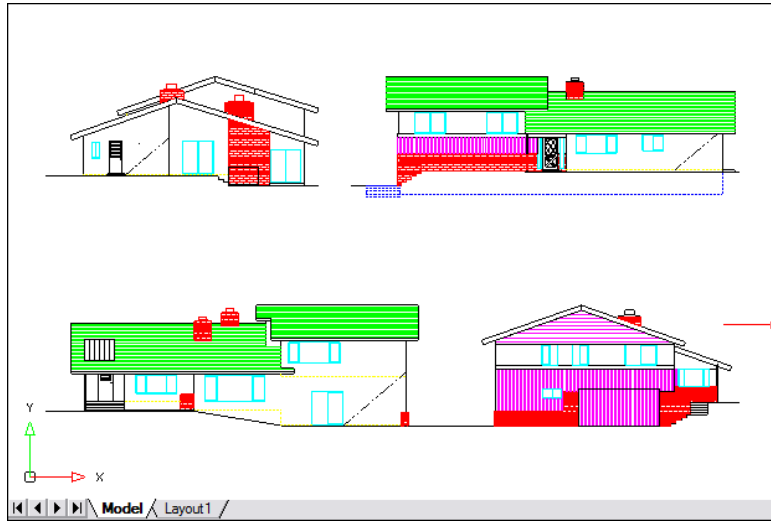
Snap and entity snap force the cursor to adhere to a specified increment or attach to key geometric points on existing entities. You can also constrain lines to vertical and horizontal axes.

Drawing efficiently

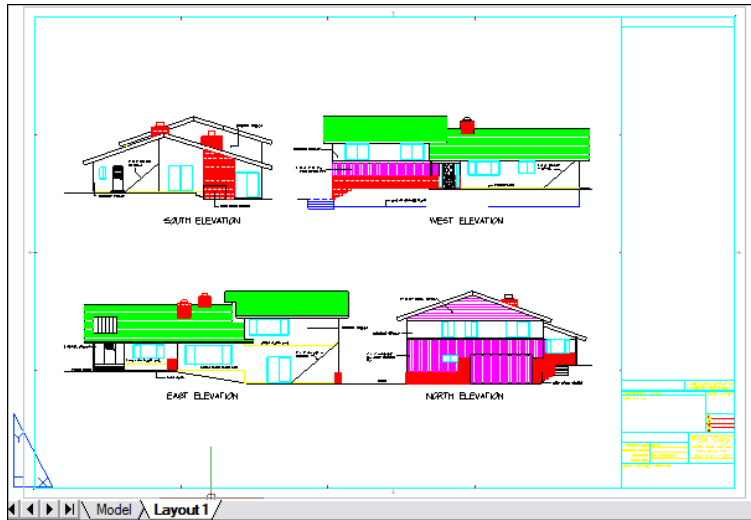
In paper-based, manual drafting, you often have to redraw the same entity several times at different scales or from different vantage points. You may also need to redraw the border and title block on each new sheet.

One of the most powerful features of PTCAD is that when you create a drawing, you can reuse individual entities, borders, and title blocks as often as you want. You need draw an entity only once; the final printed drawing can show the entity at several different scales and viewpoints.

You usually begin drawing in model space on the Model tab, creating the drawing (a floor plan, a map, or a three-dimensional part) without regard to the final layout on paper. When you are ready to print your drawing, you have the option to switch to paper space on a Layout tab, where you lay out the drawing as you want it to appear on a sheet of paper. For example, you can insert a drawing file that contains the standard border and title block that you created. You can define and arrange multiple views of the drawing at appropriate scales and with specific portions visible or invisible—again, without having to redraw the border and title block for each view.



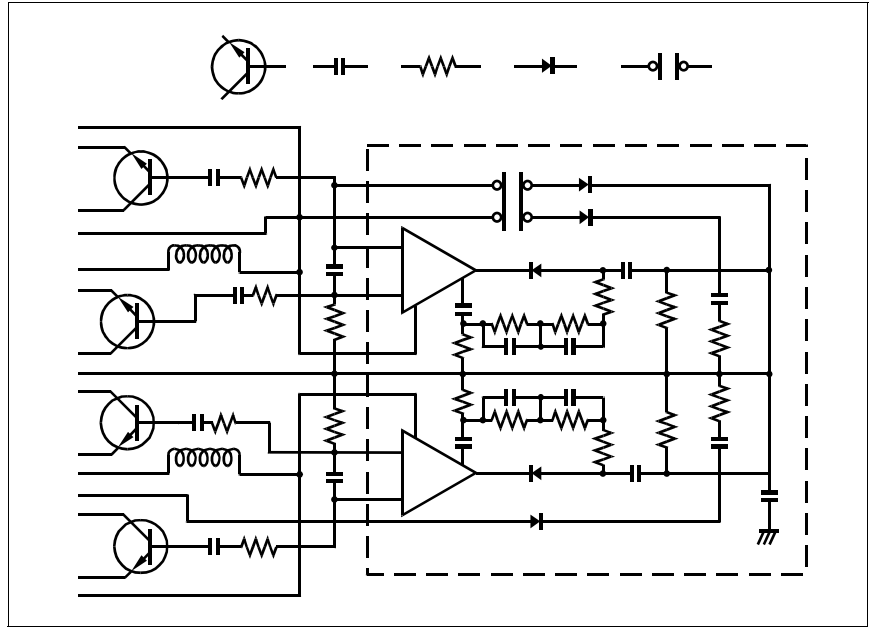
You create the basic drawing in model space on the Model tab.



When you're ready to print or plot your drawing, you can switch to paper space on a Layout tab, where you provide a layout of the drawing as you want it to appear on a sheet of paper.

Reusing CAD drawings and entities

When you create a paper drawing manually, you can draw repetitive symbols by tracing a plastic template. After you draw a symbol in PTCAD, you can reuse that symbol without having to redraw it. You simply save the symbol as a block. You can then insert copies of that block anywhere in your drawing. You can also save the symbol as a separate drawing for use in other drawings.

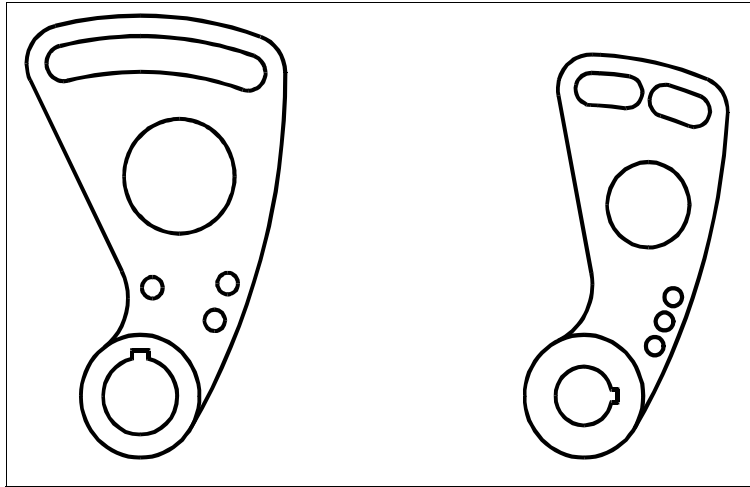


You can draw a symbol one time, save it as a block, and then insert multiple copies of that symbol anywhere in your drawing.

In addition, you can reuse entire drawings and insert individual drawings into other drawings. You can also use an external reference, which acts as a pointer to another drawing rather than a copy of the entire drawing. Using an external reference has an added advantage: when you update the externally referenced drawing, each drawing that references it can be automatically updated.

Making changes

To make changes to a paper drawing, you erase and then redraw. With PTCAD, you use commands to modify entities in the drawing. You can move, rotate, stretch, or change the scale of entities. When you want to remove an entity, you can delete it with a single click of the mouse. If you make a mistake while creating or modifying your drawing, you can easily reverse your actions.



You can easily change an entity using commands such as move, rotate, stretch, and scale instead of redrawing the entity.

Working with other data and programs

Traditional paper drawings serve only as a means of communicating information between the person who created the drawing and the person viewing the drawing. The drawings contain no more information than what is visually imparted by the creator and seen by the viewer.

PTCAD offers rich possibilities for analyzing drawings and attaching additional data to them. For instance, although it may be impractical to count entities in a complex paper drawing, this task is simple in CAD. PTCAD can calculate the number of entities in a drawing and compute area and distance.

CAD drawings can also contain information in addition to visible entities. You can attach invisible database information to visible drawing entities and extract the information for analysis in a database or spreadsheet. (Working with information in a database requires a program from a third-party vendor, or you can create your own means of exporting the data in LISP or SDS. Or, you can also use VBA.)

PTCAD provides in-place editing of Microsoft® objects, such as those created in Microsoft® Word and Microsoft® Excel software programs. In-place editing makes it easy to share data with other users and programs. For example, you can include PTCAD drawings in files created using Microsoft® Word, and you can insert files created using Microsoft® Word into your PTCAD drawings.

Using advanced CAD features

In addition to being compatible with AutoCAD, PTCAD goes several steps further by providing you with innovative features to increase your productivity.

Using PTCAD's Explorer

The PTCAD's Explorer has an interface similar to the Windows Explorer, allowing you to view and manage the elements of multiple, open drawings, such as layers, blocks, linetypes, views, user coordinate systems, materials, text styles, and more.

Editing multiple documents simultaneously

With PTCAD, you can open and edit multiple drawings simultaneously. You can also copy and paste elements between open drawings.

Editing multiple entities

PTCAD allows you to change most of the properties of all selected entities using a single, tabbed dialog box.

Using the CUI dialog box

PTCAD has a single dialog box for changing ribbon tabs (if your PTCAD version supports them), toolbars, menus, keyboard shortcuts, and command aliases. You can also use the drag-and-drop method to customize these user interface items. The simple, point-and-click action lets you easily create new items and keyboard shortcuts—no programming or manual text editing required.

Performing unlimited undo and redo

PTCAD increases your power with unlimited undo and redo of editing actions.

Getting more information

In addition to the PTCAD documentation, much of the assistance you need as you use PTCAD is specific to the commands you work with on the screen. To obtain immediate information as you work, use these additional sources of information:

- **Tooltips** — To find out what a specific tool on a toolbar does, pause the cursor over it for a moment. A ToolTip appears on the screen.
- **Status bar** — To find out more detailed information about a tool when you pause the cursor over it, look on the status bar at the bottom of the screen.
- **Online help** — PTCAD online help is available on the screen when you press F1, choose a command from the Help menu, or click the question mark in a dialog box. The online help also presents information that does not appear in this manual, including a programming reference that describes how to program in TX/IRX, LISP, DCL, SDS, and DIESEL. The programming reference also describes programming in VSTA and VBA.

Working with sample drawings

With the PTCAD program, you can create a variety of drawings, including two-dimensional architectural drawings, electrical schematics, and mechanical drawings. Viewing and working with sample files can be an easy way to quickly learn how to use various PTCAD features.

To access the sample files

- Choose File > Open, and then open the Samples folder.

New in PTCAD 2024

Performance

- Increased speed of opening drawings that contain many blocks or external references.
- Improved zoom and pan performance for drawings with a large number of entities and for drawings with geometry far from the origin.
- Faster zooming to extents and copying and pasting entities.
- Faster drawing of lines, panning, copying entities, and switching layouts when using the OpenGL ES graphics device.
- Improved scaling performance with blocks.
- Decreased memory usage when opening and closing drawings while using the OpenGL graphics device.

Working with Files

- Specify a coordinate system when importing map data from SDF, SQLITE, or SHP files.
- Export tables to .csv files.
- Import and export Facet Modeler Binary (.fmb) files.
- For versions that don't include ACIS, import entities as meshes from STEP and IGES data files.
- For PTCAD versions that support working with BIM files:
 - Select which entities to import from .ifc files and specify import options.
 - Include .rvt and .ifc underlays when exporting to .ifc files.
 - Attach roof and ceiling plane symbols to walls and reference them while interactively creating roof and ceiling planes using the Plane Solve command.
 - Assign levels to AEC entities. Levels contain default settings for bottom and top elevations, wall height, on/off status, lock/unlock status, freeze/thaw status, offset, and other settings for walls and plane symbols.
 - Customize AEC styles for walls, doors, windows, curtain walls and curtain wall units, stairs and railings, and slabs.

User interface

- Double-click an entity to display the Quick Properties pane for modifying properties that are used most often for the entity type. Use the Drawing Settings > Quick Properties tab to change the behavior of the Quick Properties pane. Use the Customize User Interface command to determine which entity types and properties are enabled to use with the Quick Properties pane.
- Use the Customize User Interface command to create and edit legacy image tile menus.

- Click the down arrow at the right end of the Layout tab row in the drawing window to display and choose from a list of Layout tabs.
- Use the new Startup dialog when the STARTUP system variable is set to 1.
- Use Diesel expressions when customizing the ribbon.

Selecting and modifying entities

- Use the Geometric Center snap to find the centroid of any closed polyline or spline.
- Trim all entities to the edge of a cutting entity in a single operation using the Quick Trim command.
- Use the Copy Multiple command to copy entities using multiple options.
- Preview modifications when using the Offset, Trim, and Extend commands.

Splines

- Draw splines using control vertices and knot parameterization. Choose Undo to erase the previously specified point.
- Choose Join when using the Edit Spline command to combine a selected spline with other splines, lines, polylines, or arcs.

Layers

- Use the Layer Isolate VP Freeze command to freeze the layers of selected entities in all layout viewports except the current viewport.
- Use the Viewport Layer command to freeze and thaw layers, and also use it from the Model tab.
- Control whether entities on layers can be cut by section planes

Tables

- Create and update tables from extracted drawing data.
- Remove formatting overrides from table cells.
- Specify border properties and apply borders to table cells.

Blocks

- Globally edit attribute text associated with blocks using the Global Edit Attributes command.
- Use the Rename Block command to create a new block by renaming a single block instance.
- Use the Change Block command to change the name of a single block instance and then modify it in the Block Editor.

Explorer

- Manage more visual style properties using the Explore Visual Styles command.
- Manage more layout settings using the Explore Layouts command.

- Specify whether to freeze layers automatically in layouts that you create using the Explore Layers command.
- Manage data links using the Explore External References command.
- Change the column order, visibility of columns, and restore the default display of columns in all Explorer panes.

3D modeling

- Use the Project Geometry command to create a projection of lines, curves, or points on a three-dimensional solid, surface, or region.
- Use the Show CV and Hide CV commands to display or hide the control vertices of selected NURBS surfaces and curves.
- Specify display settings for section planes.

Printing and publishing

- Import page setups from other drawings.
- Use presets when publishing or exporting to .pdf files.
- When publishing or exporting solid fill hatches to .pdf files, choose the Polygon option to generate an outer loop by the vectorizer using polygons and then fill it with the appropriate color.
- When using the Publish command, manage layer states by choosing Layer States Manager in the Layer States column.

More features

- Create reports and report templates using the Extract Data command.
- Use the Super Hatch command to hatch an area using a selected image, texture, block, external reference, or wipeout.
- Use the Frames command to show or hide the frames of all images and wipeouts.
- Preview modifications when editing hatches.
- View misspelled words and choose from suggested spellings when using the Edit Text command.
- Use the Set Z All command to change the z-coordinates of all entities contained in a drawing.
- Zoom using a mouse wheel while simultaneously panning with a mouse wheel.
- Access many new .NET classes, methods, and properties.

System variables

- COMMANDPREVIEW, FMDEVIATION, MILLISECS, PRINTERCONFIGDIR, PRINTMINPAGEMARGIN, PRINTPDFUNDERLAYTRANSPARENCY, QPLOCATION, QPMODE, SELECTLOCKEDLAYER, SPLDEGREE, SPLKNOTS, SPLMETHOD, SPLPERIODIC, SURFACEAUTOTRIM, VPSCENEAMBIENTCOLOR, and VPUSESCENEAMBIENTCOLOR.

- GSDEVICEMODE default is now 3.
- Updated: MTEXTFIXED.
- Environment system variables: PrinterConfigDir and PrinterStyleSheetDir.



Getting started

This section helps you get started using PTCAD software by explaining how to install it and providing basic information about how to use it.

This guide assumes that you have working knowledge of Windows-based programs.

If necessary, see the documentation that came with your operating system for information about Windows terminology and techniques.

Topics in this chapter

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<i>Installing PTCAD</i>	21
<i>Starting PTCAD</i>	22
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System requirements

You need the following software and hardware to install and run PTCAD:

- Microsoft® Windows® 11, Windows 10, and Windows 8 on a 64-bit computer
- 1 GHz or faster processor
- 4 gigabytes (GB) of RAM minimum
- 2.5 gigabytes (GB) of free hard disk space recommended for typical installation
- 1024 x 768 VGA or higher resolution, video adapter, and monitor
- Graphics card compatible with OpenGL Version 1.4 or higher
- Keyboard and mouse, or other pointing device
- CD-ROM or DVD drive for installation, if installing from a CD or DVD
- Internet access, if downloading and installing from the Internet

For digitizer tablets, PTCAD requires a 64-bit wintab-compatible driver installed.

Installing PTCAD

A setup program guides you through the PTCAD installation process. The program transfers files to a folder that it creates on your hard disk. The program also creates a menu item on the Start menu.

Installation starts automatically after you insert the PTCAD compact disc into your CD-ROM drive. If installation does not start, you can install PTCAD by using the following procedure.

To install PTCAD from a compact disc

- 1 Insert the PTCAD compact disc into your CD-ROM drive.
- 2 Do one of the following:
 - Wait for the autorun feature to start.
 - Choose Start > Run and in the Open field, type *d:\setup*, where *d* is the letter assigned to your CD-ROM drive. Click OK.
- 3 Follow the instructions on your screen.

Some PTCAD versions may not come with a compact disc.

For example, if you downloaded the program from the Internet, follow the instructions that came with the program.

Starting PTCAD

To start PTCAD, choose Start > All Programs > ITC > PTCAD (may vary, depending on your operating system).

Each time you start PTCAD, the Start Page displays on your screen (unless on the Start Page you have marked Skip Start Page Next Time). Use the Start Page to quickly open drawings, create new drawings, and send feedback about the program.

The Recent Document list on the Start Page is customizable.

Right-click an item in the Recent Document List and choose whether to move the item to the top of the list, delete it from the list (the document itself is not deleted), or copy its path. You can also choose whether single- or double-clicking a filename or thumbnail image opens the corresponding drawing.

For versions of PTCAD that don't have a Start Page, the program opens a new, blank drawing based on a default template, `icad.dwt`. Using a template as the basis for a new drawing has several advantages:

- You can use predetermined units of measure, grid settings, text heights, and other settings appropriate for the type of drawing you're creating.
- You can predefine special layers.
- You can predefine the type of print style table.
- You can include predefined title blocks and borders.

Working in PTCAD

You can work with the PTCAD window and its elements in a variety of ways. For example, you can display and rearrange tool palettes, the toolbars, display the command bar, and enable the status bar. The toolbars and command bar can also be floated anywhere on the screen or docked to the edges of the main PTCAD window.

Use a shortcut to turn window elements on or off all at once.

When multiple window elements are enabled, the size of the drawing window is reduced, which can limit how much drawing data can be displayed. If you want more screen space dedicated to drawing, simply choose Clean Screen On/Off in the status bar. The window elements will hide until you choose the toggle again.

Displaying file tabs for drawings

Multiple drawings can be open at the same time. By default, each drawing displays its own tab. Select a tab to switch the active drawing.

To toggle the display of file tabs for drawings

- 1 Do one of the following:
 - On the ribbon, choose View > File Tabs (in Display)
 - On the menu, choose View > File Tabs.
 - Type *filetab* or *filetabclose* and then press Enter.

Open drawings can also be arranged side by side.

For more details about viewing drawings, see “Working with multiple drawings” on page 246.

Displaying commands on a shortcut menu

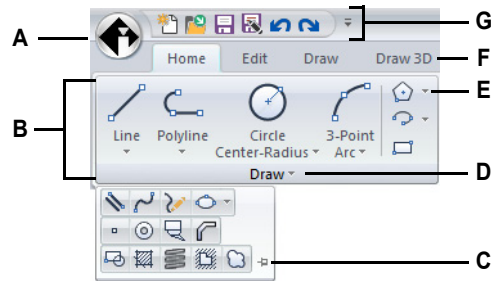
Shortcut menus provide quick access to specific commands. A shortcut menu displays when you right-click an entity, toolbar, status bar, the Model tab name, or a Layout tab name. The selections presented in the shortcut menu depend on what you right-clicked.

When you right-click the drawing, you can choose from a wide range of commands, including recently used commands. When you right-click a toolbar, the program displays a shortcut menu that lets you toggle the command bar, status bar, and various toolbars on and off. If you select one or more entities and then right-click, the program displays a shortcut menu from which you can choose a command to modify the selected entities. To display a shortcut menu from which you can choose an entity snap, press and hold down the Shift key, and then right-click anywhere within the drawing window.

Displaying and hiding the ribbon

The ribbon contains several areas from which to choose commands:

- **Application button** — The Application button in the upper left corner contains file-related commands, such as New, Open, Import, Export, and more.
- **Quick Access toolbar** — Contains commonly used commands. Click the Quick Access toolbar down arrow to choose which commands display and to customize various visual elements of the drawing window.
- **Tabs** — Contains related commands grouped together, for example, on the tabs named Home, Edit, Draw, etc.
- **Panels** — Contains sub-categories of commands within a tab, for example Draw, Modify, and Layers on the Home tab.



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| <p>A Click the Application button to view and select file-related commands.</p> <p>B Displays groups of commands in panels.</p> <p>C Click to lock the expanded panel open.</p> <p>D Click to expand the panel to show additional commands below it.</p> | <p>E Click to expand the panel and view related commands below.</p> <p>F Click a tab to view different commands.</p> <p>G Click a tool on the Quick Access Toolbar. You can also click the arrow on the right to choose additional options.</p> |
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To customize the Quick Access toolbar

- 1 To add a command to the Quick Access toolbar, right-click the command in the ribbon, then choose Add to Quick Access Toolbar.
- 2 To remove a command from the Quick Access toolbar, right-click the command to delete, then choose Remove from Quick Access Toolbar.

To customize a tab in the ribbon

- 1 Right-click anywhere in the tab of the ribbon you want to customize.
- 2 Do one of the following:
 - Choose Show Tab, then choose the tabs you want to display or hide.
 - Choose Show Panel, then choose the areas that you want to display or hide for that tab.

The ribbon can be customized in other ways.

You can use the Customize UI command or manually edit the .cui file for the ribbon. For more details, see “Customizing the ribbon” on page 879. Also see “Working with user interface customization files” on page 916.

To minimize the ribbon

- 1 Do one of the following:
 - Double-click the name of a ribbon tab.
 - Click the down arrow on the Quick Access toolbar, then choose Minimize the Ribbon.

To show both the ribbon and menus at the same time

- 1 Right-click anywhere in the ribbon.
- 2 Choose Menu Bar.

To hide the ribbon

- 1 Do one of the following:
 - Right-click anywhere in the menus or ribbon, then choose Switch to Menu Bar. This hides the ribbon and displays the menus instead.
 - Type *ribbonclose* and press Enter to hide the ribbon only.

To display the ribbon

- 1 Do one of the following:
 - Right-click a menu or any toolbar, then choose Switch to Ribbon.
 - Type *ribbon* and press Enter.

When using the ribbon, switch between drawings using file tabs or using the status bar.

Click a drawing’s file tab to display the drawing. If file tabs are not visible, type filetab and press Enter. You can also switch between drawings using Show Window Menu on the status bar.

Workspaces also can control the ribbon.

The PTCAD Classic workspace shows toolbars and the Drafting and Annotation workspace shows the ribbon. For more details about workspaces, see “Using workspaces” on page 33 in this chapter.

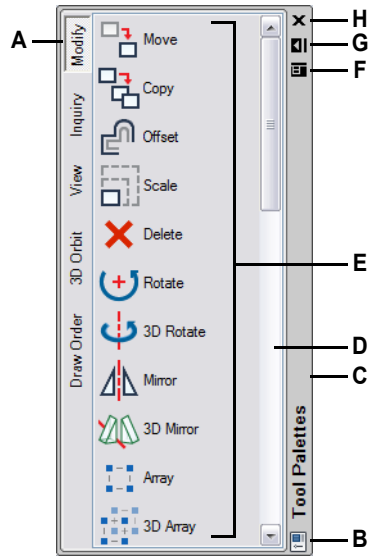
Ribbon display can also be affected by Clean Screen On/Off on the status bar.

To control whether the ribbon is included when you click Clean Screen On/Off on the status bar, right-click the icon and mark or unmark Ribbon.

Displaying and hiding the Tool Palettes pane

The Tool Palettes pane displays groups of organized commands and blocks for easy selection.

The Tool Palettes pane can be either docked or floating. A floating pane has a Close box and options along the right, can be located anywhere on the screen, and can be resized. A docked pane has a title bar, a Close button in the upper right only when the cursor moves over it, cannot be resized, and is attached along one of the edges of the drawing window.




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| <p>A Click a palette to show its commands. Right-click for more options.</p> <p>B Double-click to dock the pane. Right-click for more options.</p> <p>C Click and drag to move the pane to a new location. Right-click for more options.</p> <p>D Click to scroll the list of commands. Right-click for more options.</p> | <p>E Click a tool to start its command. Right-click for more options.</p> <p>F Click to display a menu of options: close, dockable, fix to left, fix to right, autohide, and transparency.</p> <p>G Click to auto-hide the pane.</p> <p>H Click to close the pane.</p> |
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To display the Tool Palettes pane

- 1 Do one of the following:
 - On the ribbon, choose View > Tool Palettes (in Display).
 - On the menu, choose View > Display > Tool Palettes.
 - Right-click a ribbon tab (if your version has a ribbon), toolbar, or menu and choose Tool Palettes.
 - Type *toolpalettes* and press Enter.

To close the Tool Palettes pane



- 1 Do one of the following:
 - On the ribbon, choose View > Tool Palettes (in Display).
 - On the menu, choose View > Display > Tool Palettes.
 - On the Tool Palettes pane when it is undocked, click () in the upper right corner.
 - Right-click a ribbon tab (if your version has a ribbon), toolbar, or menu and choose Tool Palettes.
 - Type `toolpalettesclose` and press Enter.

The display of tool palettes can be affected by other commands too.

To control whether tool palettes are included when you click Clean Screen On/Off on the status bar, right-click the icon and mark or unmark Tool Palettes.

To display or hide multiple items at once — tool palettes, drawing tabs, the ribbon and quick access toolbar (for versions that have a ribbon) — type `showpalettes` or `hidepalettes` in the command bar.



To auto hide the Tool Palettes pane

- 1 Do one of the following:
 - Right-click the Tool Palettes pane title bar and choose Auto Hide.
 - On the Tool Palettes pane when it is undocked, click Auto Hide ().
 - On the Tool Palettes pane when it is undocked, click the menu (), and choose Auto Hide.

To move the Tool Palettes pane

- Click and drag the title bar of the Tool Palettes pane to the desired location in the drawing window.

To dock or undock the Tool Palettes pane

- 1 On the Tool Palettes pane when it is undocked, click the menu (), and choose Dockable if it is unchecked.
- 2 Do one of the following:
 - Double-click the title bar of the Tool Palettes pane to dock or undock it.
 - Drag the Tool Palettes pane title bar to the far right or left of the drawing window to dock it, or drag it to a different area of the drawing window to undock it.
 - On the Tool Palettes pane when it is undocked, click the menu (), and choose Fix to Left to dock the palette on the left side of the drawing window, or choose Fix to Right to dock the palette on the right side.

Tool palettes are customizable.

You can change the transparency of the Tool Palettes pane; add, delete and rearrange tool palettes; and add new customized commands and blocks. For more details, see “Customizing tool palettes” on page 864.

Displaying and hiding menus

Related commands are grouped together on menus. You can use the menus with or without the ribbon.

To customize the display of menus

- 1 Right-click anywhere in the menus or ribbon.
- 2 Choose Show Menu, then choose the menus you want to display or hide.

There are more ways to customize menus.

You can use the Customize UI command or manually edit the .mnu file for the menus. For more details, see “Customizing menus” on page 869. Also see “Working with user interface customization files” on page 916.

To display or hide menus when using the ribbon

For PTCAD versions that have a ribbon.

- 1 Right-click anywhere in the menus or ribbon.
- 2 Choose Menu Bar.

Menu display can also be affected by Clean Screen On/Off on the status bar.

To control whether menus are included when you click Clean Screen On/Off on the status bar, right-click the icon and mark or unmark Menu Bar.

Displaying and hiding toolbars

When you start PTCAD the first time, multiple toolbars are displayed. PTCAD provides more than a dozen toolbars, which you can customize by adding and deleting tools. You can also move and resize toolbars, and you can create new toolbars. You can use a shortcut menu to display or hide toolbars and to lock or unlock toolbar locations.

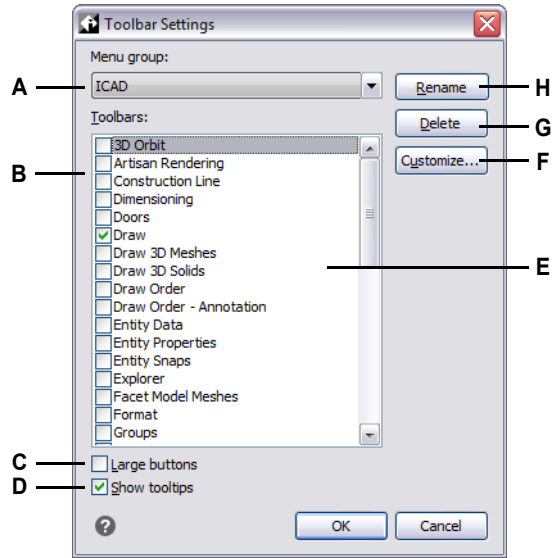
Toolbars are displayed large or small. You can choose to display or hide tooltips, which provide descriptions of tools that display when you pause the cursor over them. Go to View > Toolbars to make these selections.

Toolbars are either docked or floating. A floating toolbar can be located anywhere on the screen and can be resized. A docked toolbar cannot be resized and is attached along one of the edges of the drawing window.

- To dock a toolbar, drag it to the perimeter of your drawing; to undock it, drag it away from the perimeter.
- To position a toolbar in a docking area without docking it, press Ctrl while you drag it.
- To move a toolbar, drag it to a new location.
- To resize a toolbar, move your cursor to the edge until it changes to a resize arrow, and then drag.
- To lock toolbars in their current locations, right-click the head of any toolbar or menu, then choose Lock Location > Floating Toolbars and/or Docked Toolbars.

To choose which toolbars to display

- 1 Do one of the following:
 - On the ribbon, choose View > Toolbars (in Interface).
 - On the menu, choose View > Toolbars.
 - Right-click anywhere on a visible toolbar (docked, undocked, or the toolbar area at the top of the window) to display the toolbar shortcut menu, and then choose Toolbars. You can also select the toolbars you want displayed directly on the shortcut menu.
 - Type *toolbar* or *tbconfig* and then press Enter.
- 2 In the Toolbar Settings dialog box, mark the toolbars you want displayed, and then click OK.



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| <p>A Select the menu group that contains the toolbar you want to modify.</p> <p>B Mark to display a toolbar; clear to hide the toolbar.</p> <p>C Select to display large tools; clear to display small tools.</p> <p>D Select to display tooltips when you hover over tools on the toolbar; clear to not display tooltips.</p> | <p>E Highlight the toolbar name that you want to rename or delete.</p> <p>F Click to open the Customize User Interface dialog box and customize toolbars.</p> <p>G Click to delete the selected toolbar (does not delete all checked toolbars).</p> <p>H Click to type a new name for the selected toolbar (does not rename all checked toolbars).</p> |
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Toolbar display can also be affected by Clean Screen On/Off on the status bar.

To control whether all chosen toolbars are included when you click Clean Screen On/Off on the status bar, right-click the icon and mark or unmark Toolbars.

Using the command bar

The command bar is a dockable window in which you type PTCAD commands and view prompts and other program messages. By default the command bar displays the three most recent lines of prompts, but you can extend the window to display more lines. Move or resize the command bar by dragging it.

To display or hide the command bar

- On the ribbon, choose View > Command Bar (in Display).
- On the menu, choose View > Display > Command Bar.
- Type *cmdbar*, then choose Toggle.
- Press Ctrl+9.

Command bar display can also be affected by Clean Screen On/Off on the status bar.

To control whether the command bar is included when you click Clean Screen On/Off on the status bar, right-click the icon and mark or unmark Command Bar.

When the command bar is docked or floating, you can drag the top or bottom of the window to change the number of lines of text it displays. You can dock the command bar at the top, bottom, left, or right of the drawing. You can auto-hide and auto-display the command bar in its current location by clicking its pin button.

Colors of command bar items can be customized.

To change the color of various parts of the command bar, use the Options command. For details, see “Setting colors of the main window” on page 840.

When you type in the command bar, PTCAD suggests names of matching commands as you type. The suggested names appear in an AutoComplete window. Simply select the desired command in the list. In addition to command names, suggestions can include names of external commands, system variables, aliases, and LISP functions.

To turn on AutoComplete for typing in the command bar

- 1 Do one of the following:
 - On the menu, choose View > Display > AutoComplete.
 - Right-click the command bar and then choose AutoComplete.
 - Type *autocomplete*, press Enter, then choose AutoComplete.
 - Choose Tools > Options, click the Display tab, then mark Enable AutoComplete.

When turned on, the AutoComplete window automatically displays when you type in the command bar, and then automatically closes when you activate a command.

For more details about customizing settings for the AutoComplete window, see “Customizing how suggestions display in the command bar” on page 834.

Using the status bar

If you elect not to use the command bar, the status bar displays information about the selected command or tool. It also displays the current cursor coordinates, the name of the current layer, mode settings, and other information about current settings.

In addition to displaying information, the status bar is a quick way to access many features. You can click status bar items to make changes, and right-click items to display short-cut menus that allow you more choices.

To change the items that display on the status bar

- 1 Right-click an empty area of the status bar.
- 2 Select an item to add or remove it.

To change the setting of an item on the status bar

- 1 Do one of the following:
 - Double-click the status bar item.
 - Right-click the status bar item you want to change, and then select the desired setting.

To display or hide the status bar

- 1 Do one of the following:
 - On the ribbon, choose View > Status Bar (in Display).
 - On the menu, choose View > Display > Status Bar.
 - Type *statbar* and then press Enter.

Using view and visual style controls

The view and visual style controls display in the upper-left corner of the current viewport and allow you to easily apply a different view and visual style to the model.

To display or hide the view and visual style controls

- 1 Do one of the following:
 - On the ribbon, choose View > View Control (in Interface).
 - On the menu, choose View > Display > View Control.
 - Type *vpcontrol*, press Enter, then type *on* or *off*, and press Enter.

To use the view control

- From the list of views, choose the one you want to apply to the model.
For more details about named views, see “Creating and switching to named views” on page 690.

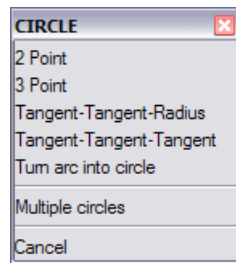
To use the visual style control

- From the list of visual styles, choose the one you want to apply to the model.
The list of visual styles depends on your graphics device. For more details, see “Customizing graphics device options for display performance” on page 846.

Using prompt boxes

PTCAD commands often provide several options. These options appear in the status bar or command bar and can also optionally appear in a prompt box (called a *context menu* in AutoCAD) initially displayed in the upper right corner of the screen.

You can move the prompt box by dragging it; future prompt boxes will appear where you last placed it. The options appear as menu selections. Choose the appropriate option by selecting it in the prompt box. To close a prompt box, click the Close box.



Command options appear in a prompt box.

The prompt box is hidden by default. To turn on the display of prompt boxes, choose Tools > Options, click the Display tab, and then click the Display Prompt Boxes check box.

Using workspaces


Workspaces are a convenient way to customize the display of commands in the PTCAD main window. Each workspace stores visibility settings for the following:

- Menus — Visibility settings are saved for each top-level menu (for example, the Draw menu) and each nested menu (for example, the Draw > Arc submenu).
- Toolbars — Visibility settings are saved for each toolbar, its orientation at the top, bottom, left, right, or floating, number of button rows, x-coordinate, and y-coordinate.
- Ribbon — For PTCAD versions that have a ribbon, visibility settings are saved for whether the ribbon itself displays, along with visibility settings for each ribbon tab (for example, the Home tab) and each panel (for example, Home > Modify).


Use a shortcut to make more space for drawing in the PTCAD main window.

Click Clean Screen On/Off on the status bar to hide multiple user interface items the same time. To customize what hides (or displays), right-click the icon and make your selections for the next time you click the icon.



To create a workspace

- 1 Organize the menus, toolbars, and ribbon the way you want them.
- 2 Do one of the following to choose Save Workspace (- On the ribbon, choose Tools > Save Workspace (in Manage).
- On the menu, choose Tools > Save Workspace.
- On the status bar, click Switch Between Workspaces, then choose Save Current As.
- Type `wssave`, press Enter.
- Type `workspace`, press Enter, then choose SaveAs.
- 3 Enter the name of the new workspace.
- 4 Click OK.



To save the current visibility settings to an existing workspace

- 1 Organize the menus, toolbars, and ribbon the way you want them.
- 2 Do one of the following to choose Save Workspace (- On the ribbon, choose Tools > Save Workspace (in Manage).
- On the menu, choose Tools > Save Workspace.
- On the status bar, click Switch Between Workspaces, then choose Save Current As.
- Type `wssave`, press Enter.
- Type `workspace`, press Enter, then choose SaveAs.
- 3 Select an existing workspace name from the list.
- 4 Click OK.

To switch to a workspace

- 1 Do one of the following:
 - On the ribbon, choose the desired workspace from the Workspaces list (.
 - On the status bar, click Switch Between Workspaces () , then choose the desired workspace.
 - On the Workspaces toolbar, choose the desired workspace.
 - Type `workspace`, press Enter, choose Set Current, then enter the name of the desired workspace.

The My Workspace tool (Workspaces toolbar) loads a specific workspace.

Click the My Workspace tool () to switch to your preferred workspace. To set which workspace loads when you click the My Workspace tool, click () on the status bar, choose Workspace Settings, then choose the workspace in My Workspace.

Selecting commands

Select commands using any of these methods:

- Choose a command from the ribbon.
- Choose a command from a menu.
- Click a tool in the Tool Palettes pane.
- Click a tool in a toolbar.
- Type the command in the command bar.

Some commands remain active until you end them, so you can repeat an action without having to select a command repeatedly. You can end a command by pressing Esc.

Using commands

You can use most commands while another command is active. For example, while drawing a line, you may want to use the Pan command to move the drawing across the screen to select the endpoint of the line. You can also change the settings of drawing aids such as snap or grid while other commands are active.

After started, many commands have options that you can select in the command bar, status bar, or prompt box.

Starting commands using the ribbon

To start a command from the ribbon, choose it from the list of options available. If a command has an arrow, click the arrow to see related commands. For example, choose Home, then click the arrow for Array (in Modify) to choose the 3D Array command.

Starting commands using tool palettes

To start a command from a tool palette, click a tool and respond to the prompts.

If the palette contains several tools, you might need to use the scroll bars to see the full list of available tools for that palette.

Starting commands using toolbars

To start a command from a toolbar, click a tool and respond to the prompts.

The available toolbars depend on the experience level that is set.

To change the current experience level, choose Tools > Options, and then click the General tab.

Some tools, such as Line or Arc, contain flyouts, which are options either for using the basic tool with different methods or that contain other, related tools. Flyouts are indicated by a small triangle in the lower right corner of a tool. To display a flyout, click the tool and hold down the mouse button. To select a tool from a flyout, point to the tool you want and then release the mouse button. The tool you select on the flyout becomes the default tool on the toolbar.

Starting commands using menus

To start a command from a menu, choose it from the list of menu options available.

The available menu options depend on the experience level that is set. To change the current experience level, choose Tools > Options, and then click the General tab.

Starting commands using the command bar

Type a command, and then press Enter. If the command bar is displayed, the command you typed appears there. If the command bar is not displayed, the command you typed appears in the status bar.

Quickly start commands using the command bar by copying, cutting, and pasting text.

In the command bar or Prompt History window, right-click to select various cut, copy, and paste commands.

Repeating a command

You can repeat a command you used previously without having to reselect it.

To repeat the command you just used

- 1 Do one of the following:
 - Press the spacebar.
 - Press Enter.
 - Click the right mouse button in the drawing.

To repeat commands you used previously

- 1 Do one of the following:
 - In the command bar or Prompt History window, press Ctrl and double-click the previous command text.
 - In the command bar or Prompt History window, right-click, choose Recent Commands, and then choose the desired command.
 - Use the Up and Down arrows to display previously used commands, if Use Up/Down Arrows for Command History Navigation is selected in Tools > Options.

Run a command multiple times using the command bar.

If you are using the command bar to type commands, you can type multiple before starting some commands (Circle, Arc, and Rectangle, for example) to repeat a command indefinitely. When you are done with a command, press Esc.

Nesting a command

If you are working in the command bar, you can use another command from within a command, called *nesting*. To use a command inside an active command, type an apostrophe before you type the command, such as *'circle*, *'line*, or *'pyramid*. You can nest commands indefinitely in PTCAD. Many menu and toolbar macros work this way by default; for example, select color, reference grid, zoom, and snap. When you have finished with the nested command, the original command resumes.

Modifying a command

If you are working in the command bar, there are special ways that you can modify a command as you work. You can modify the active command using any of the following options:

- **Entity snaps** — Type an entity snap command, such as *nearest* or *midpoint*, to enable a one-time entity snap for a single selection. You can also use a one-time entity snap to override a running entity snap.
- **Extension snaps** — Type *int* after selecting a command, such as Line or Circle, to enable a one-time snap to the logical location where two entities would intersect if they were of infinite length. Type *app* to enable a similar one-time snap if the extensions would not intersect in three-dimensional space but would intersect in the current view.
- **Midpoint snaps** — Type *m2p* or *mtp* to enable a one-time snap to the midpoint of two points that you specify.

Using the Prompt History window

The Prompt History window displays a history of the commands and prompts issued since you started the current session of PTCAD. By default, the program tracks up to 256 command lines. There is no limit to the number of command lines you can track, but program performance may be degraded if you choose to track an excessively high number of lines.

To display or close the Prompt History window

- 1 Do one of the following to choose Prompt History Window:
 - On the ribbon, choose View > Prompt History Window (in Interface).
 - On the menu, choose View > Display > Prompt History Window.
 - Type *pmthist* and then press Enter.
 - Press F2.

To view entries in the Prompt History window

- 1 Do one of the following:
 - Use the scroll bars.
 - Use the Up and Down arrows to display previously used commands, if Use Up/Down Arrows for Command History Navigation is selected in Tools > Options.

To copy or paste text in the Prompt History window

- 1 If copying text, do one of the following:
 - Highlight text using your mouse.
 - Press Ctrl + Shift + arrow keys to highlight text.
- 2 Right-click and choose whether to copy or paste.

You can also copy the entire command history or the last command line.
Choose Copy History or Copy Last Line.

To change the number of command lines to track

- 1 Do one of the following:
 - Choose Tools > Options, and then click the Display tab.
 - Type *options* and then press Enter. Click the Display tab.
- 2 In the Command Lines To Track field, type the number of command lines you want to display, and then click OK.

Using mouse shortcuts

You can use your mouse, often combined with the keyboard, to start commands and perform other actions.

Mouse shortcuts

Shortcut	Action
Hold and drag left mouse button	Move selected entities when not in a command
Ctrl + Hold and drag left mouse button	Copy and move selected entities when not in a command
Ctrl + Hold and drag right mouse button	Constrained Z Orbit command
Ctrl + Left-click mouse	Cycle-select entities located below the cursor
Ctrl + Shift + Hold and drag left mouse button	Real-Time Zoom command
Ctrl + Shift + Hold and drag middle mouse button (wheel)	Free Orbit command
Alt + Shift + Hold and drag middle mouse button or Shift + Hold and drag middle mouse button	Constrained Orbit command
Shift + Left-click mouse	Deselect entities
Shift + Right-click mouse	Entity snap shortcut menu
Hold and drag left mouse button	Move selected entities
Right-click mouse	Display shortcut menu for the selected entity
Rotate mouse wheel	Zoom In and Zoom Out commands; if over the ribbon, scrolls ribbon tabs
Hold mouse wheel, then move mouse	Real-Time Pan command
Ctrl + Rotate mouse wheel when command bar is active	Zoom in and out of the command bar

Using keyboard shortcuts

Keyboard shortcuts are key combinations that start commonly used PTCAD commands. The following table describes function key shortcuts, but there are many more shortcuts available. For a full list, choose Tools > Customize UI and choose Keyboard Shortcuts in the Current Customization list.

Function key shortcuts

Command	Typed Entry	Shortcut	Description
Coordinate	COORDINATE	F6, Ctrl+D, Ctrl+I	Switches coordinate display between On, Off, and Angle/Distance.
Entity Snap	ESNAP	F3, Ctrl+F	Turns entity snaps on and off.
Entity Snap Tracking	ENTTRACK	F11	Switches entity snap tracking on and off
Exit	EXIT	Alt+F4, Ctrl+Q	Closes all drawings and exits PTCAD.
Grid	GRID	F7, Ctrl+G	Turns the reference grid display on and off.

Function key shortcuts

Command	Typed Entry	Shortcut	Description
Grid Snap	SNAP	F9, Ctrl+B	Turns grid snap settings on and off.
Help	HELP	F1	Starts online Help.
Isometric Plane	ISOPLANE	F5, Ctrl+E	Switches the isometric plane between Top, Right, and Left.
Orthogonal	ORTHOGONAL	F8, Ctrl+L	Switches the orthogonal mode on and off.
Polar Tracking	POLARTRACK	F10, Ctrl+U	Switches polar tracking on and off
Prompt History Window	PMTHIST	F2	Turns the Prompt History window on and off.
Tablet	TABLET	F4	Controls the tablet configuration.
VBA Editor	VBA	Alt+F11	Opens the Visual Basic for Applications editor so that you can create or modify a VBA macro.
Window Close	WCLOSE	Ctrl+F4	Closes the active drawing window.

Using scripts

The PTCAD Script Recorder captures and saves many of your actions so you can play them back. After you enable the Script Recorder, all commands and options that you type in the command bar are recorded until you type a command to stop recording. When you run the script, the program carries out the recorded commands in succession.

Correcting mistakes

PTCAD tracks the commands you use and the changes you make. If you change your mind or make a mistake, you can undo, or reverse, the last action or several previous actions. You can also redo any actions that you reversed.



- A** Click Undo to reverse the last action.
- B** Click Redo to reverse the previous undos.

You can undo and redo multiple actions by choosing Undo and Redo from the Quick Access toolbar of the ribbon to view and select a range of actions. You can also type *undo* and then specify the number of actions to undo.

Many commands also have an Undo option that you can choose (or press Ctrl + Z) to undo the last action, for example, choose Undo when creating a line to remove the previously specified point.

Customizing PTCAD

You can tailor many aspects of PTCAD to better suit your needs. For example, you can easily create and modify menus using the Customize User Interface (*cui*) command or create and modify toolbars by simply dragging and dropping icons. stores your customized settings in a .cui file; you can also store them in a profile.

PTCAD supports the most important AutoCAD customization files, including linetypes, hatch patterns, text fonts, the unit conversion file, menus, toolbars, and aliases. In addition, PTCAD unifies many AutoCAD customization functions with a single *cui* command: command aliases, menus, toolbars, keyboard shortcuts, and double-click actions. And for versions of PTCAD that have a ribbon, this list for the *cui* command also includes ribbons, quick access toolbars, application buttons, and contextual menus. For more details about customizations, see Chapter 17, “Customizing PTCAD.”

You can also add custom programs written in any of several programming languages, including the following:


- ODA SDKs and IRX (similar to the Autodesk® ARX language)
- LISP (the program’s Autodesk® AutoLISP-compatible language)
- SDS (similar to the Autodesk® ADS language)
- DCL (Dialog Control Language)
- DIESEL
- Microsoft® Visual Basic for Applications (VBA)
- .NET

For more details about custom programs, see “Programming PTCAD” on page 944.

Migrating customizations from a previous PTCAD version


Customizations that you made in a previous PTCAD version can be easily migrated to your PTCAD current version. You can merge or replace custom settings made to workspaces and customization files (.cui files), including menus, the ribbon if your version supports it, toolbars, keyboard shortcuts, aliases and double-click actions.

To migrate customizations from a previous PTCAD version

- 1 Use one of the following methods to choose Migrate UI (- On the ribbon, choose Tools > Migrate UI (in Manage).
- On the menu, choose Tools > Migrate UI.
- Type *migrate* and then press Enter.
- 2 Enter the file path and filename of the .cui file that contains the customization you want to migrate to the current PTCAD version. If necessary, type ? and press Enter to see a list of available .cui files.
- 3 Choose one of the following:
 - **Replace with given** — Replaces the current customization entirely with the one from the selected .cui file.
 - **Merge prefer given** — Merges customizations with a preference to the previous installation if conflicts occur.
 - **Merge prefer current** — Merges customizations with a preference to the current installation if conflicts occur.

Getting online Help

PTCAD includes online Help, which contains task-oriented topics, a command reference, a system variables reference, and a programming language reference.

You can display online Help () in any of these ways:

- On the ribbon, choose Help, then choose a command.
- On the Standard toolbar, click Help.
- Press F1.
- Choose a command from the Help menu.
- Click the question mark in a dialog box.
- Type *help* in the command bar.

Saving a drawing

You can save your drawing at any time.

Use one of the following methods to choose Save (📁):

- On the Quick Access toolbar of the ribbon, click Save.
- On the ribbon Application button, choose Save.
- On the Standard toolbar, click Save.
- On the menu, choose File > Save.
- Type *save* and then press Enter.

When you save a drawing the first time, the program displays the Save Drawing As dialog box so that you can choose a directory and type a name for the drawing. To save the drawing later using another name, do the following:

- Choose File > Save As and type the new name.

Exiting PTCAD

When you have finished working in PTCAD:

- On the ribbon Application button, choose Exit.
- On the menu, choose File > Exit.

Working with drawings

CAD drawings help you organize information for greater efficiency. With PTCAD, you can draw entities representing different types of information on various layers and use those layers to control color, linetype, and visibility. PTCAD also includes drawing aids that help you draw accurately.

This section explains setting up drawings and using built-in drawing aids, including how to:

- Create new drawings, open existing drawings, and save changes to drawings.
- Use aids such as the grid, snap, and orthogonal settings to draw accurately.

Topics in this chapter


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Creating a new drawing

When you start PTCAD, the program automatically creates a new drawing based on a template drawing, `icad.dwt`. This template includes predefined settings such as drawing units, text size, print style table type, and drawing area. You can either use these settings or change them according to your needs. There is nothing unique about a template drawing. You can use any drawing as a template for future drawings.

You can save many steps by basing a new drawing on an existing template (.dwt file). By doing this, a new drawing will contain all the settings and entities you need. When you open a new drawing from your custom template, you can modify existing settings and delete any entities that you don't need. When you save a drawing that was created using a template, you do not change the template.

To create a new drawing

- 1 Do one of the following to choose New (

To create a new drawing based on a template

- 1 Do one of the following to choose New (

Opening a drawing

You can open drawing (.dwg) files, Drawing Exchange Format (.dxf) files, Design Web Format (.dwf) files, and drawing template (.dwt) files.

You can also open and check drawings that you suspect are damaged.

Opening an existing drawing

You can open any of these drawing files:

- Standard drawing files with a .dwg extension.
In addition to your own drawing files, you can open and use one of the sample drawings that are included with PTCAD.
- Drawing Exchange Format files with a .dxf file extension.
- Design Web Format files with a .dwf file extension.
- Drawing templates with a .dwt file extension.

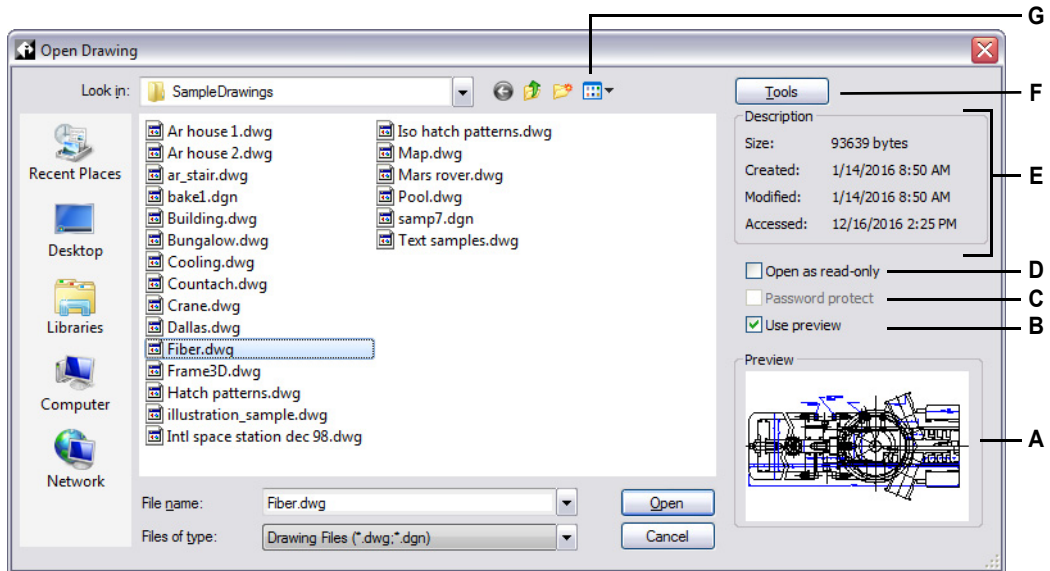
To open an existing drawing

- 1 Use one of the following methods to choose Open (📂🔍):
 - On the Quick Access toolbar of the ribbon, click the Open tool.
 - On the menu, choose File > Open.
 - On the Standard toolbar, click the Open tool.
 - Type *open* and then press Enter.
- 2 In Files of Type, choose the type of file you want to open.
- 3 Choose the folder containing the desired file.
- 4 Do one of the following:
 - Choose the drawing you want to open, and then click Open.
 - Double-click the drawing you want to open.

If the drawing requires a password, enter the password, click OK to verify the password, and then click Open again.

You can also open drawings while browsing files on your computer.

Simply double-click the file or drag it to the drawing area in PTCAD. Using programs that came with your operating system, such as Windows File Explorer or My Computer, you can find the drawing you want by viewing thumbnail images of the drawing files as you browse them. If needed, choose Tools > Options and on the General tab, click Set Files Association to specify which filetypes are associated with PTCAD.



- A If a thumbnail image exists in the selected drawing, displays an image of the drawing before you open it.
- B Turns the drawing preview on or off.
- C Unavailable when opening drawings; available only when saving drawings.
- D Opens the drawing as read-only to prevent making changes to the file.
- E Displays a description of the file size, creation date, and other information about the drawing.
- F Click to see more options for working with files.
- G Defines how drawings display in the list, including file details and thumbnail images.

The most recently opened drawings are tracked for easy opening.

To quickly open a drawing file that you recently used, choose File > filename, or if your program version has a ribbon, choose the filename from the Application button or Start Page.

Opening a damaged drawing

Files can become damaged for many reasons. For example, if you are working on a drawing during a power outage, a system crash, or a hardware failure, your drawing file may become damaged. PTCAD allows you to open and check damaged files to attempt file recovery.


Recovering a file attempts to open one of the following file types:

- Standard drawing files with a .dwg extension.
- Drawing Exchange Format files with a .dxf file extension.
- Design Web Format files with a .dwf file extension.
- Drawing templates with a .dwt file extension.

You can also audit any open file to check it for errors. You specify whether you want PTCAD to fix any errors that are found automatically. PTCAD fixes as many errors as possible and any errors that cannot be fixed are reported as “Ignored” in the Prompt History window.


In addition to recovering and auditing drawings, if you open a drawing that does not display all entities, it might be because those entities are assigned invalid z-coordinates. Use the Set Z All command to set all z-coordinates to a new value so that all missing entities are then properly displayed.

To open a damaged file

- 1 Do one of the following to choose Recover ():
 - On the ribbon Application button, choose Drawing Utilities > Recover.
 - On the menu, choose File > Recover.
 - Type *recover* and then press Enter.
- 2 In Files of Type, choose the type of file you want to recover.
- 3 Choose the directory containing the damaged file.
- 4 Choose the damaged file you want to recover.
- 5 Click Open.

If you want to check all drawings for errors automatically when you open them, choose Tools > Options > General tab and mark the check box for Open Drawings using Recover.

To check a drawing file for errors

- 1 With the drawing open that you want to check, do one of the following to choose Audit (

An ASCII file describes the audit.

If the AUDITCTL system variable is set to On and errors are found during a file recovery or audit, an ASCII file is created that describes the audit. The ASCII file is saved in the same folder as the audited drawing and has the same name as the drawing file, but with an .adt extension.

To fix a drawing that contains invisible entities assigned invalid z-coordinates

- 1 With the drawing open that you want to fix, do one of the following to choose Set Z All (


Setting up a drawing

You can specify individual settings when you create a new drawing or when you modify settings in a drawing created from a template.

Setting the current layer

Layers are like the overlays that you use in manual drafting. You use layers to organize different types of drawing information. Every drawing has at least one layer, the default layer, named “0.” Your drawing can also contain an unlimited number of additional layers. When you create an entity, it is created on the current layer.

To set the current layer

- 1 Do one of the following to choose Explore Layers (- On the ribbon, choose Home > Explore Layers (in Layers) or choose Tools > Layers (in Explorer).
- On the menu, choose Format > Explore Layers or choose Tools > Explorer > Explore Layers.
- On the Format toolbar, click the Explore Layers tool .
- Type *explayers* and then press Enter.
- Type *la* and then press Enter.
- 2 Double-click the layer name that you want to make current.

Use the status bar shortcut.

On the status bar, right-click the current layer control, and from the list, select the layer you want to make current.

Use the layer of an entity you select to set the current layer.

Type LAYMCUR in the command bar, and select the entity with the layer you want to be current.


Setting the current entity color

An entity's color determines how it is displayed and, if you are using a color printer, how it prints. Entities are created in the current color.

When you open a new drawing, entities are created in the color BYLAYER, which adopts the color of the current layer. Initially, layer 0 is both the only layer and the current layer. Its default color is white, so your entities appear as white.

There are index colors, which contain two additional color properties that are often referred to as colors, true colors, and color book colors. The two additional color properties are BYLAYER and BYBLOCK. These color properties cause an entity to adopt the color either of the layer or of the block in which it is a member.

To set the current entity color

- 1 Do one of the following to choose Drawing Settings ():
 - On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
 - On the menu, choose Tools > Drawing Settings.
 - On the Tools toolbar, click the Drawing Settings tool.
 - Type *dsettings* and then press Enter.
- 2 Click the Entity Creation tab.
- 3 Click Select Color.
- 4 In the Color dialog box, click one of the following tabs and select a color:
 - **Index Color** — Click BYBLOCK, BYLAYER, or one of the 255 index colors. You can also type the color number in the Index box.
 - **True Color** — Click a basic color, click a color in the color palette, enter the Hue, Saturation, and Luminance (HSL) values, or enter the Red, Green, Blue (RGB) values. There are more than 16 million true colors from which you can choose.
 - **Color Books** — Select a color book from the list, then click a color. You can select Show Only Color Book Colors Used in Drawing to limit the selection to only those color book colors that are used in the current drawing.
- 5 Click OK.
- 6 Click OK again.

Use the status bar shortcut.

On the status bar, right-click the current color, and select from the list the color you want to use for new entities. You can also click Select Color to choose additional colors.

For more details about using color in the many aspects of your drawing, see “Working with colors” on page 71 in this chapter.

Setting the current linetype


Linetypes help convey information. You use different linetypes to differentiate the purpose of one line from another. A linetype consists of a repeating pattern of dots, dashes, or blank spaces. Linetypes determine the appearance of entities both on the screen and when printed. By default, every drawing has at least three linetypes: CONTINUOUS, BYLAYER, and BYBLOCK. Your drawing may also contain an unlimited number of additional linetypes.

When you create an entity, it is created using the current linetype. By default, the current linetype is BYLAYER. PTCAD indicates that the entity linetype is determined by the current layer's linetype by assigning the BYLAYER property as the default linetype setting. When you assign BYLAYER, changing a layer's linetype changes the linetype of all the entities assigned that layer (if they were created using the linetype BYLAYER).

You can also select a specific linetype as the current linetype, which overrides the layer's linetype setting. Entities are then created using that linetype, and changing the layer linetype has no effect on them.

As a third option, you can use the linetype BYBLOCK property, in which case new entities are drawn using the CONTINUOUS linetype until you group them into a block. The entities then inherit the block's linetype setting when you insert the block into a drawing.

To set the current linetype

- 1 Do one of the following to choose Drawing Settings ():
 - On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
 - On the menu, choose Tools > Drawing Settings.
 - On the Tools toolbar, click the Drawing Settings tool.
 - Type *dsettings* and then press Enter.
- 2 Click the Entity Creation tab.
- 3 In the Linetype list, choose the linetype that you want to make current.
- 4 Click OK.

Use the status bar shortcut.

On the status bar, right-click the word BYLAYER for the current linetype, click Properties, and then choose the linetype that you want to make current.


Setting the linetype scale

You can specify the linetype scale. The smaller the scale, the more repetitions of the linetype pattern are generated per drawing unit. For example, a linetype pattern is defined as a sequence of dashed lines and open spaces, each 0.25 units long. The linetype scale uses the drawing scale factor to determine the length. A scale factor of 0.5 would reduce the length of each line and space to 0.125 units; a scale factor of 2 would increase the length of each to 0.5 units.


Note that setting the linetype scale too large or too small may result in a line pattern looking like a solid line, depending on what the scale view is or at what scale the drawing is printed.

You can control a new entity's individual linetype scale factor as well as the overall or global scale factor applied to all the entities in the drawing. If you use annotation scaling for entities such as text and dimensions, you can also set up linetype scaling to coordinate with various annotation scaling scenarios.

To set the current individual linetype scale

- 1 Do one of the following to choose Drawing Settings (- On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
- On the menu, choose Tools > Drawing Settings.
- On the Tools toolbar, click the Drawing Settings tool.
- Type *dsettings* and then press Enter.
- 2 Click the Entity Creation tab.
- 3 In the Linetype Scale field, type or choose the linetype scale that you want to make current.
- 4 Click OK.

To change the global linetype scale

- 1 Do one of the following to choose Drawing Settings (- On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
- On the menu, choose Tools > Drawing Settings.
- On the Tools toolbar, click the Drawing Settings tool.
- Type *dsettings* and then press Enter.
- 2 Click the Entity Creation tab.
- 3 In the Global Linetype Scale field, type or choose the global linetype scale that you want to change.
- 4 Click OK.

To set up linetype scaling to be the same in model space and paper space

- 1 Set the global linetype scale (mentioned previously) to the drawing scale. For example, enter .25 for the scale factor 1/4" - 1".
- 2 Type *PSLTSCALE* and press Enter, then enter 0 to turn this system variable off. Customized linetype scaling specific to paper space will be turned off.
- 3 Type *MSLTSCALE* and press Enter, then enter 0 to turn this system variable off. Customized linetype scaling specific to model space will be turned off.

To set up linetype scaling to be different in model space and paper space

- 1 Set the global linetype scale (mentioned previously) to 1. This sets the default linetype scale factor to be the same as the drawing scale.
- 2 Type *PSLTSCALE* and press Enter, then enter 1 to turn this system variable on. Customized linetype scaling specific to paper space will be turned off.
- 3 Type *MSLTSCALE* and press Enter, then enter 1 to turn this system variable on. Customized linetype scaling specific to model space will be turned off.
- 4 For drawings that also use annotation scaling, do the following:
 - On the status bar, right-click Annotations Scales List.
 - Choose the current annotation scale, for example 1/4" - 1". This makes the linetype scale set to the annotation scale.

Linetypes will be the same for all viewports that have the same annotation scale.

Setting up linetype scaling so it can be different in model space and paper space is the preferred method for drawings with details, profiles, or plan views that are not always the same scale. Paper space viewports will look and print as intended, and you can adjust the model space annotation scale for it to match the display in paper space. For more information about annotation scaling, see “Understanding scale factors” on page 62 in this chapter.

Setting the current lineweight

Lineweights help differentiate the purpose of one line from another. Lineweights determine how thick or thin entities appear both on the screen and when printed. Every drawing has these lineweights: DEFAULT, BYLAYER, BYBLOCK, and many additional lineweights in millimeters (or you can use inches).

When you create an entity, it is created using the current lineweight. By default, the current lineweight for a new entity is BYLAYER. This means that the entity lineweight is determined by the current layer. When you assign BYLAYER, changing a layer’s lineweight changes the lineweight of all the entities assigned that layer (if they were created using the lineweight BYLAYER).


You can also select a specific lineweight (or DEFAULT) as the current lineweight, which overrides the layer's lineweight setting. Entities are then created using that lineweight (or the DEFAULT lineweight), and changing the layer lineweight has no effect on them.

As a third option, you can use the lineweight BYBLOCK property, in which case new entities are drawn using the DEFAULT lineweight until you group them into a block. The entities then inherit the block's lineweight setting when you insert the block into a drawing.

If you choose a lineweight that is less than .025 millimeter, it displays as one pixel when you create your drawing. When you print your drawing, it prints at the thinnest lineweight that is available for your printer.

You cannot assign lineweights to planes, points, TrueType fonts, or raster images (if supported in your version of PTCAD).

To set the current lineweight

- 1 Do one of the following to choose Drawing Settings ():
 - On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
 - On the menu, choose Tools > Drawing Settings or choose Tools > Lineweight.
 - On the Tools toolbar, click the Drawing Settings tool.
 - Type *dsettings* and then press Enter.
- 2 Click the Entity Creation tab.
- 3 In the Lineweight list, choose the lineweight that you want to make current.
- 4 Click OK.

Use the status bar shortcut.

On the status bar, right-click the word BYLAYER for the current lineweight, and then choose the current lineweight. You can also double-click the word LWT to toggle the display of lineweights on and off.

Lineweights need to be turned on to be visible.

To see lineweights in your drawing, you may need to turn on their visibility. For details, see “Displaying lineweights” on page 251.

Setting the current print style

Print styles are used to change the appearance of an entity when it prints, without actually changing the entity in the drawing.


If your drawing uses named print style tables, you can specify a print style for any entity. Named print style tables contain print styles that you set up. If your drawing uses color-dependent print style tables, the print style is BYCOLOR, which cannot be changed. These types of print style tables determine printing requirements by the color assigned to an entity or layer. For details about converting a drawing that uses color-dependent print style tables to use named print style tables, see “Changing the print style table type of a drawing” on page 657.

When you create an entity in a drawing that uses named print style tables, the entity is created using the current print style. By default, the current print style is BYLAYER. When you assign BYLAYER, changing a layer’s print style changes the print style of all the entities assigned that layer if they were created using the print style BYLAYER.

You can also select a specific print style as the current print style, which overrides the layer’s print style setting. Entities are then created using that print style, and changing the layer print style has no effect on them.

As a third option, you can use the print style BYBLOCK, in which case new entities use the Normal print style until you group them into a block. The entities then inherit the block’s print style setting when you insert the block into a drawing.

To set the current print style in a drawing that uses named print style tables

- 1 Do one of the following to choose Drawing Settings ():
 - On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
 - On the menu, choose Tools > Drawing Settings.
 - On the Tools toolbar, click the Drawing Settings tool.
 - Type *dsettings* and then press Enter.
- 2 Click the Entity Creation tab.
- 3 In the Print Style list, choose the print style that you want to make current. If necessary, choose Other and then select a print style.
- 4 Click OK.

Use the status bar or command bar shortcut.

On the status bar, right-click the word BYLAYER for the current print style, click Other, and then choose the print style that you want to make current. Or, type printstyle to choose the current print style.


Setting drawing units

With PTCAD, you typically draw at full-size (1:1 scale), and then set a scale factor when you print or plot your drawing. Before you begin drawing, however, you need to determine the relationship between drawing units and real-world units.

For example, you can decide whether one linear drawing unit represents an inch, a foot, a meter, or a mile. In addition, you can specify the way the program measures angles. For both linear and angular units, you can also set the degree of display precision, such as the number of decimal places or smallest denominator used when displaying fractions. The precision settings affect only the *display* of distances, angles, and coordinates. PTCAD always stores distances, angles, and coordinates using floating-point accuracy.

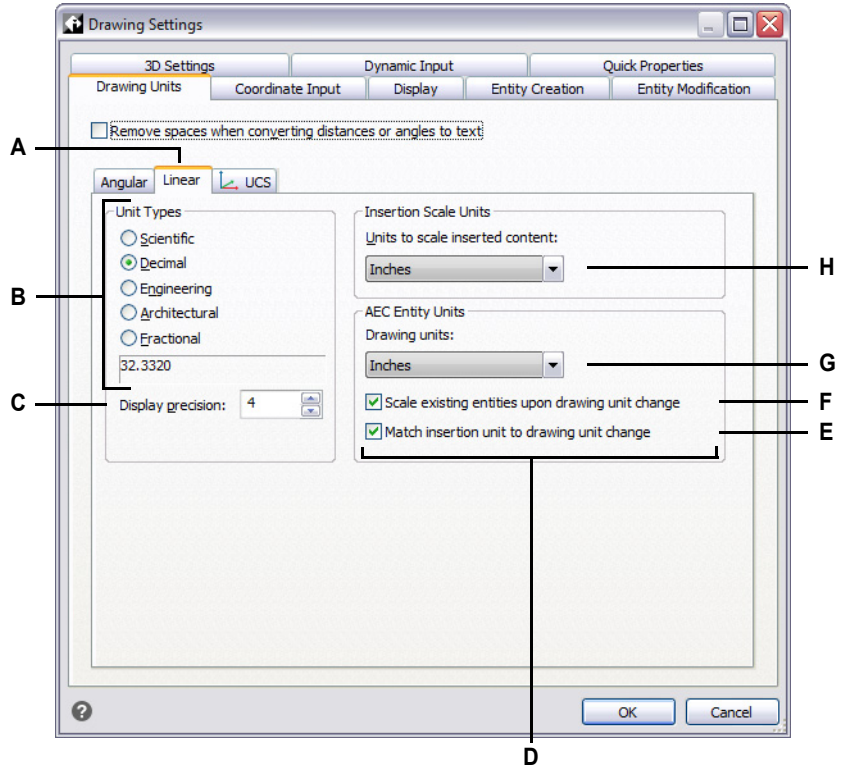
If your drawing contains Architecture, Engineering, and Construction (AEC) entities, you also set the AEC drawing units. To provide compatibility for your drawing with Autodesk Architecture, drawings with AEC entities require two drawing units: one for .dwg file compatibility and one for architecture object compatibility.

To set the linear drawing units

- 1 Do one of the following to choose Drawing Settings (- On the ribbon, choose Application button > Drawing Utilities; Application Button > Drawing Utilities > Drawing Units; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
- On the menu, choose Tools > Drawing Settings.
- On the Tools toolbar, click the Drawing Settings tool.
- Type *dsettings* and then press Enter.
- 2 Click the Drawing Units tab.
- 3 Click the Linear Units tab.
- 4 Under Unit Types, select a unit type.
- 5 For Display Precision, type the display precision according to the number of decimal places you want, or click the arrows to select it.


The box above this setting shows an example of the linear unit type at the current precision.
- 6 For Insertion Scale Units, select the units used when inserting blocks, external references and images from drawings that use different units than the current drawing.
- 7 For AEC Entity Units, select the drawing units to use for inserting and scaling AEC entities.
- 8 Mark Scale Existing Entities upon Drawing Unit Change to scale AEC entities in the current drawing if the AEC units change. If not marked, existing AEC entities are not scaled and the new AEC units affect only new AEC entities.
- 9 Mark Match Insertion Unit to Drawing Unit Change to make the Insertion Scale Units change to match AEC units if the AEC units change.

10 Click OK.



- A Determines the type of units you are controlling.
- B Select the type of linear units.
- C Choose the display precision for linear units.
- D Displays only if your program version supports working with AEC entities.
- E Mark to make the insertion scale units change to match AEC units if AEC units change.
- F Mark to scale AEC entities if AEC units change.
- G Select drawing units to use for inserting and scaling AEC entities.
- H Select the units used when inserting items from drawings that use different units than the current drawing.

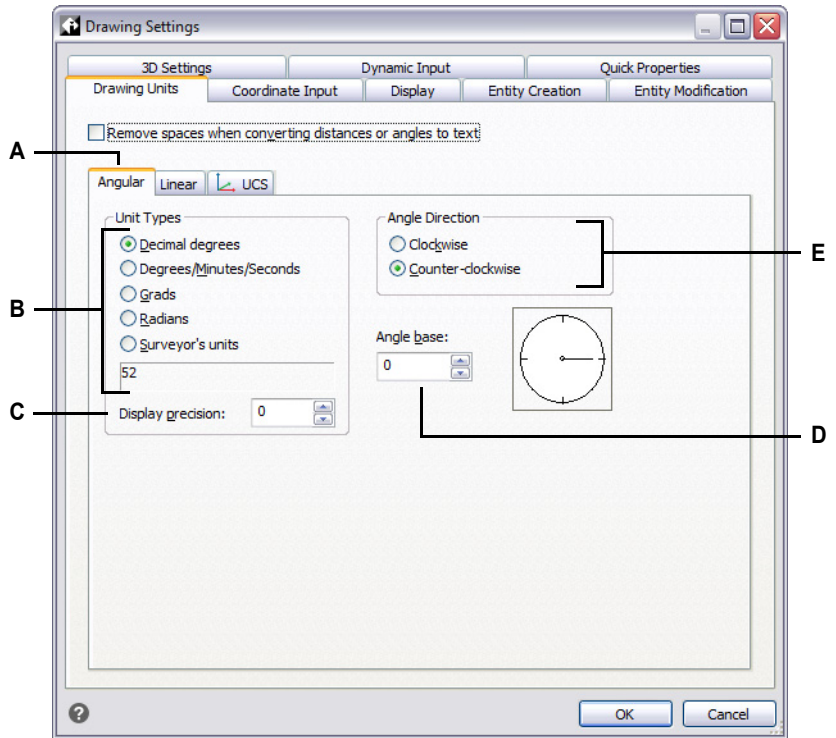
To set the angular drawing units

- 1 Do one of the following to choose Drawing Settings (- On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
- On the menu, choose Tools > Drawing Settings.
- On the Tools toolbar, click the Drawing Settings tool.
- Type *dsettings* and then press Enter.
- 2 Click the Drawing Units tab.
- 3 Click the Angular Units tab.
- 4 Under Unit Types, select a unit type.
- 5 Under Display Precision, type the display precision according to the number of decimal places you want, or click the arrows to select it.

The field above this setting shows an example of the angular unit type at the current precision.
- 6 Under Angle Direction, select the direction in which angles increase when you specify a positive angle value.
- 7 Under Angle Base, specify the compass location for the zero angle.

For example, by default, angle 0 is at the “three o’clock” or “east” position.

The adjacent icon shows the current location of the angle base.
- 8 Click OK.



- A** Determines the type of units you are controlling.
- B** Select the type of angular units.
- C** Choose the display precision for angular units.
- D** Select the angle base, the direction of the zero angle.
- E** Select the direction in which angles increase.

Understanding scale factors

Instead of drawing to a particular scale, you draw everything in the program full-size. Although it's a good idea to keep your scale factor in mind when setting up a drawing, you don't need to set the scale until you print it. For example, when you draw a mechanical part 40 inches in length with PTCAD, you actually draw it as 40 inches, rather than applying a scale factor as you draw. When you print your drawing, you can assign the scale at which the drawing is to print.

Scale, however, does affect the way a few elements such as text, arrows, or linetypes print or plot and even display in your drawing. You can set up annotation scaling to control the scale of entities such as text, arrows, and linetypes, or you can make manual adjustments when you first set up your drawing so that annotations print and display at the correct size. For example, when you draw text, you need to determine the text size so that when you print it later at a particular scale, the text height is correct.

After you determine the eventual scale of your finished drawing, you can calculate the scale factor for the drawing as a ratio of one drawing unit to the actual scale unit represented by each drawing unit. For example, if you plan to print your drawing at $1/8" = 1'-0"$, your scale factor ratio is 1:96 ($1/8" = 12"$ is the same as $1 = 96$). If you want your printed scale to be 1 inch = 100 feet, your scale factor ratio is 1:1200.

The following table shows some standard architectural and engineering scale ratios and equivalent text heights required to create text that measures 1/8-inch high when you print the drawing at the specified scale.

Standard scale ratios and equivalent text heights

Scale	Scale factor	Text height
$1/16" = 1'-0"$	192	24"
$1/8" = 1'-0"$	96	12"
$3/16" = 1'-0"$	64	8"
$1/4" = 1'-0"$	48	6"
$3/8" = 1'-0"$	32	4"
$1/2" = 1'-0"$	24	3"
$3/4" = 1'-0"$	16	2"
$1" = 1'-0"$	12	1.5"
$1\ 1/2" = 1'-0"$	8	1"
$3" = 1'-0"$	4	0.5"
$1" = 10'$	120	15"
$1" = 20'$	240	30"
$1" = 30'$	360	45"
$1" = 40'$	480	60"

Standard scale ratios and equivalent text heights

Scale	Scale factor	Text height
1" = 50'	600	75"
1" = 60'	720	90"
1" = 100'	1200	150"

You can use these scale factors to predetermine the size of your drawing to make sure that it fits on a specific size paper when you print it. You control the size of your drawing by the drawing limits. To calculate the drawing limits to match the size of your paper, multiply the dimensions of your paper size by your scale factor.

For example, if the paper you use to print measures 36 inches x 24 inches and you print your drawing at 1/8" = 1'-0" (in other words, using a scale factor of 96), the size of your drawing measured in drawing units is 36 x 96 (or 3,456 units) wide and 24 x 96 (or 2,304 units) high.

Keep in mind that you can print the finished drawing at any scale, regardless of the scale factor you calculate. You can also print on paper of a different size and use the Layout tabs to create different views of your drawing and to position and scale those views differently. The scaling factor is not related to the size of the entities you draw; it simply provides a preliminary guide to help you establish the text height and drawing limits when you begin your drawing. You can change the text height and drawing limits at any time.

Setting up annotation scaling

Annotation scaling allows you to control individual entities so their size will consistently display when a drawing is displayed or printed at different scales. You don't have to use annotation scaling, but it is a convenient way to control the scaling of the following entities: text, tolerances, dimensions, leaders, multileaders, attributes, hatches, and blocks.

These individual entities can be annotative, and so can text styles, dimension styles, and multileader styles; text, dimensions, or multileaders assigned an annotative style will have annotation scaling turned on by default.

PTCAD comes ready to use annotation scaling, however, you may want to customize some of the settings according to your needs.


Customizing the scales list

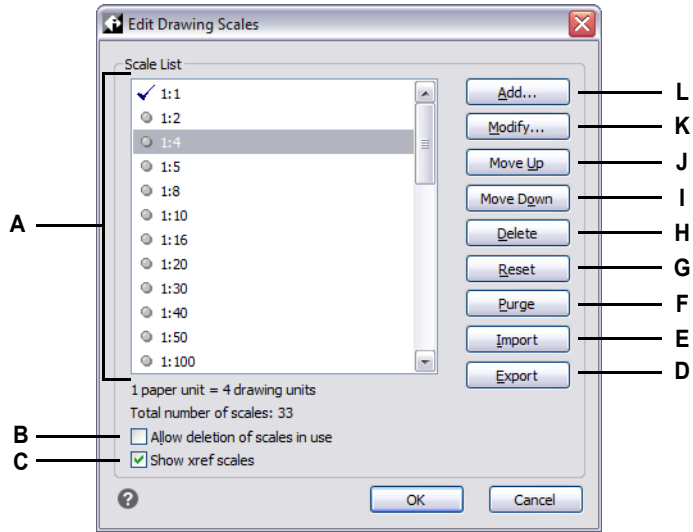
The scales list defines all of the scales that are available to assign to annotative entities. For example, to assign an annotation scale to a text entity, you choose it from the scales list. The scales list displays when you do any of the following:

- Set the current annotation scale — On the status bar, right-click Annotations Scales List.
- Assign an annotation scale to an entity — Select an entity and use the Properties or Entity Scale command.
- Print — Choose the Print command.

After you set up your scales list, you may want to create a drawing template with the default scales or export your scale list so you can easily import the list into other drawings.

To customize the scales list

- 1 Do one of the following to choose Scale List ():
 - On the ribbon, choose Annotate > Scale List (in Annotation Scaling).
 - On the menu, choose Format > Scale List.
 - On the status bar, click Annotation Scales List.
 - Type *scaletlistedit* and then press Enter.
- 2 To add a scale to the list, do the following:
 - Click Add.
 - Enter the name of the scale, which will appear in the list.
 - Enter the paper units to drawing units ratio.
 - Click OK.
- 3 Do any of the following to further customize the list:
 - Select a scale and click Modify to change a scale's name or ratio.
 - Select a scale and click Delete to remove it from the list.
 - Select a scale and click Move Up or Move Down to reposition it in the list.
- 4 Optionally click Export to save your scales list to a file that you can easily import into other drawings.
- 5 Click OK.



- A Displays the scales available for annotative entities.
- B Click to allow deleting scales that are used in the drawing.
- C Click to display scales from attached external references in the list.
- D Click to save the scales list to a file.
- E Click to open and load a scales list from a file.
- F Click to remove all unused scales from the list.
- G Click to remove all custom scales and restore the default scale list.
- H Click to remove the selected scale.
- I Click to move the selected scale down one position in the list.
- J Click to move the selected scale up one position in the list.
- K Click to modify the selected scale.
- L Click to create a new scale and add it to the list.

Purging unused annotation scales can increase performance.

Older drawings from other CAD programs can sometimes have thousands of unused annotation scales. Click Purge in the Edit Drawing Scales dialog box to remove unused annotation scales and increase performance.

Customizing styles to be annotative

Text styles and dimension styles determine whether text and dimension entities that are assigned those styles are annotative by default. Multileader styles can also be annotative, but PTCAD only supports the display of multileaders, not the creation of multileaders or multileader styles.

For text styles, see “Working with text styles” on page 474.

For dimension styles, see “Controlling dimension fit” on page 532.

Setting up automatic annotation scaling

Automatic annotation scaling automatically assigns the current annotation scale to annotation entities that have annotation scaling turned on.

To set automatic annotation scaling and the current annotation scale


- 1 On the status bar, click Automatic Annotation On/Off.
- 2 On the status bar, click Annotations Scales List.
- 3 Choose the current annotation scale.

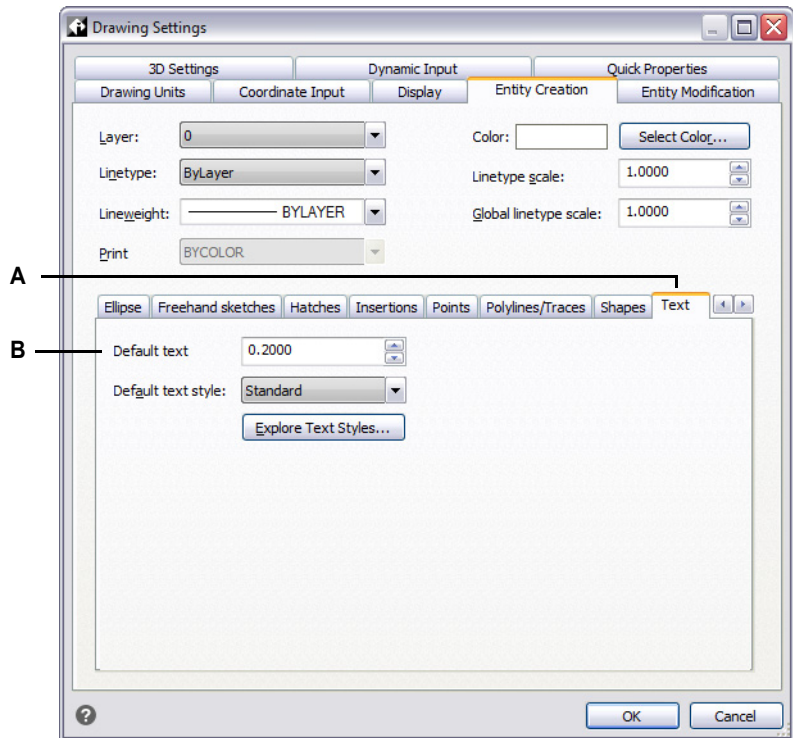
Setting the text height

The text height setting controls the height of text, measured in drawing units. Set this value initially so that text used for your most common annotations, when scaled to the size at which you will print a drawing, measures 1/8-inch high on the printed drawing.

For example, if you plan to print your drawing at 1/8" = 1'-0" and you want your text to be 1/8-inch high in the final drawing, create that text 1 foot high (in your real-world drawing units) so that when you print it, it appears 1/8-inch high on the paper. You must create text 4 feet high that you want to print 1/2-inch high.

To set the text height

- 1 Do one of the following to choose Drawing Settings ():
 - On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
 - On the menu, choose Tools > Drawing Settings.
 - On the Tools toolbar, click the Drawing Settings tool.
 - Type *dsettings* and then press Enter.
- 2 Click the Entity Creation tab.
- 3 Click the Text tab. If necessary, click the right arrow to scroll the tabs horizontally to view the Text tab.
- 4 In the Default Text Height field, select the text height or type the text height value that you want. If you have chosen an annotative text style, you enter the paper text height.
- 5 Click OK.



A Choose Text.

B Specify the text height in drawing units.


The default text height applies only if the current text style height is 0.0.
Otherwise, the text height for the current style takes precedence.

Setting the drawing limits

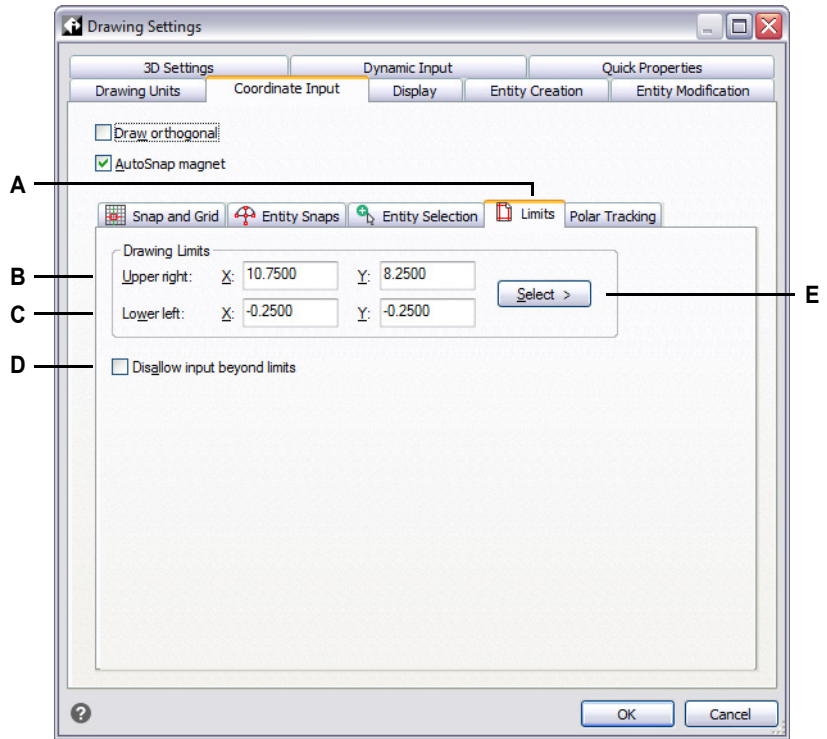
You can specify the drawing limits that form an invisible boundary around your drawing. You can use the drawing limits to make sure that you do not create a drawing larger than can fit on a specific sheet of paper when printed at a specific scale.

For example, if you plan to print your drawing at $1/8'' = 1'-0''$ (in other words, using a scale factor of 96) on a sheet of paper measuring 36 inches x 24 inches, you can set drawing limits to 3,264 units wide (that is, 34×96) and 2,112 units high (22×96), which allows a 1-inch margin around the edges of the printed image.

To set the drawing limits

- 1 Do one of the following to choose Drawing Settings ():
 - On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
 - On the menu, choose Tools > Drawing Settings.
 - On the Tools toolbar, click the Drawing Settings tool.
 - Type *dsettings* and then press Enter.
- 2 Click either the Coordinate Input tab or the Display tab.
- 3 Click the Limits tab.
- 4 Specify the x-coordinate and y-coordinate of the upper right drawing limit and the lower left drawing limit.

You can also click Select to specify the drawing limits by selecting points in the drawing.
- 5 To constrain your drawing to these drawing limits, click the Disallow Input Beyond Limits check box.
- 6 Click OK.



- A** Choose Limits.
- B** Specify the x-coordinate and y-coordinate of the upper right drawing limit.
- C** Specify the x-coordinate and y-coordinate of the lower left drawing limit.
- D** When selected, the program prevents you from drawing outside the drawing limits.
- E** Specify the drawing limits by selecting points in the drawing.

Working with colors

An entity's color determines how it is displayed and, if you are using a color printer, how it prints. Entities are created in the current color specified for the drawing.

Layers can also control the color of entities. When you open a new drawing, entities are created in the color `BYLAYER`, which adopts the color of the current layer. Initially, layer 0 is both the only layer and the current layer. Its default color is white, so your entities appear as white.

For entities and layers in PTCAD, there are three different types of colors:

- Index colors
- True colors
- Color book colors

True colors and color books are unavailable in some cases.

For example, for dimension entities and cursor display.

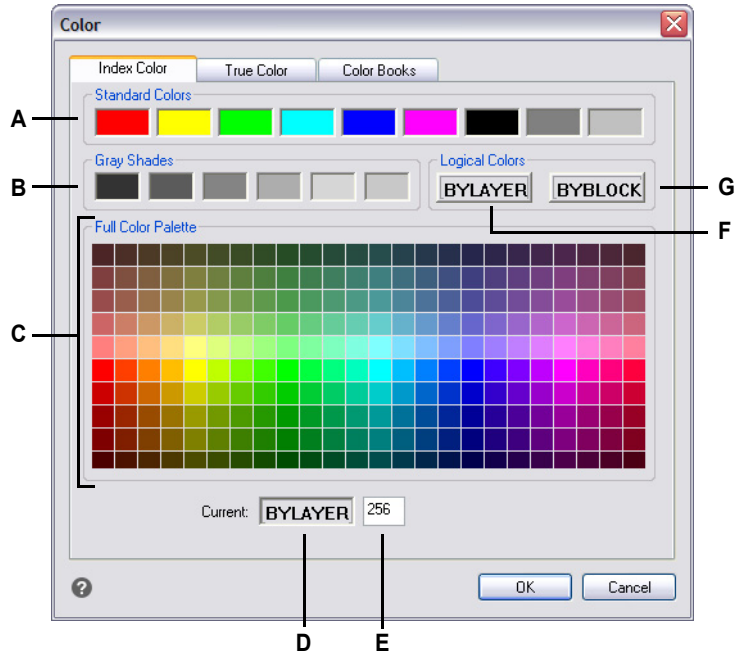
You can choose colors by selecting them from the Color dialog box. In the command bar or in some dialog boxes, you specify a color either by name or by number.

Using index colors

There are 255 standard index colors and two additional color properties that are often referred to as colors (`BYLAYER` and `BYBLOCK`). You can use seven of the 255 standard index colors by name: red, yellow, green, cyan, blue, magenta, and white. (Numbers eight and nine are not named.) Each index color has a unique number from 1 to 255. The two additional color properties are `BYLAYER` and `BYBLOCK`. These color properties cause an entity to adopt the color either of the layer or of the block in which it is a member. `BYLAYER` is color number 256, and `BYBLOCK` is color number 0. In all commands for which you would use a color, you can indicate `BYLAYER` and `BYBLOCK` as well as by numbers 256 and 0, respectively.

To select an index color

- 1 Click Select Color in the desired dialog box, such as Layers, Drawing Settings, Properties, or Multiline Text.
The Color dialog box opens.
- 2 Click the Index Color tab.
- 3 Do one of the following:
 - Click `BYBLOCK`.
 - Click `BYLAYER`.
 - Click the color of your choice, or type the color number in the Current box.
- 4 Click OK.



- | | |
|--|--|
| <p>A Click to select one of the standard colors.</p> <p>B Click to select one of the gray shades.</p> <p>C Click to select any one of the available index colors.</p> | <p>D Indicates the current color.</p> <p>E Displays the color number, or enter the color number.</p> <p>F Click to set the color to BYLAYER.</p> <p>G Click to set the color to BYBLOCK.</p> |
|--|--|

Using true colors

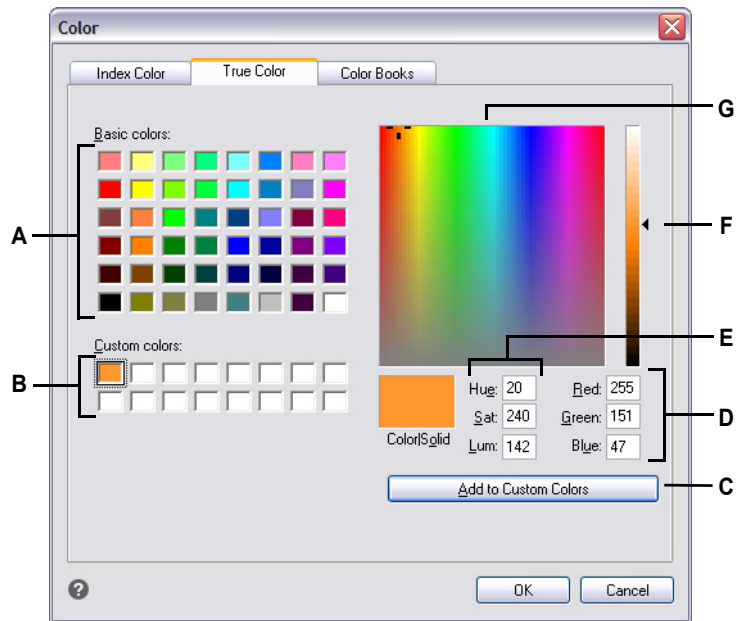
There are more than 16 million true colors from which you can choose. True colors are defined using 24-bit color.

Even with so many colors available, you can quickly choose a color from the display of basic colors or by clicking the color palette. Alternatively, if you know the values used to define the desired color, you can enter the Hue, Saturation, and Luminance (HSL) values, or you can enter the Red, Green, Blue (RGB) values.

To select a true color

- 1 Click Select Color in the desired dialog box, such as Layers, Drawing Settings, Properties, or Multiline Text.
The Color dialog box opens.
- 2 Click the True Color tab.

- 3 Do one of the following:
- Click a basic true color.
 - Click a true color in the color palette.
 - Enter HSL values for the desired true color.
 - Enter RGB values for the desired true color.
- 4 Click OK.



- A Click to select one of the basic true colors.
- B Click to select one of the custom true colors, if any are defined.
- C Click to add the selected color to the custom color area.
- D Enter (or view) the amount of red, green, and blue of the desired true color.
- E Enter (or view) the amount of hue, saturation, and luminance of the desired true color.
- F Slide or click to adjust the color luminance.
- G Click to select a true color in the color palette.




Using color books

PTCAD uses color books to store collections of colors. For example, you can store a unique color scheme for a client in a color book and then use colors only from that color book for the client's drawings.

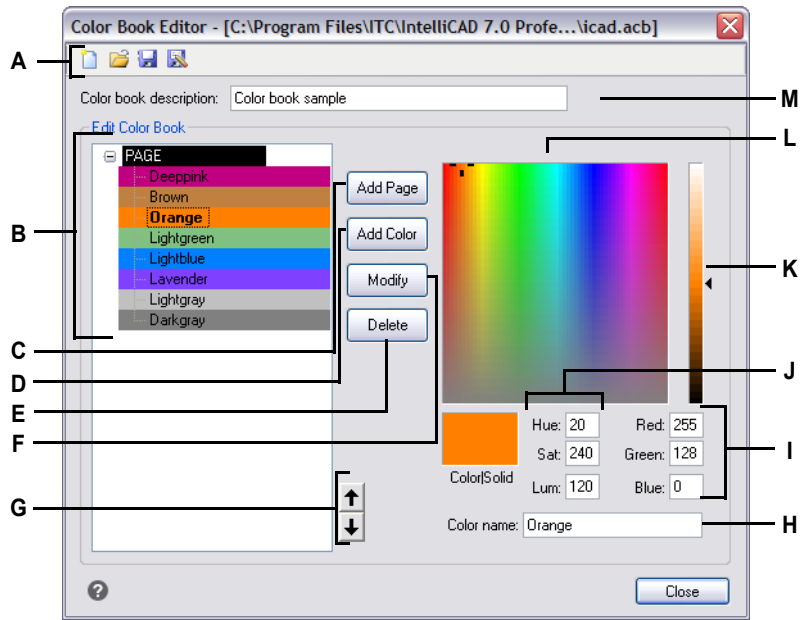
Selecting a color book color

To select a color book color

- 1 Click Select Color in the desired dialog box, such as Layers, Drawing Settings, Properties, or Multiline Text.
The Color dialog box opens.
- 2 Click the Color Books tab.
- 3 Select a color book from the list.
- 4 If you want to narrow your color search, do one of the following:
 - In the list of colors, select a color book page, if one is available. Pages are particularly helpful in large color books — they help group colors so you can find them quickly.
 - Select Show Only Color Book Colors Used in Drawing. Only those color book colors that are used in the current drawing will display in the list.
- 5 Select the desired color.
- 6 Click OK.

- 6 Do the following to define organizational pages in the color book:
 - In the contents of the color book, click an existing page or color where you want to add a page.
 - Define a color in the palette.
 - Enter the name of the page, and then click Add Page.
- 7 Do the following to define colors in the color book:
 - In the contents of the color book, click an existing page or color where you want to add a color.
 - Define a color in the palette.
 - Enter the name of the color, and then click Add Color.
- 8 Do any of the following to change existing pages and colors in the color book:
 - Modify pages and colors — Select a page or color in the color book, define its new color settings in the palette, enter any changes to its name, and then click Modify.
 - Delete pages and colors — Select a page or color in the color book, and then click Delete.
 - Rearrange pages and colors — Select a page or color in the color book, and then click the up arrow () or down arrow ().
- 9 In the Color Book Editor dialog box, click the Save tool ().
- 10 Enter a filename for the color book, and then click Save.

By default, the file is saved in the default folder where PTCAD searches for color books.







- A** Click New to create a color book; click Open to open a color book; click Save to save the color book; click Save As to save the color book with a new filename.
- B** Select a page or color to modify, delete, or reorder it, or to view its settings.
- C** Click to create a new page in the color book using the current color settings.
- D** Click to create a new color in the color book using the current color settings.
- E** Click to delete the selected page or color from the color book.
- F** Click to redefine the selected page or color to use the current color settings.
- G** Click to move the selected page or color up or down one position in the color book.
- H** Enter the name of the page or color as you want it to appear in the color book.
- I** Enter (or view) the amount of red, green, and blue of the desired color.
- J** Enter (or view) the amount of hue, saturation, and luminance of the desired color.
- K** Slide or click to adjust the color luminance.
- L** Click to select a color.
- M** Enter the name of the color book.

Modifying color books

You can modify your own color books and the color books that came with PTCAD. If you modify a color book that came with PTCAD, it is recommended that you save it with a new filename first so the original color book is not overwritten.

To modify a color book

- 1 Click Select Color in the desired dialog box, such as Layers, Drawing Settings, Properties, or Multiline Text.
The Color dialog box opens.
- 2 Click the Color Books tab.
- 3 Select the color book you want to modify.
- 4 Click Color Book Editor.
- 5 In Color Book Name, enter any changes to the color book name. This name appears in the list of color books on the Color Books tab in the Color dialog box.
- 6 Do any of the following to modify pages or colors in the color book:
 - Add pages — In the contents of the color book, click an existing page or color where you want to add a page. Define a color in the palette, enter the name of the page, and then click Add.
 - Add colors — In the contents of the color book, click an existing page or color where you want to add a color. Define a color in the palette, enter the name of the color, and then click Add.
 - Modify pages and colors — Select a page or color in the color book, define its new color settings in the palette, enter any changes to its name, and then click Modify.
 - Delete pages and colors — Select a page or color in the color book, and then click Delete.
 - Rearrange pages and colors — Select a page or color in the color book, and then click the up arrow () or down arrow ().
- 7 Do one of the following to save the color book:
 - To save the color book with the same filename, click the Save tool () in the Color Book Editor.
 - To save the color book with a new filename or in a new location, click the Save As tool () in the Color Book Editor.

Loading color books

If you have a color book given to you by a client or developed by a third-party, simply save it on your computer in a folder where PTCAD can find it. By default, color books are stored in following folder: \Documents and Settings\YourName\My Documents\Color Books.

To load a color book

- 1 Save the color book in the folder where PTCAD searches for color books.
To verify the folder location, choose Tools > Options, click the Paths/Files tab, and find the Color Book folder in the Paths list.
- 2 In a Color dialog box, click the Color Books tab.
The previously loaded color book displays in the list of color books.

Using the grid, snap alignment, cursor restriction, and dynamic input

Grid and snap settings are effective tools to use in your drawing to ensure accuracy. Although many users find it convenient to match grid points to snap settings, they are independent of each other and should not be confused. Grid points are for visual reference only; they do not affect your drawing and they do not print. Snap points are, by themselves, not visible; however, when set, they constrain the creation of new entities.


In addition, the cursor can be restricted to move orthogonally only or guides can display on the screen automatically at specified polar angle increments.

Setting a reference grid


A reference grid displays as a pattern of regularly spaced dots or lines. You can turn the grid on and off, and you can specify how far apart the dots or lines are spaced.

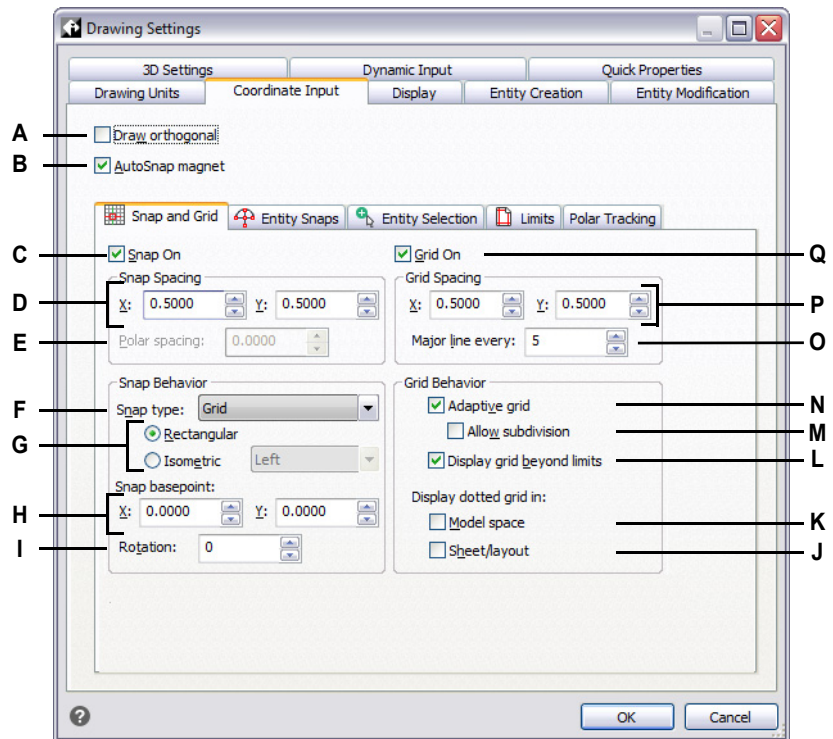
By default, the reference grid displays as lines, which has much better display performance than a grid that displays as points (dots), and it is adaptive (it proportionately scales according to the zoom ratio), helping you to align entities and visualize distances between entities. If desired, the grid can be constrained to only display within the limits of the drawing.

To turn the grid on or off and set the grid spacing

- 1 Do one of the following to choose Drawing Settings (- On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
- On the menu, choose Tools > Drawing Settings.
- On the Tools toolbar, click the Drawing Settings tool.
- Type *dsettings* and then press Enter.
- 2 Click the Coordinate Input tab.
- 3 Click the Snap and Grid tab.
- 4 Click the Grid On check box to turn the grid on or off.
- 5 Under Grid Spacing, in the X field, choose the horizontal grid spacing.
- 6 Under Grid Spacing, in the Y field, choose the vertical grid spacing.
- 7 Click OK.

Use the shortcuts for toggling the grid display on and off.

Double-click the GRID setting on the status bar, click the Reference Grid tool () on the Settings toolbar, or press F7.




- A Click to enable orthogonal mode.
- B Click to enable crosshairs locking to the nearest snap point.
- C Click to enable snap mode.
- D Specify the x and y snap spacing.
- E Specify the polar snap spacing (available if Snap Type is set to Polar).
- F Select whether to use standard grid snapping (Grid) or to use polar snapping (Polar).
- G Click to use a rectangular or isometric snap. If isometric, select the initial isoplane: top, left (default), or right.
- H Specify the x- and y-coordinates of the snap origin.
- I Specify the snap rotation angle. The grid will display rotated to match the snap setting.
- J Click to display a dotted grid for paper sheets and layouts (Layout tabs); otherwise a lined grid displays.
- K Click to display a dotted grid for model space (Model tab); otherwise a lined grid displays.
- L Click to display the grid beyond the limits area.
- M Click to split the grid if the zoom level is smaller than the grid unit (available if adaptive grid is turned on).
- N Click to enable an adaptive grid, which proportionally scales according to the zoom ratio.
- O Enter the number of minor grid lines to display for every major grid line.
- P Specify the x and y grid spacing.
- Q Click to display the reference grid.

Setting snap spacing

Another way to ensure drawing accuracy is to turn on and set snap spacing. When snap is turned on, the program restricts the selection points to predetermined snap intervals. Although it is often helpful to match the snap spacing to some interval of the grid spacing or another related setting, the settings do not have to match.

To turn snap settings on and set snap spacing

- 1 Do one of the following to choose Drawing Settings ():
 - On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
 - On the menu, choose Tools > Drawing Settings.
 - On the Tools toolbar, click the Drawing Settings tool.
 - Type *dsettings* and then press Enter.
- 2 Click the Coordinate Input tab.
- 3 Click the Snap and Grid tab.
- 4 Click the Snap On check box to turn Snap on.
- 5 Under Snap Spacing, in the X field, choose the horizontal snap spacing.
- 6 Under Snap Spacing, in the Y field, choose the vertical snap spacing.
- 7 Click OK.

Use the shortcuts for toggling the snap settings on and off.


?lick the Entity Snap Settings button on the status bar or press F9.

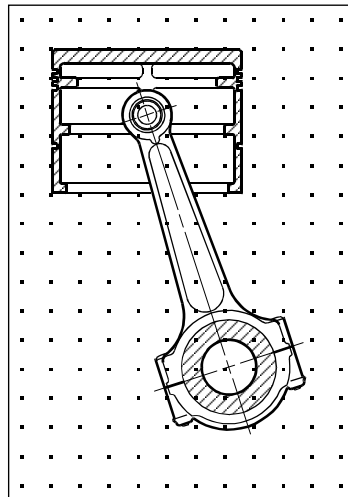
In addition to setting the snap spacing, you can change the snap and grid orientation. You can also rotate the alignment of the grid or set it to create isometric drawings.

Changing the snap and grid angle and base point

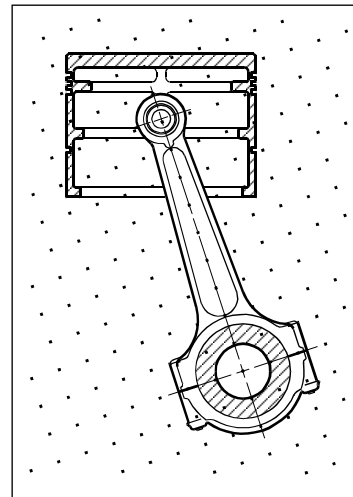
The snap and grid are both normally based on the drawing origin, the 0,0 coordinate in the World Coordinate System (WCS). You can relocate the snap and grid origin, however, to help you draw entities in relation to a different location. You can also rotate the grid to a different angle to realign the crosshairs to the new grid angle. If the grid is on, and the grid spacing is 0,0, then the grid defaults to the snap spacing.

To change the snap angle and base point

- 1 Do one of the following to choose Drawing Settings ():
 - On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
 - On the menu, choose Tools > Drawing Settings.
 - On the Tools toolbar, click the Drawing Settings tool.
 - Type *dsettings* and then press Enter.
- 2 Click the Coordinate Input tab.
- 3 Click the Snap and Grid tab.
- 4 Click the Snap On check box to turn Snap on.
- 5 Click the Grid On check box to turn Grid on.
- 6 In Snap Basepoint, type the x-coordinate and y-coordinate of the new snap origin.
- 7 In Rotation, type the snap rotation angle, which also changes the display of the grid.
- 8 Click OK.



Default grid and snap alignment.



Rotated grid and snap alignment.

Using isometric snap and grid

You can use the Isometric Snap and Grid option to create two-dimensional isometric drawings. With the isometric option, you are simply drawing a simulated three-dimensional view on a two-dimensional plane, much the same as you might draw on a piece of paper. Do not confuse isometric drawings with three-dimensional drawings. You create three-dimensional drawings in three-dimensional space.


The isometric option always uses three preset planes, which are denoted as left, right, and top. You cannot alter the arrangement of these planes. If the snap angle is 0, the three isometric axes are 30 degrees, 90 degrees, and 150 degrees.

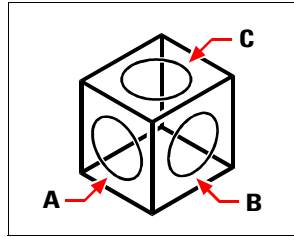
When you use the Isometric Snap option and select an isometric plane, the snap intervals, grid, and crosshairs align with the current plane. The grid is always shown as isometric and uses *y*-coordinates to calculate the grid spacing. If you click the Draw Orthogonal check box, the program restricts the drawing of entities to the current isometric plane.

Use the shortcut to toggle between isometric planes.

Press F5.

To turn the Isometric Snap and Grid option on

- 1 Do one of the following to choose Drawing Settings ():
 - On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
 - On the menu, choose Tools > Drawing Settings.
 - On the Tools toolbar, click the Drawing Settings tool.
 - Type *dsettings* and then press Enter.
- 2 Click the Coordinate Input tab.
- 3 Click the Snap and Grid tab.
- 4 Click the Snap On check box to turn Snap on.
- 5 Click the Grid On check box to turn Grid on.
- 6 Under Snap Type, choose Isometric, then choose the option for the initial isometric plane you want (Top, Left, or Right).
- 7 Click OK.



Isometric planes left (A), right (B), top (C).


Using orthogonal locking

You can restrict cursor movement to the current horizontal and vertical axes so that you can draw at right angles, or orthogonally. For example, with the default 0-degree orientation (angle 0 at the “three o’clock” or “east” position), when the Draw Orthogonal option is enabled, lines are restricted to 0 degrees, 90 degrees, 180 degrees, or 270 degrees. As you draw lines, the rubber-banding line follows either the horizontal or vertical axis, depending on which axis is farthest from the cursor. When you enable the isometric snap and grid, cursor movement is restricted to orthogonal equivalents within the current isometric plane.



Sometimes orthogonal locking is not used even when it is turned on.

PTCAD ignores orthogonal locking when you type coordinates in the command bar or when you use entity snaps. Additionally, orthogonal locking and polar tracking cannot be used at the same time — turning one option on turns the other option off.

To enable orthogonal locking

- 1 Do one of the following to choose Drawing Settings ():
 - On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
 - On the menu, choose Tools > Drawing Settings.
 - On the Tools toolbar, click the Drawing Settings tool.
 - Type `dsettings` and then press Enter.
- 2 Click the Coordinate Input tab.
- 3 Click the Draw Orthogonal check box.
- 4 Click OK.

Toggle orthogonal locking on and off.


On the status bar, click Orthogonal On/Off () or press F8. You can also click the Draw Orthogonal tool () on the Settings toolbar.

Using polar tracking

When polar tracking is turned on, guides display on the screen automatically at the polar angle increment that you specify. For example, if you draw a line with polar tracking turned on at 45 degrees, the rubber-banding line displays at 45 degree angle increments.

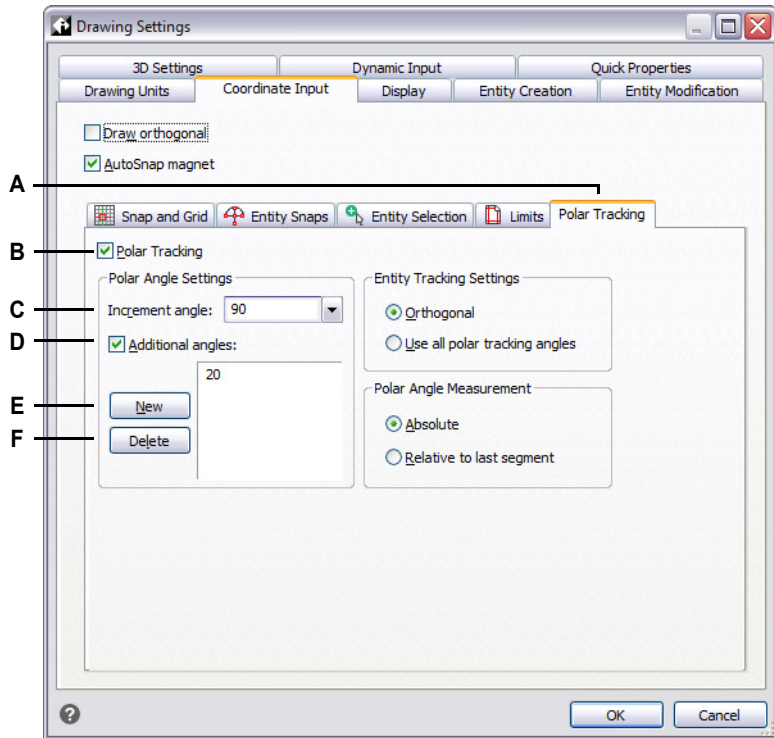
Polar tracking and orthogonal locking cannot be used at the same time — turning one option on turns the other option off.

To enable polar tracking and specify the polar angle increment

- 1 Do one of the following to choose Drawing Settings ():
 - On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
 - On the menu, choose Tools > Drawing Settings.
 - On the Tools toolbar, click the Drawing Settings tool.
 - Type *dsettings* and then press Enter.
- 2 Click the Coordinate Input tab.
- 3 Click the Polar Tracking tab.
- 4 Click the Polar Tracking check box.
- 5 Do one of the following to specify the polar angle increments:
 - In Increment, select an angle.
 - Mark the Additional Angles check box and click New to define a custom angle increment.
- 6 Click OK.

Toggle polar tracking on and off.

Click Polar Tracking On/Off on the status bar, type POLARTRACK in the command line, press F10, or press Ctrl+U.



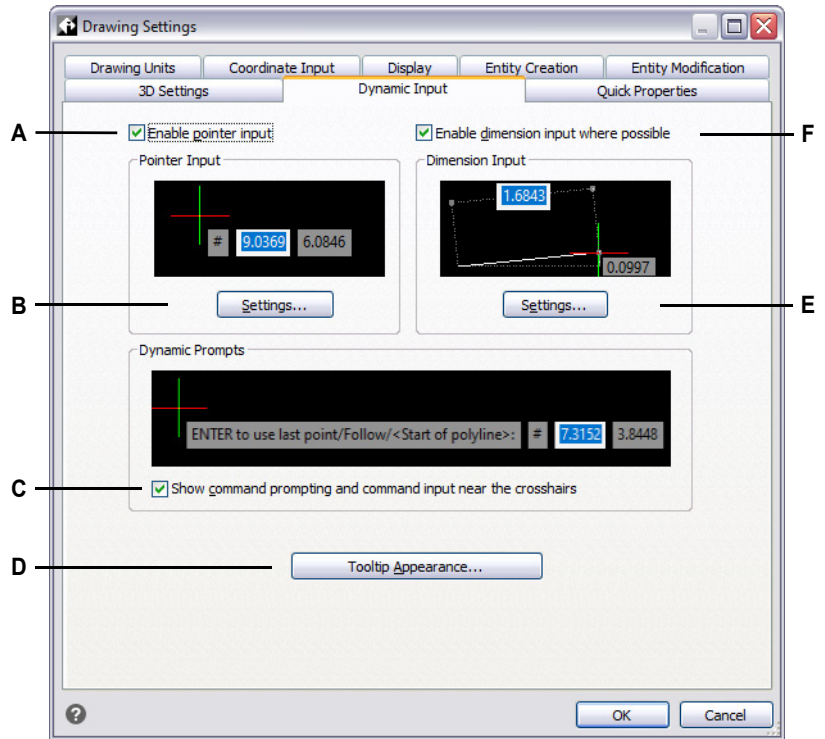
- A** Select Polar Tracking.
- B** Select to display polar tracking guides on the screen.
- C** Select the angle at which you want the guides to display.
- D** Select to specify angle increments that are not defined in the Increment Angle list.
- E** Click to add a new angle increment.
- F** Click to delete the currently selected angle from the Additional Angles list.

Using dynamic input

When dynamic input is turned on, guides display on the screen automatically as you draw and edit your drawing. Dynamic input can display when you are prompted for a point, prompted for dimensions such as distances and angles, and prompted for command input that you would typically enter at the command line. You can determine many settings for how dynamic input prompts for input and even how the tooltips display on your screen.

To turn on dynamic input

- 1 Do one of the following to choose Drawing Settings (



- A Mark box to turn on pointer input.
- B Click to specify various options for pointer input.
- C Mark to display command prompts and allow command input in dynamic input tooltips.
- D Click to choose tooltip appearance settings.
- E Click to specify various options for dimension input.
- F Mark to turn on dimension input.

Toggle dynamic input on and off.

On the status bar, click Dynamic Input On/Off to turn pointer input, dimension input, and command prompting on or off all at once.

To customize pointer input settings

- 1 In the Drawing Settings dialog box, click the Dynamic Input tab.
- 2 In Pointer Input, click Settings.
- 3 Choose whether you want pointer input to default to Polar or Cartesian format for default second and next points.
- 4 Choose whether you want pointer input to default to Relative or Absolute coordinates for second and next points.
- 5 Choose when you want pointer input tooltips to display:
 - **After Starting to Type Coordinate Data** Pointer input tooltips appear only when you start typing coordinate data.
 - **When a Command Prompts for a Point** Pointer input tooltips automatically appear when a command prompts for a point.
 - **Always - Even When Not in a Command** Pointer input tooltips always display.
- 6 If you want to always display a symbol indicating the point format, mark Show '@' or '#' Before Coordinates.
- 7 Click OK.

To customize dimension input settings

- 1 In the Drawing Settings dialog box, click the Dynamic Input tab.
- 2 In Dimension Input, click Settings.
- 3 Choose one of the following to determine when dimension input displays during grip editing:
 - **One Dimension Input Field Only** Displays one dimension input field which can vary depending on the entity selected.
 - **Two Dimension Input Fields** Displays two dimension input fields which can vary depending on the entity selected.
 - **Any Combination of Dimension Input Fields** Displays all enabled dimension input fields that you select: resulting dimension mark, length change mark, absolute angle mark, angle change mark, and arc radius mark.
- 4 If you want to use the ANGBASE system variable when specifying the north direction relative to the current UCS, mark Use ANGBASE Angle. If unmarked, ANGBASE is ignored which can lead to unpredictable results in some cases.
- 5 Click OK.

To customize the appearance of dynamic input tooltips

- 1 In the Drawing Settings dialog box, click the Dynamic Input tab.
- 2 Click Tooltip Appearance.
- 3 In Model Space Colors, select a color for the text and background of dynamic input tooltips that display in model space.
- 4 In Paper Space Colors, select a color for the text and background of dynamic input tooltips that display in paper space.
- 5 In Size, move the slider to specify the size of dynamic input tooltips and text.
- 6 In Transparency, move the slider to specify the transparency of dynamic input tooltips and text.
- 7 Click OK.

Using entity snaps

Entity snaps enable you to quickly select exact geometric points on existing entities without having to know the exact coordinates of those points. With entity snaps, you can select the endpoint of a line or arc, the center point of a circle, the intersection of any two entities, or any other geometrically significant position. You can also use entity snaps to draw entities that are tangent or perpendicular to an existing entity.

You can use entity snaps any time the program prompts you to specify a point—for example, if you are drawing a line or other entity. You can work with entity snaps in one of two ways:

- Enable a running entity snap that remains in effect until you turn it off by choosing an entity snap when no other command is active.
- Enable a one-time entity snap for a single selection by choosing an entity snap when another command is active. You can also use a one-time entity snap to override a running entity snap.

If you type the name of entity snaps, you don't need to type the whole name.

Type only the first three letters of the snap name.

When using entity snaps, the program recognizes only visible entities or visible portions of entities. You cannot snap to entities on layers that have been turned off or frozen.

When you specify one or more entity snaps, an entity snap target box is added to the crosshairs. In addition, an icon appears adjacent to the crosshairs indicating the active entity snap. When you select an entity, the program snaps to the snap point closest to the center of the target box.

Setting entity snaps

You can set entity snaps using any of the following methods:


- On the ribbon, choose Draw and in Entity Snaps, choose the entity snap you want to set.
- Choose Tools > Entity Snaps, and choose the entity snap you want to set.
- On the Entity Snaps toolbar, click one of the entity snap tools.
- In the command bar, type an entity snap command.
- In the status bar, click Entity Snap Settings and choose the entity snap you want to set.
- Press and hold down the Shift key while right-clicking anywhere within the drawing window to display the entity snap shortcut menu, and then choose the entity snap you want to set.

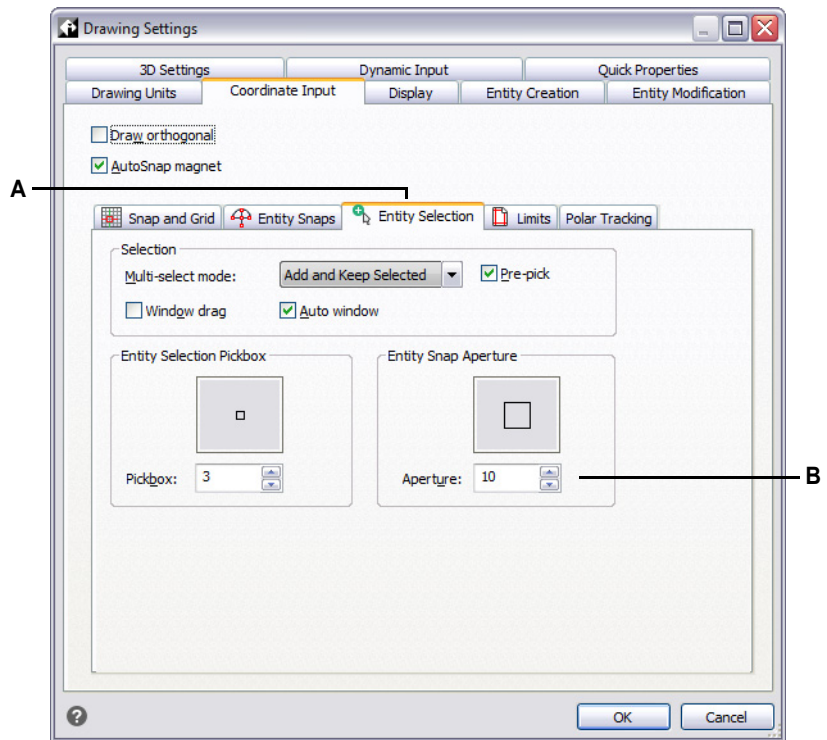
You can also set entity snaps using the Drawing Settings dialog box. To do this, choose Tools > Entity Snaps > Entity Snap Settings. The Drawing Settings dialog box is displayed with the Coordinate Input tab active. In the Entity Snap Modes list, click the check box for each of the entity snaps that you want to set.

There are several indicators if an entity snap is active.

If you select an entity snap, a check mark appears next to the entity snap in the menu, the associated tool if the Entity Snaps toolbar is active, and the corresponding box is checked in the Coordinate Input tab of the Drawing Settings dialog box.

To change the size of the entity snap target box

- 1 Do one of the following to choose Drawing Settings (- On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
- On the menu, choose Tools > Drawing Settings.
- On the Tools toolbar, click the Drawing Settings tool.
- Type *dsettings* and then press Enter.
- 2 Click the Coordinate Input tab.
- 3 Click the Entity Selection tab.
- 4 Under Entity Snap Aperture, change the value in the Aperture field.
- 5 Click OK.




A Choose Entity Selection.

B Type or select the entity snap aperture size.

Nearest snap

Use the Nearest snap to snap to the nearest point of another entity. You can snap to the nearest point on an arc, circle, ellipse, elliptical arc, line, point, polyline segment, ray, spline, infinite line, or hatch pattern that is visually closest to the cursor.

To set the Nearest snap


Do one of the following to choose Nearest ():

- On the ribbon, choose Draw > Nearest (in Entity Snaps).
- On the menu, choose Tools > Entity Snaps > Nearest.
- On the Entity Snaps toolbar, click the Nearest tool.
- Type *nearest* and then press Enter.

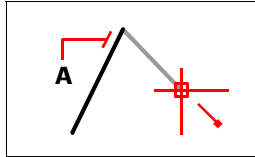
Endpoint snap

Use the Endpoint snap to snap to the endpoint of another entity. You can snap to the closest endpoint of an arc, line, polyline segment, ray, hatch pattern, plane, or three-dimensional face. If an entity has thickness, the Endpoint snap also snaps to the endpoints of the edges of the entity.

To set the Endpoint snap

Do one of the following to choose Endpoint ():

- On the ribbon, choose Draw > Endpoint (in Entity Snaps).
- On the menu, choose Tools > Entity Snaps > Endpoint.
- On the Entity Snaps toolbar, click the Endpoint tool.
- Type *endpoint* and then press Enter.



To snap to the endpoint, select anywhere on the entity near its endpoint (A).


Midpoint snap

Use the Midpoint snap to snap to the midpoint of another entity. You can snap to the midpoint of an arc, ellipse, line, polyline segment, plane, infinite line, spline, or hatch pattern. In the case of infinite lines, the midpoint snaps to the first defined point. If an entity has thickness, the midpoint entity snap also snaps to the midpoint of the edges of the entity.

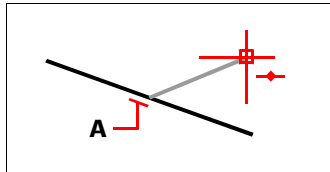
Type *m2p* or *mtp* to enable a one-time snap to the midpoint between two points.

You specify the points, such as two points selected using entity snaps.

To set the Midpoint snap

Do one of the following to choose Midpoint ():

- On the ribbon, choose Draw > Midpoint (in Entity Snaps).
- On the menu, choose Tools > Entity Snaps > Midpoint.
- On the Entity Snaps toolbar, click the Midpoint tool.
- Type *midpoint* and then press Enter.



To snap to the midpoint, select anywhere on the entity near its midpoint (A).

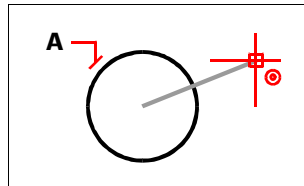
Center snap

Use the Center snap to snap to the center point of another entity. You can snap to the center of an arc, circle, polygon, ellipse, or elliptical arc. To snap to the center, you must select a visible portion of the entity.

To set the Center snap

Do one of the following to choose Center (☉):

- On the ribbon, choose Draw > Center (in Entity Snaps).
- On the menu, choose Tools > Entity Snaps > Center.
- On the Entity Snaps toolbar, click the Center tool.
- Type *center* and then press Enter.



To snap to the center, select anywhere on the visible portion of the entity (A).

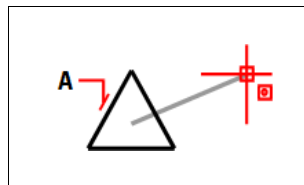
Geometric Center snap

Use the Geometric Center snap to snap to the centroid of another entity. You can snap to the geometric center of any closed polyline or spline. To snap to the geometric center, you must select a visible portion of the entity.

To set the Geometric Center snap

Do one of the following to choose Geometric Center (☐):

- On the ribbon, choose Draw > Geometric Center (in Entity Snaps).
- On the menu, choose Tools > Entity Snaps > Geometric Center.
- On the Entity Snaps toolbar, click the Geometric Center tool.
- Type *gcenter* and then press Enter.




To snap to the geometric center, select anywhere on the visible portion of the entity (A).

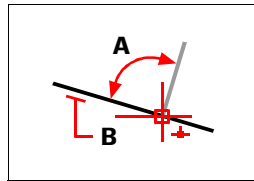
Perpendicular snap

Use the Perpendicular snap to snap to a point perpendicular to the selected entity. You can snap to an arc, circle, ellipse, line, polyline, infinite line, ray, spline, hatch pattern, or edge of a plane to form a perpendicular alignment with that entity or with an imaginary extension of that entity.

To set the Perpendicular snap

Do one of the following to choose Perpendicular ():

- On the ribbon, choose Draw > Perpendicular (in Entity Snaps).
- On the menu, choose Tools > Entity Snaps > Perpendicular.
- On the Entity Snaps toolbar, click the Perpendicular tool.
- Type *perpendicular* and then press Enter.




To form a perpendicular angle (A) to an entity (B), select anywhere on the entity.

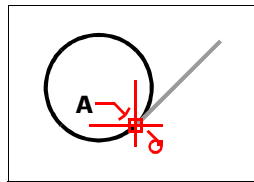
Tangent snap

Use the Tangent snap to snap to the tangent point of another entity. You can snap to the point on an arc, ellipse, spline, or circle that, when connected to the previous point, forms a line tangent to that entity.

To set the Tangent snap

Do one of the following to choose Tangent ():

- On the ribbon, choose Draw > Tangent (in Entity Snaps).
- On the menu, choose Tools > Entity Snaps > Tangent.
- On the Entity Snaps toolbar, click the Tangent tool.
- Type *tangent* and then press Enter.




To snap to a tangent, select the entity near the tangent point (A).

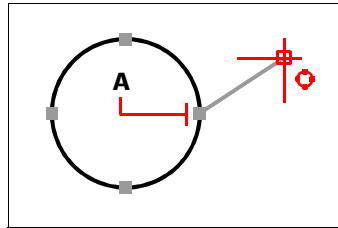
Quadrant snap

Use the Quadrant snap to snap to the quadrant point of another entity. You can snap to the closest quadrant of an arc, circle, ellipse, or elliptical arc.

To set the Quadrant snap

Do one of the following to choose Quadrant ():

- On the ribbon, choose Draw > Quadrant (in Entity Snaps).
- On the menu, choose Tools > Entity Snaps > Quadrant.
- On the Entity Snaps toolbar, click the Quadrant tool.
- Type *quadrant* and then press Enter.




To snap to a quadrant, select the entity near the quadrant point (A).

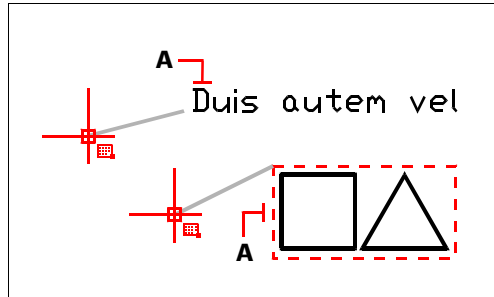
Insertion Point snap

Use the Insertion Point snap to snap to the insertion point of an attribute, block, or text entity.

To set the Insertion Point snap

Do one of the following to choose Insertion Point ():

- On the ribbon, choose Draw > Insertion Point (in Entity Snaps).
- On the menu, choose Tools > Entity Snaps > Insertion Point.
- On the Entity Snaps toolbar, click the Insertion Point tool.
- Type *insertion* and then press Enter.




To snap to an insertion point, select anywhere on the entity (A).

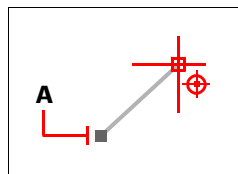
Node snap

Use the Node snap to snap to a point entity.

To set the Node snap

Do one of the following to choose Node ():

- On the ribbon, choose Draw > Node (in Entity Snaps).
- On the menu, choose Tools > Entity Snaps > Node.
- On the Entity Snaps toolbar, click the Node tool.
- Type *node* and then press Enter.




To snap to a point entity, select the entity (A).

Extension snap

Use the Extension snap to snap to a temporary extension line of an arc or line. To make the extension line visible, pause the cursor over the endpoints of the entity.

To set the Extension snap


Do one of the following to choose Extension ():

- On the ribbon, choose Draw > Extension (in Entity Snaps).
- On the menu, choose Tools > Entity Snaps > Extension.
- On the Entity Snaps toolbar, click the Extension tool.
- Type *extension* and then press Enter.


Parallel snap

Use the Parallel snap to show parallel guides when picking second and subsequent points of new linear entities. Guides display parallel to lines, infinite lines, rays, or linear segments of polylines that you indicate with the cursor.

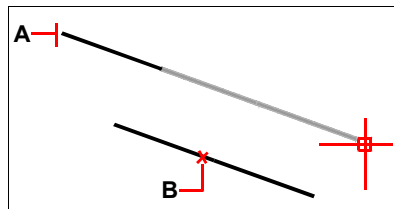
Turn Orthogonal off before using parallel snapping

On the status bar, click Orthogonal On/Off () or press F8 to turn it off or on.

To set the Parallel snap

Do one of the following to choose Parallel ():

- On the ribbon, choose Draw > Parallel (in Entity Snaps).
- On the menu, choose Tools > Entity Snaps > Parallel.
- On the Entity Snaps toolbar, click the Parallel tool.
- Type *parallel* and then press Enter.



Select the first point of an entity (A), hover over the desired parallel entity (B), then move the cursor back to the new entity to view the guide.

To draw a parallel linear entity

- 1 Choose a command, for example, *line*.
- 2 Select the entity's first point.
- 3 Type *par* and then press Enter.
- 4 Move the cursor over the desired linear entity. A small "x" marks the position on the linear entity.
- 5 Select additional entities if necessary. You can also deselect an entity by moving the cursor over its corresponding "x."
- 6 Move the cursor back to the entity you are creating and use the parallel guide to specify the next point of the entity.

Apparent Intersection snap


The Apparent Intersection snap snaps to the intersection of two entities that do not intersect in three-dimensional space but seem to intersect in the current view. You can snap to the intersection of any two entities in the combination of an arc, circle, line, infinite line, polyline, ray, ellipse, elliptical arc, spline, hatch pattern, polygon mesh, or polyface mesh. You can also snap to an intersection point within a single entity, including a polyline or spline.

The Extended Apparent Intersection snap snaps to the logical location where two entities (lines, arcs, or elliptical segments) would intersect if they were of infinite length. Dashed extension lines are drawn to help show the extended apparent intersection.

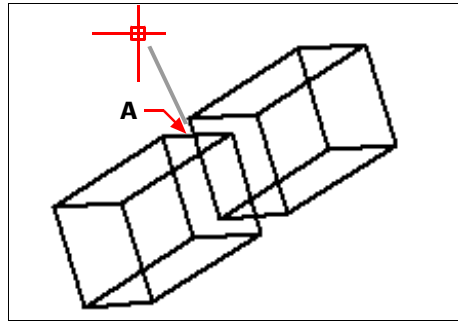
There are two types of intersection snaps.

You can set the Apparent Intersection snap or Intersection snap, but you cannot use both at the same time.

To set the Apparent Intersection snap

Do one of the following to choose Apparent Intersection (

- On the ribbon, choose Draw > Apparent Intersection (in Entity Snaps).
- On the menu, choose Tools > Entity Snaps > Apparent Intersection.
- On the Entity Snaps toolbar, click the Apparent Intersection tool.
- Type *apparent* and then press Enter.



To snap to an intersection, select the apparent intersection (A).

To snap to an extended apparent intersection point


- 1 Choose a command, for example, *line*.
- 2 Type *app* and then press Enter.
- 3 Select an extended apparent intersection point.

The Extended Apparent Intersection snap turns off automatically after you select a point.

From Point snap

Use the From Point snap to set a temporary base point from which to offset point selection. The From Point snap can be used only while another active command is requesting a point.


To set a temporary offset point

- 1 Choose a command, for example, type *line* or *move*.
- 2 Do one of the following to choose From Point ():
 - On the ribbon, choose Draw > From Point (in Entity Snaps).
 - On the menu, choose Tools > Entity Snaps > From Point.
 - On the Entity Snaps toolbar, click the From Point tool.
 - Type *from* and then press Enter.
- 3 Select where to place the temporary base point.
- 4 Enter the offset distance from the base point, for example, (@8.5,0), where you want to locate the next point. Note that entering (8.5,0) places an absolute point from the UCS origin, not a relative point from the base point.
- 5 Continue with the original command.

Track Point snap

Use the Track Point snap to set a temporary tracking point while using a command. The Track Point snap can be used only while another active command is requesting a point.


To set a Track Point snap

- 1 Choose a command, for example, type *line* or *move*.
- 2 Do one of the following to choose Track Point (- On the ribbon, choose Draw > Track Point (in Entity Snaps).
- On the menu, choose Tools > Entity Snaps > Track Point.
- On the Entity Snaps toolbar, click the Track Point tool.
- Type *tt* and then press Enter.
- 3 Select a point to mark a temporary tracking point.
- 4 Continue with the original command.

Mid Between 2 Points snap

Use the Mid Between 2 Points snap to set a temporary midpoint snap between two points. The Mid Between 2 Points snap can be used only while another active command is requesting a point.


To set a temporary midpoint snap between two points

- 1 Choose a command, for example, *line* or *move*.
- 2 Do one of the following to choose Mid Between 2 Points (- On the ribbon, choose Draw > Mid Between 2 Points (in Entity Snaps).
- On the menu, choose Tools > Entity Snaps > Mid Between 2 Points.
- On the Entity Snaps toolbar, click the Mid Between 2 Points tool.
- Type *m2p* (or *mtp*) and then press Enter.
- 3 Select the first point.
- 4 Select the second point. The midpoint is calculated automatically.
- 5 Continue with the original command.

Clearing entity snaps

Use the Clear Entity Snaps command to turn off all entity snap settings, regardless of how they were set: by menu, tool, command, or in the Drawing Settings dialog box.

To clear entity snaps

Do one of the following to choose Clear Entity Snaps (

- On the ribbon, choose Draw > Clear Entity Snaps (in Entity Snaps).
- On the menu, choose Tools > Entity Snaps > Clear Entity Snaps.

- On the Entity Snaps toolbar, click the Clear Entity Snaps tool.
- Type *none* and then press Enter.















Using fly-over snapping

Fly-over snapping is a visual aid to help you see and use entity snaps more efficiently. When fly-over snapping is turned on, PTCAD displays a colored marker at matching entity snap points as you move the crosshairs around the drawing.

Viewing fly-over snap markers

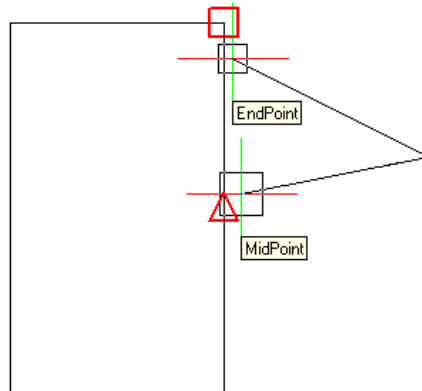
Each entity snap has its own marker.

Fly-over snap markers


Marker	Entity snap
	Endpoint
	Nearest
	Midpoint
	Center
	Geometric Center
	Perpendicular
	Tangent
	Extension
	Quadrant
	Insertion Point
	Parallel
	Node
	Intersection
	Apparent Intersection

Setting up fly-over snapping

When fly-over snapping is enabled and multiple entity snaps are on, you can press Tab to cycle through the available entity snap points of the entities covered by the target box. For example, when the Endpoint and Midpoint snaps are on and the aperture box is located on a line, press Tab to cycle between the line's closest endpoint and midpoint.



To set the fly-over snapping options

- 1 Do one of the following to choose Drawing Settings ():
 - On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
 - On the menu, choose Tools > Drawing Settings or choose Tools > Entity Snaps > Entity Snap Settings.
 - On the Tools toolbar, click the Drawing Settings tool.
 - Type *dsettings* and then press Enter.
- 2 Click the Coordinate Input tab.
- 3 Click the Entity Snaps tab.
- 4 Click Display Options. The Options dialog box opens to the Snapping tab.
- 5 Mark Enable Fly-Over Snapping to turn on fly-over snapping.
- 6 Set the fly-over options, including the color, size, and thickness of the snap marker.
- 7 Click OK.
- 8 Click OK again.

For details about the various fly-over snapping options, see “Changing the options on the Snapping tab” on page 860.

Using entity snap tracking

When entity snap tracking is turned on, guides display at specified angles outward from temporary tracking points. This can help you draw and modify entities using the relative position of existing entities, for example, inserting a block with the same y-coordinate of an existing line.


Temporary tracking points are marked with a red plus sign, and can be placed anywhere in the drawing. To select where tracking points display, first activate a command that requests a point, then:

- Move the cursor and hover over an entity snap point to add a tracking point.
- Choose the Track Point command, then select anywhere in the drawing for the tracking point to reside.
- Move the cursor and hover over an existing tracking point to remove a tracking point.

You can set up entity snap tracking to display guides at 90-degree increments or additional increments that are defined for polar tracking. You can also set up entity snap tracking to display guides relative to the current UCS or relative to the last segment drawn.

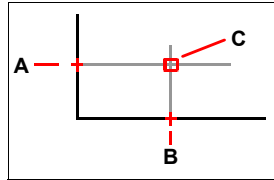
To use entity snap tracking, at least one entity snap must be turned on and running snaps cannot be turned off. Polar tracking does not have to be turned on in order to use entity snap tracking.

To turn entity snap tracking on or off

- 1 Do one of the following to choose Drawing Settings (- On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
- On the menu, choose Tools > Drawing Settings.
- On the Tools toolbar, click the Drawing Settings tool.
- Type *dsettings* and then press Enter.
- 2 Click the Coordinate Input tab.
- 3 Click the Entity Snaps tab.
- 4 Click the Entity Snap Tracking checkbox.


Toggle entity snap tracking on and off.

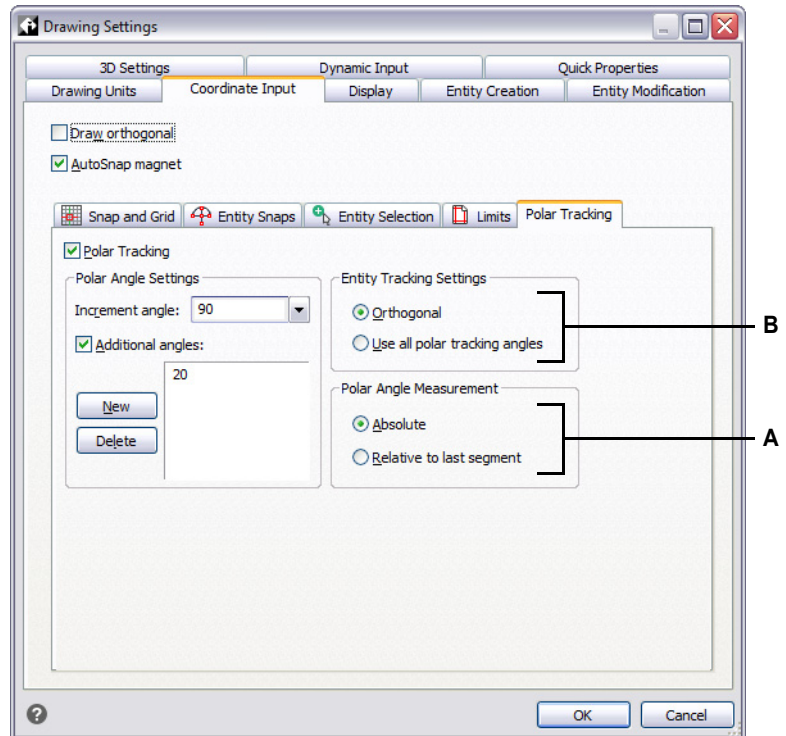
Click the *Entity Snap Tracking On/Off* on the status bar, type *ENTTRACK*, or press *F11*.



Start a command, hover over one or more entity snap points to mark them with a tracking point (A then B), then move the cursor to view the guides (C). In this example, the Midpoint snap is turned on and entity snap tracking displays at 90-degree increments.

To specify settings for entity snap tracking

- 1 Do one of the following to choose Drawing Settings ():
 - On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
 - On the menu, choose Tools > Drawing Settings.
 - On the Tools toolbar, click the Drawing Settings tool.
 - Type *dsettings* and then press Enter.
- 2 Click the Coordinate Input tab.
- 3 Click the Polar Tracking tab.
- 4 Select one of the following to specify entity tracking settings:
 - **Orthogonal** Only 90-degree angle increments are used for entity snap tracking, regardless of polar tracking settings.
 - **Use all polar tracking angles** All defined polar tracking angles, including those other than 90-degree angle increments, are used for entity snap tracking.
- 5 Select one of the following to specify how the angles of guides are calculated for entity snap tracking:
 - **Absolute** Guides display at angles relative to the current UCS.
 - **Relative to last segment** Guides display at angles relative to the last segment drawn, if creating entities with multiple segments (otherwise guides display at absolute angles).
- 6 Click OK.



- A** Select to use either 90-degree increment angles or all angles specified in Increment Angle and Additional Angles for polar tracking.
- B** Select whether angles for guides are calculated relative to the current UCS or relative to the last previously drawn segment when drawing entities with more than one segment.

Saving your drawing

When you save a drawing, your work is saved in a drawing (.dwg) file.

After you have saved your drawing for the first time, you can save it with a new name. In addition to drawing (.dwg) files, you can also save a drawing in a Drawing Exchange Format (.dxf) file or a drawing template (.dwt) file.

If you created your drawing using a template, saving the drawing does not alter the original template.

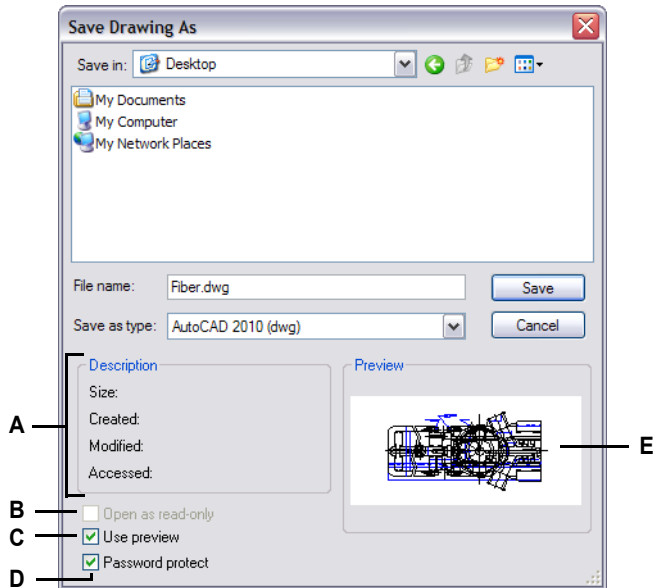
If you need an added level of security for your drawing files, you can save the drawing with a password so only you and those who know the password can open the drawing.

Saving a drawing

To save a drawing, choose any of the following methods to choose Save ():

- On the ribbon Quick Access toolbar, click Save, or on the ribbon Application button, choose Save.
- On the menu, choose File > Save.
- On the Standard toolbar, click Save.
- Type *save* and then press Enter.
- Type *qsave* and then press Enter.

When you save a drawing the first time, the program displays the Save Drawing As dialog box so that you can choose a directory and type a name for the drawing. You can use any name when you first save the drawing.



- | | |
|--|--|
| <p>A Displays a description of the file size, creation date, and other information about the drawing.</p> <p>B Unavailable when saving drawings; available only when opening drawings.</p> | <p>C Turns the drawing preview on or off.</p> <p>D Saves the drawing with a password.</p> <p>E Displays an image of the drawing before you save it.</p> |
|--|--|

Saving a drawing with a new name or file format

You can save a drawing with a new name and in any of the following file formats:

- Standard drawing files with a .dwg extension. You can choose a .dwg file format that is compatible with various versions of AutoCAD.
- Drawing Exchange Format files with a .dxf file extension. You can choose a .dxf file format that is compatible with various versions of AutoCAD.
- Drawing templates with a .dwt file extension. Drawing templates allow you to easily create new drawings that reuse your drawing settings and entities.

To save a drawing with a new name or file format

- 1 Do one of the following to choose Save As:
 - On the ribbon Quick Access toolbar, click Save As, or on the ribbon Application button, choose Save As.
 - On the menu, choose File > Save As.
 - Type *saveas* and then press Enter.
- 2 In the Save Drawing As dialog box, under Save As Type, choose the file format.
- 3 Specify the name of the file you want to create.
- 4 Click Save.

Drawings can also be saved by exporting to various file formats.

For more details, see “Exporting drawings” on page 799.

Saving a drawing with a password

Sometimes drawings may contain confidential information and you want to control who has access to that information. If you need an added level of security, you can save a drawing with a password so only you and those who know the password can open the drawing.

To save a drawing with a password, you must save it as a .dwg file for AutoCAD version 2004 or newer.

Record passwords or use familiar words.

If you forget a drawing’s password, the drawing cannot be opened or recovered. You may want to store all passwords in a safe place.

To save a drawing with a password

- 1 Do one of the following to choose Save As:
 - On the ribbon Quick Access toolbar, click Save As, or on the ribbon Application button, choose Save As.
 - On the menu, choose File > Save As.
 - Type *saveas* and then press Enter.
- 2 In the Save Drawing As dialog box, under Save As Type, choose a .dwg file for AutoCAD version 2004 or newer.
- 3 Click Password Protect.
- 4 Choose a folder where you want to save the drawing.
- 5 In File Name, specify the name of the file you want to create.
- 6 Click Save.
- 7 In the Password dialog box, enter a password.
- 8 If desired, record and store the password in a safe place. If you forget the drawing’s password, the drawing cannot be opened or recovered.
- 9 Click OK.

Creating simple entities

Simple entities include lines (both finite and infinite), circles, arcs, ellipses, elliptical arcs, points, and rays. In addition, PTCAD includes a freehand sketch tool that can be used to create simple entities.

This section explains several methods for creating simple entities, including how to:

- Use menu commands on the Draw menu.
- Use the tools on the Draw toolbar.
- Type commands in the command bar.

In some cases, there are a number of different ways to create an entity. For the most part, one or two are given in this guide. Refer to the Command Reference in the online Help to learn how to create entities using other methods.

When you use a tool or a drawing command, the program prompts you to enter coordinate points, such as endpoints or insertion points. You can enter the points or distances either using a mouse or by typing coordinate values in the command bar. As you draw, PTCAD also displays a context-sensitive prompt box with appropriate additional options for the type of entity you are drawing.

After you create entities, you can modify them using the entity modification tools.


Topics in this chapter

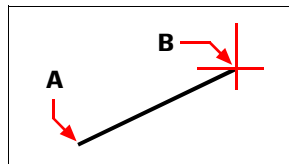
<i>Drawing lines</i>	112
<i>Drawing circles</i>	113
<i>Drawing arcs</i>	117
<i>Drawing ellipses</i>	121
<i>Drawing elliptical arcs</i>	122
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<i>Drawing construction lines</i>	128
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Drawing lines

A line consists of two points: a start point and an endpoint. You can connect a series of lines, but each line segment is considered a separate line entity.

To draw a line

- 1 Do one of the following to choose Line ():
 - On the ribbon, choose Home > Line or choose Draw > Line.
 - On the menu, choose Draw > Line.
 - On the Draw toolbar, click the Line tool.
 - Type *line* and then press Enter.
- 2 Specify the start point.
- 3 Specify the endpoint.
- 4 In the prompt box, choose Done to complete the command.




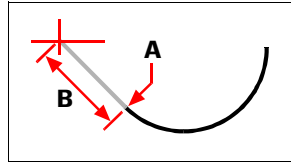
Start point (A) and endpoint (B).

The prompt box provides several options as you draw. For example, when you draw the first line segment, you can specify its length or orientation angle. After you draw at least one line segment, you can click Undo to remove the previous line segment. You can click Done to end the line command. After you draw two or more line segments, you can click Close to complete the line command by drawing a line segment that connects to the start point of the first line segment you drew.

If the last entity you drew was an arc, you can also draw a line tangent to and starting from the endpoint of the arc.

To draw a line as a continuation from the end of an arc








- 1 Do one of the following to choose Line ():
 - On the ribbon, choose Home > Line or choose Draw > Line.
 - On the menu, choose Draw > Line.
 - On the Draw toolbar, click the Line tool.
 - Type *line* and then press Enter.
- 2 In the prompt box, choose Follow.
- 3 Specify the length of the line.




Endpoint of previous arc (A) and length of the line (B).

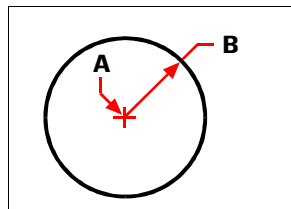
Drawing circles

The default method for drawing a circle is to specify a center point and radius. You can draw circles using any of the following methods:

- Center-Radius ()
- Center-Diameter ()
- 2-Points ()
- 3-Points ()
- Radius-Tangents ()
- Tan-Tan-Tan ()
- Convert Arc to Circle ()

To draw a circle by specifying its center and radius

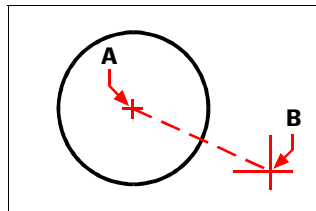
- 1 Do one of the following to choose Circle Center-Radius ():
 - On the ribbon, choose Home > Circle Center-Radius or choose Draw > Circle Center-Radius.
 - On the menu, choose Draw > Circle > Circle Center-Radius.
 - On the Draw toolbar, click the Circle Center-Radius tool.
 - Type *circle* and then press Enter.
- 2 Specify the center point.
- 3 Specify the radius of the circle.



Center point (A) and radius (B).

To draw a circle by specifying its center and diameter

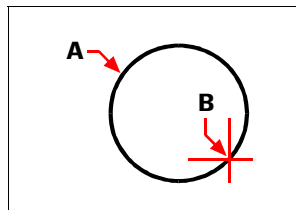
- 1 Do one of the following to choose Circle Center-Diameter (☺):
 - On the ribbon, choose Home > Circle Center-Diameter or choose Draw > Circle Center-Diameter. Skip step 3.
 - On the menu, choose Draw > Circle > Circle Center-Diameter. Skip step 3.
 - On the Draw toolbar, click the Circle tool.
 - Type *circle* and then press Enter.
- 2 Specify the center point.
- 3 In the prompt box choose Diameter.
- 4 Specify the second point indicating the diameter of the circle.



Center point (A) and diameter (B).

To draw a circle by specifying two endpoints of the diameter

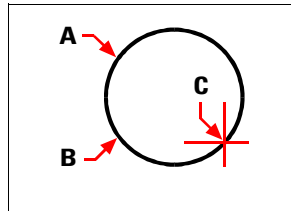
- 1 Do one of the following to choose Circle 2-Points (☺):
 - On the ribbon, choose Home > Circle 2-Points or choose Draw > Circle 2-Points. Go to step 3.
 - On the menu, choose Draw > Circle > Circle 2-Points. Go to step 3.
 - On the Draw toolbar, click the Circle tool.
 - Type *circle* and then press Enter.
- 2 In the prompt box choose 2 Points.
- 3 Specify the first endpoint of the diameter of the circle.
- 4 Specify the second endpoint of the diameter of the circle.



First endpoint (A) and second endpoint (B).

To draw a circle by specifying three points on the circumference

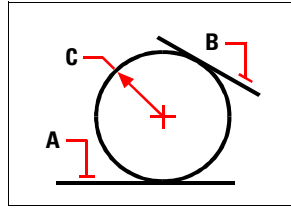
- 1 Do one of the following to choose Circle 3-Points (☺):
 - On the ribbon, choose Home > Circle 3-Points or choose Draw > Circle 3-Points. Go to step 3.
 - On the menu, choose Draw > Circle > Circle 3-Points. Go to step 3.
 - On the Draw toolbar, click the Circle tool.
 - Type *circle* and then press Enter.
- 2 In the prompt box choose 3 Points.
- 3 Specify the first point on the circumference of the circle.
- 4 Specify the second point on the circumference of the circle.
- 5 Specify the third point on the circumference of the circle.



First point (A), second point (B), third point (C).


To draw a circle by specifying its radius and two tangent points

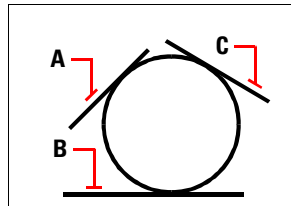
- 1 Do one of the following to choose Circle Radius-Tangents (☺):
 - On the ribbon, choose Home > Circle Radius-Tangents or choose Draw > Circle Radius-Tangents. Go to step 3.
 - On the menu, choose Draw > Circle > Circle Radius-Tangents. Go to step 3.
 - On the Draw toolbar, click the Circle tool.
 - Type *circle* and then press Enter.
- 2 In the prompt box, choose Tangent-Tangent-Radius.
- 3 Select the first entity that you want the circle to touch
- 4 Select the second entity that you want the circle to touch.
- 5 Specify the radius of the circle.



Tangent lines (A) and (B) and radius of circle (C).


To draw a circle tangent to three entities

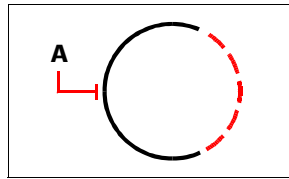
- 1 Do one of the following to choose Tan-Tan-Tan ():
 - On the ribbon, choose Home > Circle Tan-Tan-Tan or choose Draw > Circle Tan-Tan-Tan. Go to step 3.
 - On the menu, choose Draw > Circle > Tan-Tan-Tan. Go to step 3.
 - On the Draw toolbar, click the Circle tool.
 - Type *circle* and then press Enter.
- 2 In the prompt box, choose Tangent-Tangent-Tangent.
- 3 Select the first entity that you want the circle to touch
- 4 Select the second entity that you want the circle to touch.
- 5 Select the third entity you want the circle to touch.



First tangent entity (A), second tangent entity (B) and third tangent entity (C).

To convert an arc to a circle


















- 1 Do one of the following to choose Convert Arc to Circle ():
 - On the ribbon, choose Home > Convert Arc to Circle or choose Draw > Convert Arc to Circle.
 - On the menu, choose Draw > Circle > Convert Arc to Circle.
 - On the Draw toolbar, click the Convert Arc to Circle tool. Go to step 3.
 - Type *circle* and then press Enter.
- 2 In the prompt box, choose Turn Arc Into Circle.
- 3 Select the arc you want to convert to a circle.




Select an arc (A) to convert to a circle.

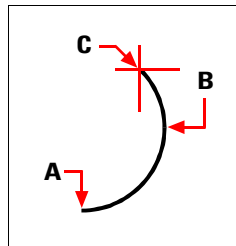
Drawing arcs

An arc is a portion of a circle. The default method for drawing an arc is to specify three points—the start point, a second point, and the endpoint. You can draw arcs using any of the following methods:

- Three points on an arc ().
- Start point-center-endpoint (), or Start point-endpoint-center (), or Center-start point-endpoint ().
- Start point-center-included angle (), or Start point-included angle-center (), or Center-start point-included angle ().
- Start point-center-chord length (), or Center-start point-chord length ().
- Start point-endpoint-radius (), or Start point-radius-endpoint ().
- Start point-endpoint-included angle (), or Start point-included angle-endpoint ().
- Start point-endpoint-starting direction (), or Start point-starting direction-endpoint ().
- Start point-radius-angle ().
- As a tangent continuation of the previous arc or line ().

To draw an arc by specifying three points


- 1 Do one of the following to choose 3-Point Arc (- On the ribbon, choose Home > 3-Point Arc (in Draw) or choose Draw > 3-Point Arc (in Draw).
- On the menu, choose Draw > Arc > 3-Point Arc.
- On the Draw toolbar, click the 3-Point Arc tool.
- Type *arc* and then press Enter.
- 2 Specify the start point.
- 3 Specify a second point.
- 4 Specify the endpoint.

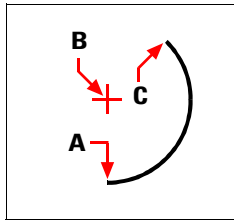


Start point (A), second point (B), and endpoint (C).

The prompt box provides additional options for drawing arcs. For example, after you specify the start point of an arc, you can choose Angle, Center, Direction, Endpoint, or Radius. You can select the options in a different order as well. For instance, you can draw an arc by specifying its start point, endpoint, and radius, or you can specify the start point, radius, and then endpoint.

To draw an arc by specifying its start point, center point, and endpoint

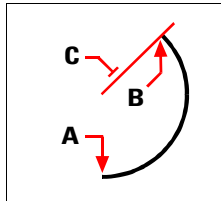
- 1 Do one of the following to choose Arc Start-Center-End (- On the ribbon, choose Home > Arc Start-Center-End (in Draw) or choose Draw > Arc Start-Center-End (in Draw).
- On the menu, choose Draw > Arc > Arc Start-Center-End.
- On the Draw toolbar, click the Arc Start-Center-End tool.
- Type *arc* and then press Enter.
- 2 Specify the start point.
- 3 In the prompt box, choose Center.
- 4 Specify the center point.
- 5 Specify the endpoint.



Start point (A), center point (B), and endpoint (C).

To draw an arc by specifying two points and an included angle


- 1 Do one of the following to choose Arc Start-End-Angle (↺):
 - On the ribbon, choose Home > Arc Start-End-Angle (in Draw) or choose Draw > Arc Start-End-Angle (in Draw).
 - On the menu, choose Draw > Arc > Arc Start-End-Angle.
 - On the Draw toolbar, click the Arc Start-End-Angle tool.
 - Type *arc* and then press Enter.
- 2 Specify the start point.
- 3 In the prompt box, choose Angle or type *angle*.
- 4 To draw an arc in a counterclockwise direction, enter a positive value for the included angle.
To draw an arc in a clockwise direction, enter a negative value for the included angle.
- 5 Specify the endpoint.

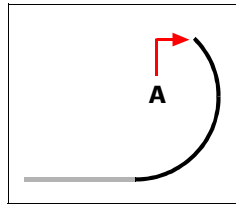


Start point (A), endpoint (B), and included angle (C).

If the last entity you drew was an arc or a line, you can also draw an arc tangent to and starting from the endpoint of the arc or line.


To draw an arc tangent to an arc or line

- 1 Do one of the following to choose Tangent Arc (- On the ribbon, choose Home > Tangent Arc (in Draw) or choose Draw > Tangent Arc (in Draw).
- On the menu, choose Draw > Arc > Tangent Arc.
- On the Draw toolbar, click the Tangent Arc tool. Go to step 3.
- Type *arc* and then press Enter.
- 2 In the prompt box, choose Follow.
- 3 Specify the endpoint.







Endpoint (A).

Arcs can be converted to circles


On the Draw toolbar, click the Convert Arc To Circle flyout tool ()

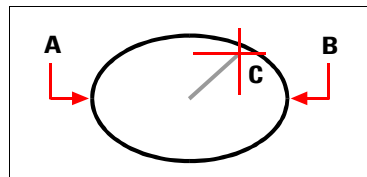
Drawing ellipses

The default method for drawing an ellipse is to specify the endpoints of one axis of the ellipse, and then specify a distance representing half the length of the second axis. The endpoints of the first axis determine the orientation of the ellipse. The longer axis of the ellipse is called the major axis, and the shorter one is the minor axis. The order in which you define the axes does not matter. The program determines the major and minor axes based on their relative lengths. You can draw ellipses using any of the following methods:

- Axis-Axis ()
- Axis-Rotation ()
- Center-Axes ()
- Center-Rotation ()

To draw an ellipse by specifying the axis endpoints





- 1 Do one of the following to choose Ellipse Axis-Axis ():
 - On the ribbon, choose Home > Ellipse Axis-Axis or choose Draw > Ellipse Axis-Axis.
 - On the menu, choose Draw > Ellipse > Ellipse Axis-Axis.
 - On the Draw toolbar, click the Ellipse Axis-Axis tool.
 - Type *ellipse* and then press Enter.
- 2 Specify the first endpoint.
- 3 Specify the second endpoint.
- 4 Specify the half-length of the other axis.




First axis endpoint (A), second axis endpoint (B), and half-length of other axis (C).

Drawing elliptical arcs

An elliptical arc is a portion of an ellipse. The default method for drawing an elliptical arc is to specify the endpoints of one axis of the ellipse, and then specify a distance representing half the length of the second axis. Then you specify the start and end angles for the arc, measured from the center of the ellipse in relation to its major axis. You can draw elliptical arcs using any of the following methods:

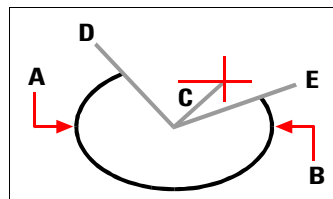
- Axis-Axis ()
- Axis-Rotation ()
- Center-Axes ()
- Center-Rotation ()

To draw an elliptical arc by specifying the axis endpoints

- 1 Do one of the following to choose Elliptical Arc Axis-Axis ():
 - On the ribbon, choose Home > Elliptical Arc Axis-Axis or choose Draw > Elliptical Arc Axis-Axis.
 - On the menu, choose Draw > Elliptical Arc > Elliptical Arc Axis-Axis.
 - On the Draw toolbar, click the Elliptical Arc Axis-Axis tool.
 - Type *ellipse* and then press Enter, and then type *a* (for Arc) and press Enter.
- 2 Specify the first endpoint.
- 3 Specify the second endpoint.
- 4 Specify the half-length of the other axis.
- 5 Specify the start angle of the arc.
- 6 Specify the end angle.

PTCAD draws elliptical arcs in the direction you specify.

Go to Tools > Drawing Settings > Drawing Units tab. Click the Angular Units tab. The default setting is counterclockwise.



First axis endpoint (A), second axis endpoint (B), half-length of other axis (C), start angle of arc (D), and end angle (E).

Drawing point entities

A point entity is a single x,y,z-coordinate location formatted as either a single dot or as one of 19 other possible display styles.

Drawing points

You can draw points one at a time or several at a time.

To draw a point

- 1 Do one of the following to choose Point (\square):
 - On the ribbon, choose Draw > Point.
 - On the menu, choose Draw > Point.
 - On the Draw toolbar, click the Point tool.
 - Type *point* and then press Enter.
- 2 Specify the location of the point.

To draw several points

- 1 Do one of the following to choose Point (\square):
 - On the ribbon, choose Draw > Point.
 - On the menu, choose Draw > Point.
 - On the Draw toolbar, click the Point tool.
 - Type *point* and then press Enter.
- 2 In the prompt box, choose Multiple Points.
- 3 Specify the location of each point.
- 4 In the prompt box, choose Done to complete the command.

Changing the size and appearance of point entities

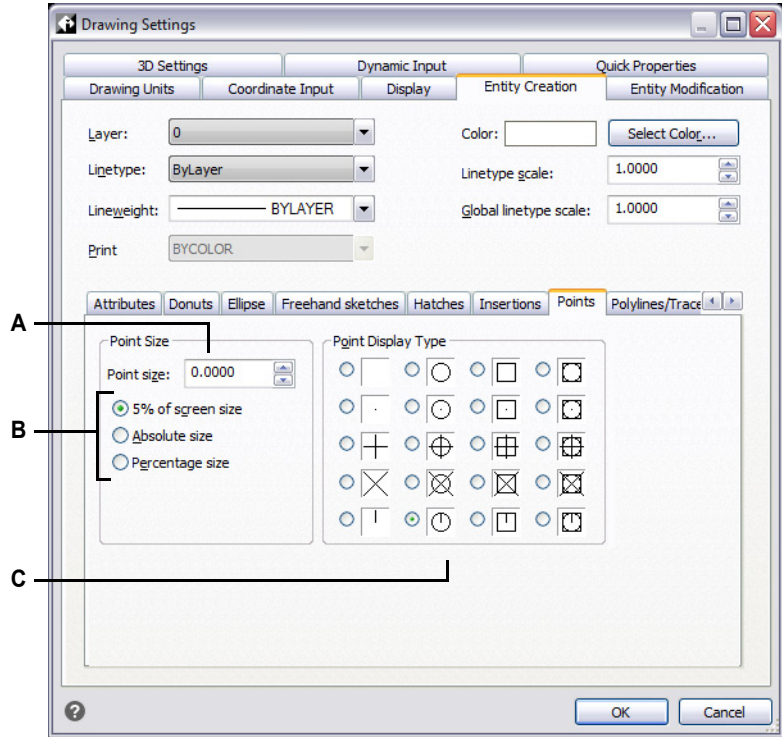
Changing the size and appearance of point entities affects all point entities already in the drawing, as well as all points that you subsequently draw. Positive values represent the absolute size of the point entity measured in drawing units. Negative values represent a percentage relative to the drawing screen, so that points retain their visual size as you use the Zoom command to change the magnification of the drawing.

To change the size and appearance of point entities

- 1 Do one of the following:
 - Choose Format > Point Style.
 - Type *ddptype* and then press Enter.
- 2 Under Point Display Type, select the style you want.
- 3 Under Point Size, select the point size, or choose one of the options.

4 Click OK.

When you regenerate the drawing, all point entities change to reflect the new size and appearance settings.




- A To increase or decrease the point size, type or select a value.
- B To use one of the preset point size options, click the one that you want.
- C Select the point display type that you want.

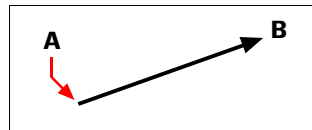
Drawing rays

A ray is a line in three-dimensional space that starts at a point and extends to infinity. Because rays extend to infinity, they are not calculated as part of the drawing extents. The default method for drawing a ray is to select the start point of the ray and then specify its direction. You can draw a ray in any of the following ways:

- Horizontal draws the ray parallel to the x-axis of the current user coordinate system (UCS).
- Vertical draws the ray parallel to the y-axis of the current UCS.
- Angle draws the ray parallel to a specified angle.
- Bisect draws the ray perpendicular to an existing entity.
- Offset draws the ray parallel to an existing entity.

To draw a ray

- 1 Do one of the following to choose Ray ():
 - On the ribbon, choose Home > Ray or choose Draw > Ray.
 - On the menu, choose Draw > Ray.
 - On the Draw toolbar, click the Ray tool.
 - Type *ray* and then press Enter.
- 2 Specify the start point.
- 3 Specify the direction.
- 4 To complete the command, press Enter.



Start point (A) and direction (B).


Drawing infinite lines

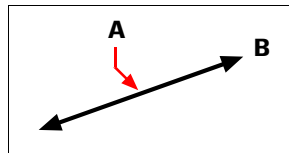
An infinite line is a line through a given point, oriented at a specified angle in three-dimensional space and extending to infinity in both directions. Because infinite lines extend to infinity, they are not calculated as part of the drawing extents.

The default method for drawing an infinite line is to select a point along the line and then specify the direction of the line. You can draw an infinite line in any of the following ways:

- Horizontal draws the infinite line parallel to the x-axis of the current UCS.
- Vertical draws the infinite line parallel to the y-axis of the current UCS.
- Angle draws the infinite line parallel to a specified angle.
- Bisect draws the infinite line perpendicular to an existing entity.
- Parallel draws the infinite line parallel to an existing entity.

To draw an infinite line


- 1 Do one of the following to choose Infinite Line ():
 - On the ribbon, choose Home > Infinite Line or choose Draw > Infinite Line.
 - On the menu, choose Draw > Infinite Line.
 - On the Draw toolbar, click the Infinite Line tool.
 - Type *infinite* and then press Enter.
- 2 Specify a point along the line.
- 3 Specify the direction.
- 4 To complete the command, press Enter.

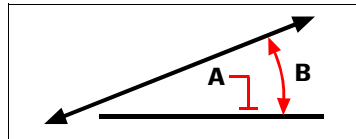


Point along the infinite line (A) and the direction (B).

You can also draw infinite lines at a specific angle or at an angle relative to an existing entity.

To draw an infinite line at a specified angle relative to another entity

- 1 Do one of the following to choose Infinite Line ():
 - On the ribbon, choose Home > Infinite Line or choose Draw > Infinite Line.
 - On the menu, choose Draw > Infinite Line.
 - On the Draw toolbar, click the Infinite Line tool.
 - Type *infinite* and then press Enter.
- 2 In the prompt box, choose Angle.
- 3 In the prompt box, choose Reference.
- 4 Select the reference entity.
- 5 Specify the angle of the infinite line in relation to the selected entity.
- 6 Specify the location of the infinite line.
- 7 To complete the command, press Enter.



Reference entity (A) and angle in relation to entity (B).

Drawing construction lines

A construction line is a special type of infinite line that is typically used when quick drafting techniques are required, for example, to make projections of front and side views.

Similar to an infinite line, a construction line is a line through a given point, oriented at a specified angle in three-dimensional space and extending to infinity in both directions. Because construction lines extend to infinity, they are not calculated as part of the drawing extents.

Construction lines are drawn automatically on their own layer, which by default is named CLINE. This makes it easy to change the appearance of construction lines; simply change the properties of the layer such as its color, transparency, etc. Additionally, you can specify a different layer for locating construction lines using the CLINELAYER system variable.

You can easily draw construction lines in any of the following ways:

- Select a point along the construction line and then specify the direction of the line (default drawing method).
- Horizontally, vertically, or both.
- By bisecting an existing point, an entity, or two entities.
- According to a specified angle.
- Parallel to or on top of an existing entity.

Construction lines are drawn to match the current UCS.

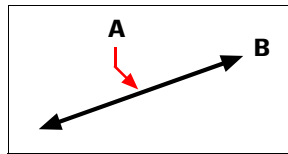
You can quickly rotate the UCS using Draw > Construction Line > Rotate UCS Z and Rotate UCS World (also available on the Construction Lines toolbar).

Drawing construction lines by specifying a basepoint

There are many quick ways to draw construction lines, but the default method is by specifying a basepoint along the construction line and then its direction. Using this method you can quickly draw an array of construction lines.

To draw a construction line by specifying a basepoint

- 1 Do one of the following to choose Construction Line > Array from Basepoint (✳):
 - On the ribbon, choose Draw > Construction Line > Array from Basepoint.
 - On the menu, choose Draw > Construction Line > Array from Basepoint.
 - On the Construction Line toolbar, click the Array from Basepoint tool.
 - Type *cline* then press Enter.
- 2 Specify a point along the line.
- 3 Specify the direction.
- 4 To complete the command, press Enter.



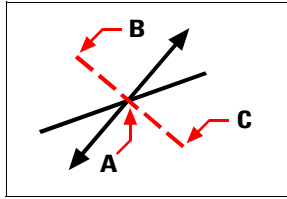
Point along the construction line (A) and the direction (B).

Drawing construction lines by bisecting an angle or entity

Bisection is a quick way to draw construction lines. You can bisect an angle that you specify, or you can bisect one or two lines, arcs, or polyline segments.


To draw a construction line by bisecting an angle

- 1 Do one of the following to choose Construction Line > Bisect Angle (↗):
 - On the ribbon, choose Draw > Construction Line > Bisect Angle.
 - On the menu, choose Draw > Construction Line > Bisect Angle.
 - On the Construction Line toolbar, click the Bisect Angle tool.
 - Type *cline*, press Enter, then choose Bisect in the prompt box.
- 2 Select an existing vertex point from which to reference the angle.
- 3 Specify the first point of the bisection angle.
- 4 Specify the second point of the bisection angle.
- 5 To complete the command, press Enter.




Point to bisect (A), start of the bisection angle (B), and the end of the bisection angle (C).

To draw a construction line by bisecting a line, arc, or polyline segment

- 1 Do one of the following to choose Construction Line > Bisect Entity (- On the ribbon, choose Draw > Construction Line > Bisect Entity.
- On the menu, choose Draw > Construction Line > Bisect Entity.
- On the Construction Line toolbar, click the Bisect Entity tool.
- Type *cline*, press Enter, choose Bisect in the prompt box, then choose Entity in the prompt box.
- 2 Select a line, arc, or polyline segment to bisect.

To draw a construction line by bisecting two lines, arcs, or polyline segments

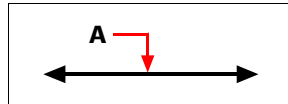
- 1 Do one of the following to choose Construction Line > Bisect Two Entities (- On the ribbon, choose Draw > Construction Line > Bisect Two Entities.
- On the menu, choose Draw > Construction Line > Bisect Two Entities.
- On the Construction Line toolbar, click the Bisect Two Entities tool.
- Type *cline*, press Enter, choose Bisect in the prompt box, then choose Two Entities in the prompt box.
- 2 Select a line, arc, or polyline segment.
- 3 Select the second line, arc, or polyline segment.

Drawing construction lines horizontally and vertically

Construction lines can be drawn parallel to the x-axis of the current UCS (horizontal), parallel to the y-axis of the current UCS (vertical), or parallel to both the x- and y-axis of the current UCS (both horizontal and vertical).

To draw a horizontal construction line

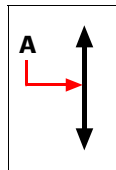
- 1 Do one of the following to choose Construction Line > Horizontal (↔):
 - On the ribbon, choose Draw > Construction Line > Horizontal.
 - On the menu, choose Draw > Construction Line > Horizontal.
 - On the Construction Line toolbar, click the Horizontal tool.
 - Type *cline* and then press Enter.
- 2 Specify a point along the line.
- 3 To complete the command, press Enter.



Point through which you want the horizontal construction line to pass (A).


To draw a vertical construction line

- 1 Do one of the following to choose Construction Line > Vertical (⋈):
 - On the ribbon, choose Draw > Construction Line > Vertical.
 - On the menu, choose Draw > Construction Line > Vertical.
 - On the Construction Line toolbar, click the Vertical tool.
 - Type *cline* and then press Enter.
- 2 Specify a point along the line.
- 3 To complete the command, press Enter.



Point through which you want the vertical construction line to pass (A).

To draw both horizontal and vertical construction lines

- 1 Do one of the following to choose Construction Line > Both Horizontal and Vertical (- On the ribbon, choose Draw > Construction Line > Both Horizontal and Vertical.
- On the menu, choose Draw > Construction Line > Both Horizontal and Vertical.
- On the Construction Line toolbar, click the Both Horizontal and Vertical tool.
- Type *cline* and then press Enter.
- 2 Specify a point where both construction lines intersect.
- 3 To complete the command, press Enter.


Horizontal and vertical construction lines can also be drawn automatically.

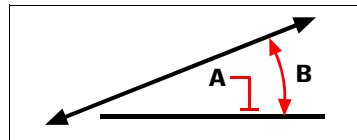
Choose Draw > Construction Line > Automatic Horizontal, Automatic Vertical, or Automatic Both, then select the entities around which you want to draw construction lines. Construction lines are drawn automatically without creating duplicates for overlapping vertices or entities.

Drawing construction lines by specifying the angle

You can also draw construction lines at a specific angle or at an angle relative to an existing entity.

To draw a construction line at a specified angle relative to another entity

- 1 Do one of the following to choose Construction Line > Angled (- On the ribbon, choose Draw > Construction Line > Angled.
- On the menu, choose Draw > Construction Line > Angled.
- On the Construction Line toolbar, click the Angled tool.
- Type *cline*, choose Angle, and then press Enter.
- 2 In the prompt box, choose Reference.
- 3 Select the reference entity.
- 4 Specify the angle of the construction line in relation to the selected entity.
- 5 Specify the location of the construction line.
- 6 To complete the command, press Enter.




Reference entity (A) and angle in relation to entity (B).

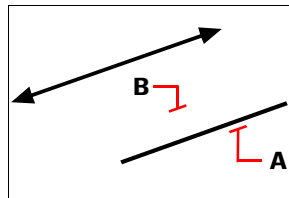
Drawing construction lines that parallel other entities

There are two options for drawing construction lines that parallel existing entities:

- **Offset** Draws a construction line parallel to an existing entity by a specified offset distance.
- **Match** Draws a construction line on top of an existing entity.


To draw a construction line that parallels another entity and specify an offset distance

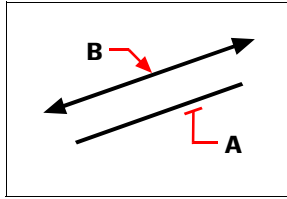
- 1 Do one of the following to choose Construction Line > Offset ():
 - On the ribbon, choose Draw > Construction Line > Offset.
 - On the menu, choose Draw > Construction Line > Offset.
 - On the Construction Line toolbar, click the Offset tool.
 - Type *cline*, choose Offset, and then press Enter.
- 2 Enter the distance you want the construction line drawn from an entity.
- 3 Select the entity that you want the construction line to parallel.
- 4 Select the side of the entity where you want to draw the construction line.
- 5 To complete the command, press Enter.



Entity you want to parallel (A) and the side for the construction line (B).


To draw a construction line that parallels another entity and specify a through point

- 1 Do one of the following to choose Construction Line > Offset ():
 - On the ribbon, choose Draw > Construction Line > Offset.
 - On the menu, choose Draw > Construction Line > Offset.
 - On the Construction Line toolbar, click the Offset tool.
 - Type *cline*, choose Offset, and then press Enter.
- 2 Choose Through point.
- 3 Select the entity that you want the construction line to parallel.
- 4 Specify the point through which you want the construction line to pass.
- 5 To complete the command, press Enter.



Entity you want to parallel (A) and the point through which you want the construction line to pass (B).


To draw a construction line that parallels another entity and matches its location

- 1 Do one of the following to choose Construction Line > On Entity ():
 - On the ribbon, choose Draw > Construction Line > On Entity.
 - On the menu, choose Draw > Construction Line > On Entity.
 - On the Construction Line toolbar, click the On Entity tool.
 - Type *cline*, choose Match, and then press Enter.
- 2 Select the entity that you want the construction line to match.
- 3 To complete the command, press Enter.


Erasing construction lines

You can delete construction lines as you would any other entity, but there are two tools designed to make it easy to filter them automatically when selecting them for deletion.

To delete one or more construction lines

- 1 Do one of the following to choose Construction Line > Erase ():
 - On the ribbon, choose Draw > Construction Line > Erase.
 - On the menu, choose Draw > Construction Line > Erase.
 - On the Construction Line toolbar, click the Erase tool.
 - Type *cline*, choose Erase, and then press Enter.
- 2 Select one or more construction lines to delete.
- 3 To complete the command, press Enter.

To delete all construction lines in a drawing

- 1 Do one of the following to choose Construction Line > Clear ():
 - On the ribbon, choose Draw > Construction Line > Clear.
 - On the menu, choose Draw > Construction Line > Clear.
 - On the Construction Line toolbar, click the Clear tool.
 - Type *cline*, choose Clear, and then press Enter.

Drawing freehand sketches

A freehand sketch consists of many straight line segments, created either as individual line entities or as a polyline.

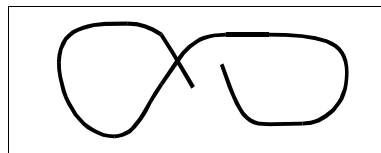
Creating freehand sketches

Before you begin creating a freehand sketch, you must set the length, or increment, of each segment. The smaller the segments, the more accurate your sketch, but segments that are too small can greatly increase the file size

After you specify the length of the sketch segments, the crosshairs change to a Pencil tool. Your freehand sketch is not added to the drawing until you “write” the sketch into your drawing. This means that you temporarily save the segment that you’ve drawn and the segment length, and you can continue sketching.

To create a freehand sketch

- 1 Do one of the following to choose Freehand (🖍️):
 - On the ribbon, choose Draw > Freehand (in Draw) or Draw 3D > Freehand (in Curves).
 - On the menu, choose Draw > Freehand.
 - On the Draw toolbar, click the Freehand tool.
 - Type *freehand* and then press Enter.
- 2 Specify the length of the sketch segments.
- 3 Click the mouse button to place the Pencil tool (🖍️) on the drawing to begin sketching.
- 4 Move the pencil image to draw a temporary freehand sketch.
- 5 Click the mouse button to lift the pencil up to stop sketching.
- 6 In the prompt box, choose Write, Then Resume to write the temporary freehand sketch into the drawing.
- 7 Click the mouse button to put the pencil down again and resume sketching.
- 8 Click the mouse button again to lift the pencil up to stop sketching.
- 9 In the prompt box, choose Done to write the temporary freehand sketch into the drawing and end the command.





A freehand sketch consists of individual line entities or a polyline.

Erasing freehand sketch lines

You can erase temporary freehand sketch lines that have not yet been written into the drawing by choosing the Delete On option in the prompt box. The pencil changes to an Eraser tool. You can erase portions of the line when you move the eraser over a temporary freehand line without clicking the mouse button.

To erase freehand sketch lines

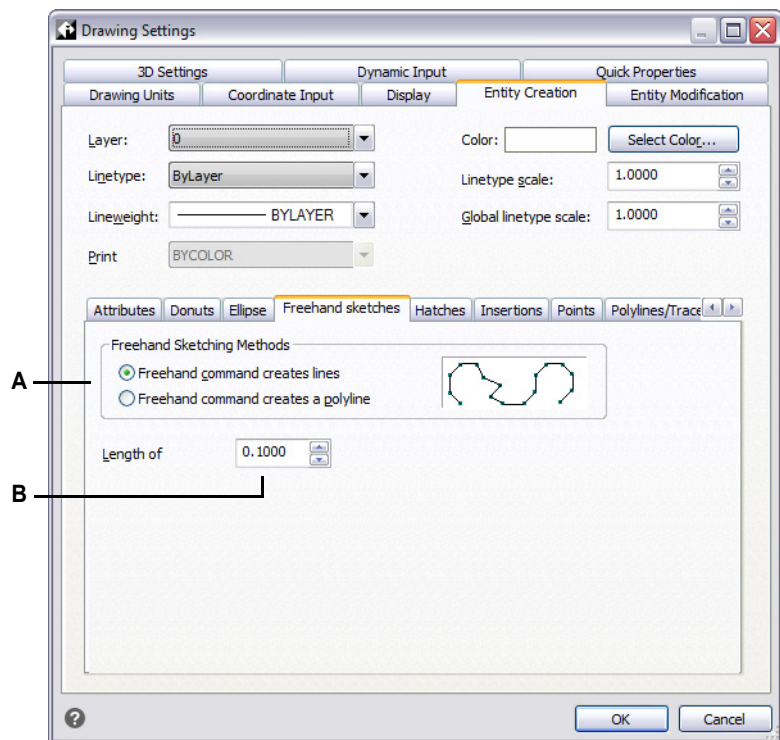
- 1 Do one of the following to choose Freehand (- On the ribbon, choose Draw > Freehand (in Draw) or Draw 3D > Freehand (in Curves).
- On the menu, choose Draw > Freehand.
- On the Draw toolbar, click the Freehand tool.
- Type *freehand* and then press Enter.
- 2 Specify the length of the sketch segments.
- 3 Select a point on the drawing to display the Pencil tool and begin sketching.
- 4 Click the mouse button to lift the pencil up to stop sketching.
- 5 In the prompt box, choose Delete On.
- 6 Move the Eraser tool () to the beginning or end of the freehand sketch line that you drew, and then move it as far along the line as you want to erase.
- 7 Click the mouse button to put the Pencil tool down to resume sketching.

Setting the sketch method and accuracy

Using polylines for freehand sketches makes it easier to go back and edit sketches. You control whether to create freehand sketches using line segments or polylines in the Drawing Settings dialog box. You can also control the length of sketch segments in this dialog box.

To specify lines or polylines when sketching

- 1 Do one of the following to choose Drawing Settings (📄):
 - On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
 - On the menu, choose Tools > Drawing Settings.
 - On the Tools toolbar, click the Drawing Settings tool.
 - Type *dsettings* and then press Enter.
- 2 Choose the Entity Creation tab.
- 3 Click the Freehand Sketches tab.
- 4 Under Freehand Sketching Methods, click either Freehand Command Creates Lines or Freehand Command Creates A Polyline.
- 5 Click OK.



A Select the sketch method.

B Specify the default length of sketch segments.

Creating complex entities

Complex entities include polylines (including rectangles, squares, and polygons), spline curves, donuts, and planes. In addition, PTCAD includes tools for adding tables, wipeouts, revision clouds, leaders, hatching, and gradients to your drawings. In some cases, there are a number of different ways to create an entity. For the most part, one or two are given in this guide. Refer to the Command Reference in the online Help to learn how to create entities using other methods.

When you type a drawing command or select a tool, the program prompts you to enter coordinate points, such as endpoints or insertion points. As you draw, the program displays a context-sensitive prompt box with appropriate additional options for the type of entity you are drawing.

After you create complex entities, you can modify them using the entity-modification tools. Several of these entity types require special editing commands.

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Drawing rectangles and squares

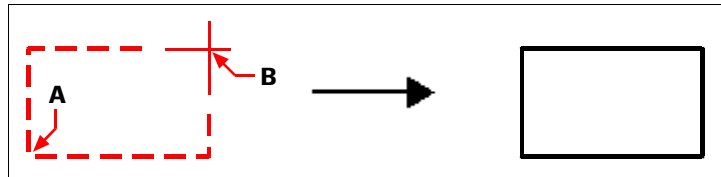
Rectangles are closed polylines with four sides and a square is a rectangle with four equal sides.

You draw a rectangle by specifying its opposite corners. The rectangle is normally aligned parallel to the current snap and grid alignment, but you can use the Rotated option to align the rectangle to any angle.




You draw a square using the Square option of the Rectangle command. The square is normally aligned parallel to the current snap and grid alignment, but you can use the Rotated option to align the square to any angle.

To draw a rectangle

- 1 Do one of the following to choose Rectangle ():
 - On the ribbon, choose Home > Rectangle or choose Draw > Rectangle.
 - On the menu, choose Draw > Rectangle.
 - On the Draw toolbar, click the Rectangle tool.
 - Type *rectangle* and then press Enter.
- 2 Specify one corner of the rectangle.
- 3 Specify the opposite corner of the rectangle.




Opposite corners (A and B) and resulting rectangle.

You can edit each side of a rectangle individually using the Edit Polyline tool () on the Modify toolbar. You can convert the sides into individual line entities using the Explode tool () on the Modify toolbar. You can control whether wide rectangle lines are shown filled or as outlines using the Fill tool () on the Settings toolbar.

You can also use the Rectangle tool to draw a square. Instead of specifying the opposite corners, you specify the length of one side and the alignment of the square.

To draw a square

- 1 Do one of the following to choose Rectangle ():
 - On the ribbon, choose Home > Rectangle or choose Draw > Rectangle.
 - On the menu, choose Draw > Rectangle.
 - On the Draw toolbar, click the Rectangle tool.
 - Type *rectangle* and then press Enter.
- 2 In the prompt box, choose Square.
- 3 Specify one corner of the square.
- 4 Specify the length of one side of the square by selecting its other end.
The point you specify determines both the size and alignment of the square.






Endpoints of one side of the square (A and B) and resulting square.

You can control the line width of rectangles.

Choose the Width Of Line option in the prompt box. After you change the width of the line, the new width setting remains in effect for subsequent rectangles until you change it again.

Drawing polygons


Polygons are closed polylines composed of a minimum of three and a maximum of 1,024 equal-length sides. The default method for drawing a polygon is to specify the center of the polygon and the distance from the center to each vertex. You can draw a polygon using any of the following methods:

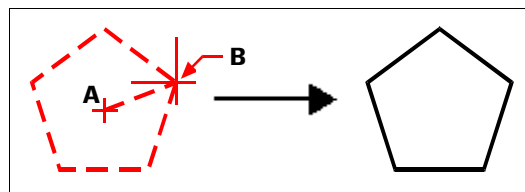
- Center-Vertex ()
- Center-Side ()
- Edge ()

Drawing polygons by vertex

You can create an equal-sided polygon defined by its center point and the distance to its vertices. First specify the number of sides, then the center point, and then the location of one vertex, which determines both the size and orientation of the polygon.

To draw a polygon by vertex

- 1 Do one of the following to choose Polygon Center-Vertex ():
 - On the ribbon, choose Home > Polygon Center-Vertex or choose Draw > Polygon Center-Vertex.
 - On the menu, choose Draw > Polygon > Polygon Center-Vertex.
 - On the Draw toolbar, click the Polygon Center-Vertex tool.
 - Type *polygon* and then press Enter.
- 2 Type 5 to specify five sides for the polygon.
- 3 Specify the center of the polygon.
- 4 Specify the vertex of the polygon.




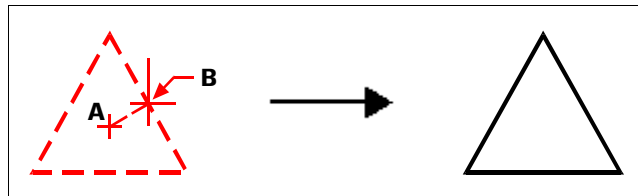
The center (A) and vertex (B) and resulting polygon.

Drawing polygons by side

You can create a polygon with equal-length sides defined by its center point and the distance to the midpoint of a side. First specify the number of sides, then the center point, and then the location of the midpoint of one side, which determines both the size and orientation of the polygon.

To draw a polygon by side

- 1 Do one of the following to choose Polygon Center-Side ():
 - On the ribbon, choose Home > Polygon Center-Side or choose Draw > Polygon Center-Side.
 - On the menu, choose Draw > Polygon > Polygon Center-Side.
 - On the Draw toolbar, click the Polygon Center-Side tool.
 - Type *polygon* and then press Enter.
- 2 Type 3 to specify three sides for the polygon.
- 3 Specify the center of the polygon.
- 4 Specify the midpoint of the side.




The center (A) and midpoint of one side (B) and resulting polygon.

Drawing polygons by specifying the length of an edge




You can create a polygon with equal-length sides defined by the length of one of the polygon edges. First specify the number of sides, then the edge start point, and then the edge end point, which determines both the size and orientation of the polygon.

To draw a polygon by specifying the length of an edge

- 1 Do one of the following to choose Polygon Edge ():
 - On the ribbon, choose Home > Polygon Edge or choose Draw > Polygon Edge.
 - On the menu, choose Draw > Polygon > Polygon Edge.
 - On the Draw toolbar, click the Polygon Edge tool.
 - Type *polygon* and then press Enter.
- 2 Type 5 to specify three sides for the polygon.
- 3 Specify the first point of an edge on the polygon.
- 4 Specify the second point of the edge on the polygon.

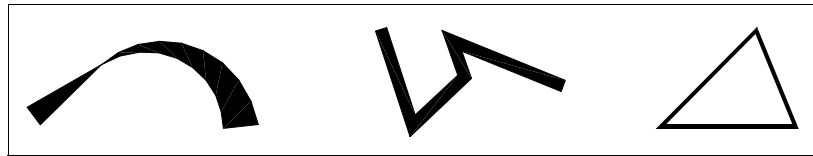


Start point of side (A) and end point of side (B) and resulting polygon.




You can edit each side of a polygon individually using the Edit Polyline tool () on the Modify toolbar. You can convert the sides into individual line entities using the Explode tool () on the Modify toolbar. You can control whether wide polygon lines are shown filled or as outlines using the Fill tool () on the Settings toolbar.

Drawing polylines

A polyline is a connected sequence of arcs and lines that is treated as a single entity. You can draw a polyline with any linetype and using a width that either remains constant or tapers over the length of any segment. When editing a polyline, you can modify the entire polyline or change individual segments.




Curved polyline with tapered width, polyline with straight segments, and closed polyline.


You can edit entire polylines and individual segments using the Edit Polyline tool () on the Modify toolbar. You can convert polylines into arc and line entities using the Explode tool () on the Modify toolbar. You can control whether wide polylines are shown filled or as outlines using the Fill tool () on the Settings toolbar.

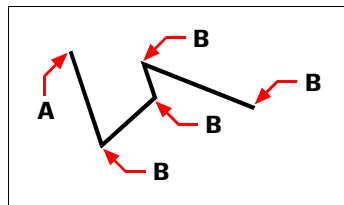
Drawing a polyline with straight segments

After you specify the start point of a polyline, a prompt box provides several options as you draw, such as Distance, Halfwidth, and Width. You can specify different starting and ending widths to create a tapered polyline segment.

After you draw at least one polyline segment, you can use the Undo tool () to remove the previous segment. After you draw two or more polyline segments, you can use the Close option to complete the command by drawing a segment that ends at the start point of the first polyline segment you drew. Choose Done to complete the command without closing the polyline.

To draw a polyline with straight segments

- 1 Do one of the following to choose Polyline ():
 - On the ribbon, choose Home > Polyline or choose Draw > Polyline.
 - On the menu, choose Draw > Polyline.
 - On the Draw toolbar, click the Polyline tool.
 - Type *polyline* and then press Enter.
- 2 Specify the start point.
- 3 Specify the endpoint of each segment.
- 4 To complete the command, choose Close or Done.



Polyline start point (A) and segment endpoints (B).


Drawing a polyline with arc segments

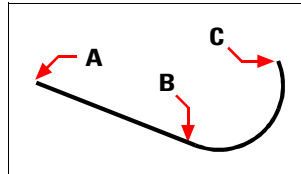
With the Draw Arcs option, you can continually draw arc segments until you select the Draw Lines option to go back to drawing line segments. When you draw arc segments, the first point of the arc is the endpoint of the previous segment. By default, you draw arc segments by specifying the endpoint of each segment. Each successive arc segment is drawn tangent to the previous arc or line segment. If you choose Close while in the Draw Arcs option, the closing segment is created as an arc.

You can also specify the arc using any of the following methods:

- Start point, included angle, center point
- Start point, included angle, radius
- Start point, center point, endpoint
- Start point, included angle, endpoint
- Start point, center point, included angle
- Start point, center point, chord length
- Start point, direction, endpoint
- Start point, radius, included angle
- Start point, second point, endpoint

To draw a line segment followed by an arc polyline segment

- 1 Do one of the following to choose Polyline ():
 - On the ribbon, choose Home > Polyline or choose Draw > Polyline.
 - On the menu, choose Draw > Polyline.
 - On the Draw toolbar, click the Polyline tool.
 - Type *polyline* and then press Enter.
- 2 Specify the start point.
- 3 Specify the endpoint.
- 4 In the prompt box, choose Draw Arcs.
- 5 Specify the endpoint of the arc segment.
- 6 To complete the command, choose Done.




Polyline start point (A), line endpoint/arc start point (B), and arc endpoint (C).

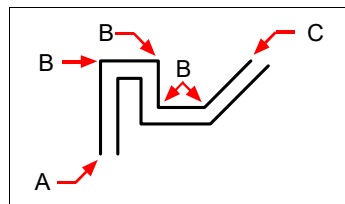
Drawing multilines

A multiline is made from multiple parallel lines (two lines by default), which consist of linear segments connected together. The end of the current segment is the start of the next segment. Ends of segments are the vertices of the multiline.

Drawing a multiline

To draw a multiline

- 1 Do one of the following to choose Multiline (- On the ribbon, choose Draw > Multiline.
- On the menu, choose Draw > Multiline.
- On the Draw toolbar, click the Multiline tool.
- Type *mline* and then press Enter.
- 2 Specify the start point.
- 3 Specify additional vertices.
- 4 After specifying the last endpoint, choose Done or press Enter.



Multiline startpoint (A), vertices (B), and endpoint (C).


Specifying justification and scale

When you draw a multiline, you specify the vertices of one of the lines that makes up the multiline — the additional parallel lines are drawn in position according to the justification. Vertices can be specified on the top, middle, or bottom of a multiline, according to the selected justification.

You can also determine the overall width of the multiline, which affects the distance between parallel lines, by adjusting the multiline scale.

NOTE *If you change the multiline scale, you might need to make equivalent changes to the linetype scale to prevent dots or dashes from being disproportionately sized.*

To draw a multiline with different justification or scale

- 1 Do one of the following to choose Multiline (- On the ribbon, choose Draw > Multiline.
- On the menu, choose Draw > Multiline.
- On the Draw toolbar, click the Multiline tool.
- Type *mline* and then press Enter.
- 2 Select justification and choose one of the following options:
 - **Top** Specified vertices define the top line; additional parallel lines are drawn below the specified vertices.
 - **Zero** Specified vertices define the middle of the multiline.
 - **Bottom** Specified vertices define the bottom line; additional parallel lines are drawn above the specified vertices.
- 3 Select Scale and enter a new scale value according to the following:
 - Greater than 1 — Multilines are wider.
 - Less than 1 — Multilines are narrower.
 - Equal to 1 — Multilines collapse into a single line.
 - Negative value — Flips the justification when multilines are drawn and alters the scale according to the set value.
- 4 Specify the start point of the multiline.
- 5 Specify additional vertices.
- 6 After specifying the endpoint, choose Done or press Enter.

Working with multiline styles

When you create a multiline, the program draws it on the current layer, using the current multiline style. Each multiline has a corresponding multiline style, which controls the appearance of the multiline, such as the number of parallel lines, the spacing between each line and each line's color and linetype, start and end caps, and more.

You can create, save, modify, and delete named multiline styles. You can also set any multiline style to be the current one that is assigned to new multilines as you add them to your drawing. Multiline styles can be exported and imported as .mln files which makes it easy to share them between computers.


Multiline styles can also be managed using the Explorer. For more details, see "Working with multiline styles" on page 315.

Creating or modifying a multiline style


Multiline styles provide a way for you to change various settings that control the appearance of multilines. You can then save those settings for reuse. If you don't define a multiline style before creating multilines, the program uses the Standard multiline style, which stores the default multiline settings.

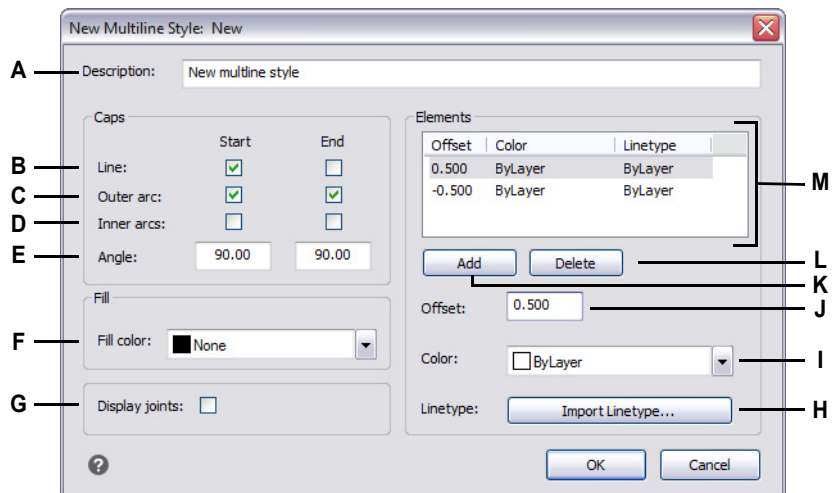
If you modify a multiline style, all multilines entities that are assigned that style will be updated.

To create a multiline style

- 1 Do one of the following to choose Multiline Styles Manager (- On the ribbon, choose Express Tools > Multiline Styles Manager (in Tools).
- On the menu, choose Format > Multiline Styles Manager or choose Express Tools > Tools > Multiline Styles Manager.
- On the Styles toolbar, click the Multiline Styles Manager tool.
- Type *mlstyle* and then press Enter.
- 2 Click New.
- 3 In Style Name, type the name of the new multiline style.
- 4 In Start With, select an existing multiline style on which to base the new multiline style.
- 5 Click Continue.
- 6 In the Create New Multiline Style dialog box, make your selections for the multiline style as necessary.
- 7 When finished, click OK.

To modify a multiline style

- 1 Do one of the following to choose Multiline Styles Manager ():
 - On the ribbon, choose Express Tools > Multiline Styles Manager (in Tools).
 - On the menu, choose Format > Multiline Styles Manager or choose Express Tools > Tools > Multiline Styles Manager.
 - On the Styles toolbar, click the Multiline Styles Manager tool.
 - Type *mstyle* and then press Enter.
- 2 Select the multiline style you want to modify, then click Modify.
- 3 In the Modify Multiline Style dialog box, make your selections for the multiline style as necessary.
- 4 When finished, click OK.




- A Enter a description for the multiline style.
- B Select whether to include a line to cap the start or end of multilines assigned this style.
- C Select whether to include an arc for outer lines to cap the start or end of multilines assigned this style.
- D Select whether to include an arc for inner lines to cap the start or end of multilines assigned this style.
- E Select whether to include an angled line to cap the start or end of multilines assigned this style.
- F Select the color fill for multilines assigned this style.
- G Select whether to include a line at joints of multilines assigned this style. If turned on, a line is drawn from the two outer vertices of the multiline's joint.
- H Click to select the linetype for the element.
- I Select the line color for the element.
- J Enter the distance to offset the element, or line, from the previous element.
- K Click to add a new element according to the current offset, color, and linetype selections.
- L Click to delete the selected element.
- M Displays a list of the offset, color, and linetype combinations assigned to each element, or line.

Setting the current multiline style

When you create a multiline, it is automatically assigned the multiline style that is set to current. By default, the program uses the Standard multiline style, which stores the default multiline settings. Or you can choose a different multiline style to be the current one.

If you choose a different multiline style as current, that style is assigned to multilines that you create in the future, not existing multilines.

To set a multiline style as current

- 1 Do one of the following to choose Multiline Styles Manager (- On the ribbon, choose Express Tools > Multiline Styles Manager (in Tools).
- On the menu, choose Format > Multiline Styles Manager or choose Express Tools > Tools > Multiline Styles Manager.
- On the Styles toolbar, click the Multiline Styles Manager tool.
- Type *mstyle* and then press Enter.
- 2 In the Multiline Styles Manager dialog box, select a multiline style from the list.
- 3 Click Set Current.
- 4 Click OK.

Assigning a style to a multiline

You can assign a different multiline style to a multiline at any time.


To assign a style to a multiline

- 1 Select a multiline in the drawing.
- 2 Do one of the following:
 - On the Styles toolbar, select the style in the Multiline Style list.
 - Right-click the multiline, choose Properties, then in Style, select the style.

Renaming a multiline style

Use the Multiline Styles Manager dialog box to rename a multiline style. Note that the Standard multiline style cannot be renamed.


To rename a multiline style

- 1 Do one of the following to choose Multiline Styles Manager (- On the ribbon, choose Express Tools > Multiline Styles Manager (in Tools).
- On the menu, choose Format > Multiline Styles Manager or choose Express Tools > Tools > Multiline Styles Manager.
- On the Styles toolbar, click the Multiline Styles Manager tool.
- Type *mlstyle* and then press Enter.
- 2 In the Multiline Styles Manager dialog box, select the multiline style to rename.
- 3 Click Rename.
- 4 Type the new name, then click OK.
- 5 Click OK.

Deleting a multiline style

Use the Multiline Styles Manager dialog box to delete a multiline style. Note that the Standard multiline style cannot be deleted, and multiline styles that are in use also cannot be deleted.


To delete a named multiline style

- 1 Do one of the following to choose Multiline Styles Manager (- On the ribbon, choose Express Tools > Multiline Styles Manager (in Tools).
- On the menu, choose Format > Multiline Styles Manager or choose Express Tools > Tools > Multiline Styles Manager.
- On the Styles toolbar, click the Multiline Styles Manager tool.
- Type *mlstyle* and then press Enter.
- 2 In the Multiline Styles Manager dialog box, select the multiline style to delete.
- 3 Click Delete.
- 4 To confirm the deletion, click OK.
- 5 Click OK.


Importing and exporting multiline style files

Multiline styles can be imported and exported as .mln files which makes it easy to move multiline styles between computers.

To export a multiline style file

- 1 Do one of the following to choose Multiline Styles Manager (- On the ribbon, choose Express Tools > Multiline Styles Manager (in Tools).
- On the menu, choose Format > Multiline Styles Manager or choose Express Tools > Tools > Multiline Styles Manager.
- On the Styles toolbar, click the Multiline Styles Manager tool.
- Type *mlstyle* and then press Enter.
- 2 In the Multiline Styles Manager dialog box, select the multiline style to export.
- 3 Click Export.
- 4 Specify the name and location for the .mln file, then click Save.
- 5 Click OK.

To import a multiline style file

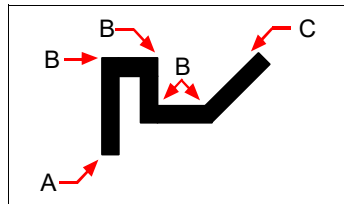
- 1 Do one of the following to choose Multiline Styles Manager (- On the ribbon, choose Express Tools > Multiline Styles Manager (in Tools).
- On the menu, choose Format > Multiline Styles Manager or choose Express Tools > Tools > Multiline Styles Manager.
- On the Styles toolbar, click the Multiline Styles Manager tool.
- Type *mlstyle* and then press Enter.
- 2 In the Multiline Styles Manager dialog box, click Import.
The Import Multiline Style from File dialog box displays a list of the multiline styles that have already been loaded.
- 3 Click File.
- 4 Navigate to and select the .mln file to import, then click Open.
- 5 Click OK.

Drawing traces

With the Trace command, you can draw a two-dimensional solid line of specified width. The Trace command is not commonly used — most users use the Polyline command instead.

To draw a trace

- 1 Type *trace* and then press Enter. Or on the ribbon, choose Draw > Trace.
- 2 Specify the width of the trace.
- 3 Specify the start point.
- 4 Specify additional vertices.
- 5 After specifying the last endpoint, choose Done or press Enter.



Trace startpoint (A), vertices (B), and endpoint (C).

Drawing splines

A spline is a smooth curve. You can use splines to create sculptured shapes such as the cross section of a turbine blade or an airplane wing.

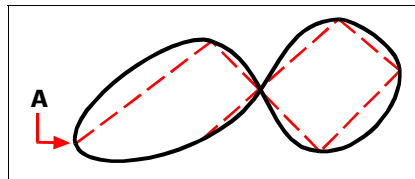
You can define the shape of the spline with either fit points or control vertices.

Drawing a spline by specifying fit points

Splines can pass through or near fit points that you specify to define the spline shape.


By default, a spline passes through all of the fit points, but when you draw a spline, you can change this by specifying the fit tolerance. The fit tolerance value determines how closely the spline fits the set of points you specify. For example, a fit tolerance value of 0 causes the spline to pass through the fit points. A value of 0.01 creates a spline that passes through the start and endpoints and within 0.01 units of the intermediate fit points.

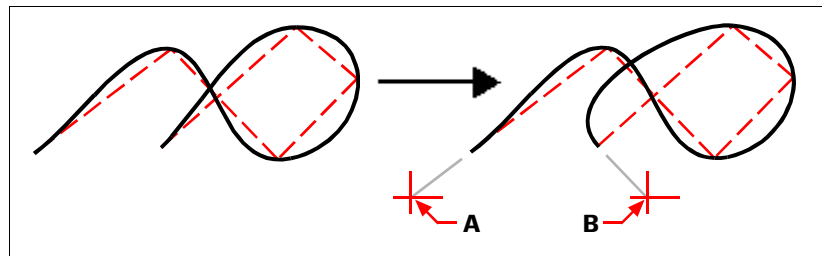
Fit-point splines are additionally affected by knot parametrization, can include tangents, and can be open or closed.



The start point and endpoint (A) of a closed fit-point spline.

To draw a spline that passes through or near fit points

- 1 Do one of the following to choose Spline Fit ():
 - On the ribbon, choose Draw > Spline Fit (in Draw) or Draw 3D > Spline Fit (in Curves).
 - On the menu, choose Draw > Spline > Spline Fit.
 - On the Draw toolbar, click the Spline Fit tool.
 - Type *spline*, press Enter, choose Method, then choose Fit.
 - 2 Specify the first fit point of the spline.
 - 3 Optionally choose Knots, and choose Chord, Square root, or Uniform to determine how the curve segments between successive fit points within a spline are blended.
 - 4 Optionally choose Fit Tolerance, enter a new fit tolerance, and then press Enter.
 - 5 Specify the second fit point of the spline.
 - 6 Continue specifying as many fit points as you want, then press Enter or choose Close to close the spline.
 - 7 Specify the start tangent point to give the spline additional shape, or press Enter if you do not want to add a start tangent.
 - 8 Specify the end tangent point to give the spline additional shape, or press Enter if you do not want to add an end tangent.
- If you are drawing a closed spline, you are prompted for only one tangent.



Initial spline and resulting spline with starting tangent point (A) and ending tangent point (B).


The display of control vertices can be turned on or off, even for fit-point splines.

Use the *Show CV* command to display control vertices; use the *Hide CV* command to hide them.

Drawing a spline by specifying control vertices

Splines drawn using control vertices display with a control frame when the spline is selected. You can then use the control frame to modify the shape of the spline. Control-vertex splines are affected by polynomial degrees and can be open or closed.


To draw a spline defined by control vertices

- 1 Do one of the following to choose Spline CV (- On the ribbon, choose Draw > Spline CV (in Draw) or Draw 3D > Spline CV (in Curves).
- On the menu, choose Draw > Spline > Spline CV.
- On the Draw toolbar, click the Spline CV tool.
- Type *spline*, press Enter, choose Method, then choose CV.
- 2 Specify the first control vertex of the spline.
- 3 Optionally choose Degree and enter the polynomial degree for the spline: 1 for linear, 2 for quadratic, 3 for cubic, etc. up to 10.
- 4 Specify the second control vertex of the spline.
- 5 Continue specifying as many control vertices as you want.
- 6 When you have finished, press Enter or choose Close to close the spline.

Converting a spline-fit polyline to a spline

Both 2D and 3D spline-fit polylines can be converted to splines. If necessary, you can use the Polyline Edit command (Spline option) to convert a polyline to a spline-fit polyline.

To convert a spline-fit polyline to a spline

- 1 Do one of the following to choose Spline CV or Spline Fit (- On the ribbon, choose Draw > Spline CV (in Draw) or Draw 3D > Spline CV (in Curves).
- On the menu, choose Draw > Spline > Spline CV.
- On the Draw toolbar, click the Spline CV tool.
- Type *spline* and then press Enter.
- 2 Choose Entity.
- 3 Select the 2D or 3D polyline that fits the spline, then press Enter.

Other two-dimensional entities can be converted to splines using the Convert to NURBS command.

The Convert to NURBS command is typically used to convert solids and surfaces to NURBS surfaces, but can also be used to convert two-dimensional entities (for example, lines, polylines, arcs, circles, etc.) to splines. For more details, see “Converting surfaces and solids to NURBS surfaces” on page 745.


Drawing helices

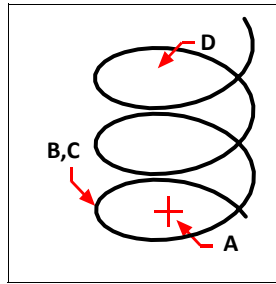
A helix is a three-dimensional spiral that is open at both ends. The base and top can be equal values, similar to a coil or spring, or they can be different values, similar to a cone.

Helices are often used to create other entities.

In particular, the Sweep, Loft, and Revolve commands can all be used with helices in more complex drawings to create spiral stairs, spiral parts, and more.

To draw a helix

- 1 Do one of the following to choose Helix (- On the ribbon, choose Draw > Helix (in Draw) or Draw 3D > Helix (in Curves).
- On the menu, choose Draw > Helix.
- On the Draw toolbar, click the Helix tool.
- Type *helix* and then press Enter.
- 2 Specify the center point of the helix.
- 3 Specify the radius (or diameter) of the bottom of the helix.
- 4 Specify the radius (or diameter) of the top of the helix.
- 5 Optionally set any of the following:
 - **Turns** Specify the number of full turns. The default number of turns is three. The maximum is 500.
 - **Turn height** Specify the height of one full turn, which updates the number of turns automatically. This option is available only if you have not specified a number of turns.
 - **Twist** Specify the direction of helix turns: clockwise (CW) or counterclockwise (CCW).
- 6 Specify the height and position of the helix by choosing one of the following:
 - Choose **Axis endpoint** and specify the endpoint of the helix, which determines the position and direction of the helix.
 - Specify the height of the helix. If the height is zero, the helix will be a two-dimensional spiral, but if the start and end radius are the same it will look like a circle.




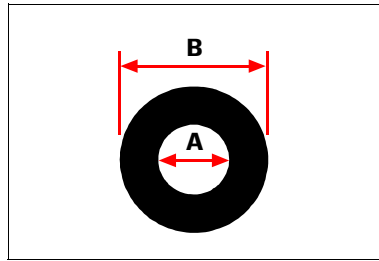
Helix center point (A), bottom radius (B), top radius (C), and height (D).

Drawing donuts

Donuts are solid, filled circles or rings created as closed, wide polylines. You can draw a donut using one of several methods. The default method is to specify the inside and outside diameters of the donut, and then specify its center. You can then create multiple copies of the same donut by specifying different center points until you press Enter to complete the command.

To draw a donut

- 1 Do one of the following to choose Donut ():
 - On the ribbon, choose Draw > Donut.
 - On the menu, choose Draw > Donut.
 - On the Draw toolbar, click the Donut tool.
 - Type *donut* and then press Enter.
- 2 Specify the inside diameter of the donut.
- 3 Specify the outside diameter of the donut.
- 4 Specify the center of the donut.
- 5 Specify the center point to draw another donut, or choose Done to complete the command.



Inside diameter (A) and outside diameter (B).

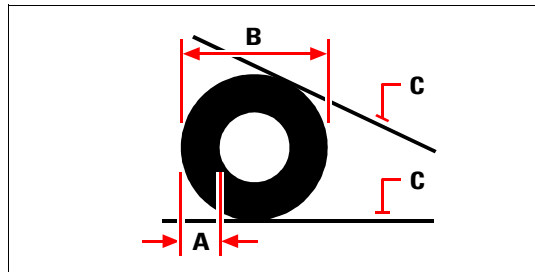
The prompt box provides additional options for drawing donuts. For example, you can specify the width of the donut and two points on the diameter of the donut, or you can specify the width and three points on the donut. You can also draw a donut tangent to existing entities.

A donut can be a completely filled circle.




Donuts are completely filled if the inside diameter is zero.

To draw a donut tangent to existing entities

- 1 Do one of the following to choose Donut (☉):
 - On the ribbon, choose Draw > Donut.
 - On the menu, choose Draw > Donut.
 - On the Draw toolbar, click the Donut tool.
 - Type *donut* and then press Enter.
- 2 In the prompt box, choose Radius Tangent Tangent.
- 3 Specify the width of the donut.
- 4 Specify the diameter of the donut.
- 5 Select the first tangent entity to which to draw the donut.
- 6 Select the second tangent entity to which to draw the donut.



Width (A) and diameter (B) of the donut and tangent entities (C).

You can edit donuts using the Edit Polyline tool () on the Modify toolbar. You can convert donuts into arc entities using the Explode tool () on the Modify toolbar. You can control whether donuts are shown filled or as outlines using the Fill tool () on the Settings toolbar.

You can control the default outside and inside diameter of donuts.


Choose Tools > Drawing Settings, click the Entity Creation tab, and choose the options you want.

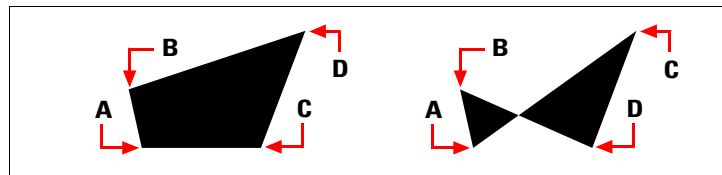
Creating planes

With the Plane tool, you can draw rectangular, triangular, or quadrilateral areas filled with a solid color. The default method is to specify the corners of the plane. After you specify the first two corners, the plane is displayed as you specify the remaining corners. Specify corner points in a triangular manner. The program prompts you for the third point and then the fourth point. If you continue specifying points, the third- and fourth-point prompts toggle until you press Enter to complete the command.

Creating planes in PTCAD is similar to the Solid command in AutoCAD.

To draw a quadrilateral plane


- 1 Do one of the following to choose Plane ():
 - On the ribbon, choose Draw > Plane.
 - On the menu, choose Draw > Plane.
 - On the Draw toolbar, click the Plane tool.
 - Type *plane* and then press Enter.
- 2 Specify the first point.
- 3 Specify the second point.
- 4 Specify the third point.
- 5 Specify the fourth point.
- 6 To complete the command, press Enter.

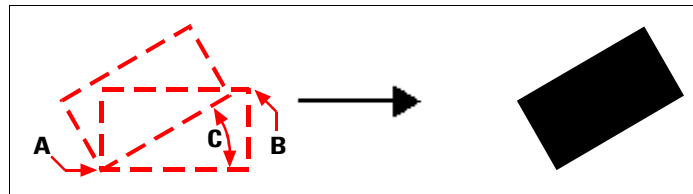


After you select the first two points (A) and (B), the sequence in which you select the third (C) and fourth (D) points determines the shape of the resulting quadrilateral plane.



The prompt box provides additional options for drawing planes. For example, you can draw rectangular, square, or triangular planes.

To draw a rectangular plane

- 1 Do one of the following to choose Plane ():
 - On the ribbon, choose Draw > Plane.
 - On the menu, choose Draw > Plane.
 - On the Draw toolbar, click the Plane tool.
 - Type *plane* and then press Enter.
- 2 Choose Rectangle.
- 3 Specify the first point.
- 4 Specify the opposite corner.
- 5 Specify the rotation angle.
- 6 To complete the command, specify the opposite corner to draw another rectangle, or press Enter.



Opposite corners (A and B) and rotation angle (C) and resulting plane.

You can control whether planes are shown filled or as outlines using the Fill tool () on the Settings toolbar. You can convert planes into individual line entities corresponding to the outline of the plane using the Explode tool () on the Modify toolbar.

Drawing architectural entities


Architectural entities, commonly called AEC entities, are custom entities that you can use to quickly draw walls, doors, windows, slabs, roofs, and more.

Drawing a wall

Walls can be drawn as interior, exterior, fireproof, foundational, and more.

After you draw a wall, you can modify it as you would other entities using modify entity commands. You can also use the Properties pane to change the width, style, automatic cleanup, etc. And when you select a wall, grips display allowing you to resize and reorient the wall in real-time.

To draw a wall


- 1 Do one of the following to choose Walls ():
 - On the ribbon, choose AEC > Walls and then the wall style (in Draw).
 - On the menu, choose AEC > Walls and then the wall style.
 - On the Walls toolbar, click a wall style.
 - Type *wall* and then press Enter.
- 2 Specify the start point of the wall.
- 3 Specify the endpoint of each segment.
- 4 After specifying the last endpoint, press Enter.

Drawing a door

Doors can be drawn in a wide variety of styles: single swing, double swing, sliding, revolving, and many more.

After you draw a door, you can modify it as you would other entities using modify entity commands. For example, doors can be easily moved and resized within a wall. You can also use the Properties pane to change the width, style, etc. And when you select a door, grips display allowing you to resize and reorient the door in real-time. There is also a special arrow grip for doors that allows you to change the swing with a single click.

To draw a door


- 1 Do one of the following to choose Doors ():
 - On the ribbon, choose AEC > Doors and then the door style (in Draw).
 - On the menu, choose AEC > Doors and then the door style.
 - On the Doors toolbar, click a door style.
 - Type *door* and then press Enter.
- 2 Select the wall where you want to locate the door, or press Enter if not choosing an existing wall.
- 3 Specify the point where you want to locate the door.
- 4 After specifying the last location, press Enter.

Drawing a window

Windows can be drawn as single hung, double hung, picture, pivot, and many more.

After you draw a window, you can modify it as you would other entities using modify entity commands. For example, windows can be easily moved and resized within a wall. You can also use the Properties pane to change the width, style, etc. And when you select a window, grips display allowing you to resize and reorient the window in real-time. There is also a special arrow grip for windows that allows you to change the opening orientation with a single click.

To draw a window


- 1 Do one of the following to choose Windows ():
 - On the ribbon, choose AEC > Windows and then the window style (in Draw).
 - On the menu, choose AEC > Windows and then the window style.
 - On the Windows toolbar, click a window style.
 - Type *window* and then press Enter.
- 2 Select the wall where you want to locate the window, or press Enter if not choosing an existing wall.
- 3 Specify the point where you want to locate the window.
- 4 After specifying the last location, press Enter.

Drawing stairs

Stairs can be drawn in different types and styles such as spiral, straight, cantilever, a ramp, and more.

After you draw stairs, you can modify them as you would other entities using modify entity commands. For example, stairs can be easily moved and resized relative to a wall or slab. You can also use the Properties pane to change the stair width, tread dimensions, landing dimensions, turn on/off stringers, etc. When stairs are modified, any attached railings are updated to match the stair modifications (length, angle, width, etc.). And when you select stairs, grips display allowing you to resize and reorient them in real-time.

To draw stairs


- 1 Do one of the following to choose Stairs ():
 - On the ribbon, choose AEC > Stairs and then the stair style (in Draw).
 - On the menu, choose AEC > Stairs and then the stair style.
 - On the Stairs toolbar, click a stair style.
 - Type *stair* and then press Enter.
- 2 If necessary, choose an option to change the default value for the stairs you want to draw.
- 3 When done entering values, specify the point where you want to start the stairs.
- 4 Enter the stair segment length, then press Enter.
- 5 Specify the endpoint of the stairs.

Drawing a railing

Railings can be drawn in different styles such as standard, bollard, vertical or horizontal cable, and many more.


After you draw a railing, you can modify it as you would other entities using modify entity commands. For example, standalone railings can be easily moved and resized. You can also use the Properties pane to change the style, rail height, post placement, etc. And when you select a railing, grips display allowing you to resize and reorient the railing in real-time.

To draw a railing attached to stairs

- 1 Do one of the following to choose Railings ():
 - On the ribbon, choose AEC > Railings and then the railing style (in Draw).
 - On the menu, choose AEC > Railings and then the railing style.
 - On the Railings toolbar, click a railing style.
 - Type *railing* and then press Enter.
- 2 Choose Attach.

- 3 Specify the stairs where you want to attach a railing.
- 4 Choose the location of the railing on the stairs: left, right, or center.

To draw a railing that is not attached to stairs


- 1 Do one of the following to choose Railings ():
 - On the ribbon, choose AEC > Railings and then the railing style (in Draw).
 - On the menu, choose AEC > Railings and then the railing style.
 - On the Railings toolbar, click a railing style.
 - Type *railing* and then press Enter.
- 2 Specify the start point where you want to locate the railing.
- 3 Specify additional points to create railing segments.
- 4 After specifying the last point, press Enter.

Drawing a curtain wall

Curtain walls can be drawn in different styles and can also be converted from existing two-dimensional entities such as lines, arcs, circles, and polylines.


After you draw a curtain wall, you can modify it as you would other entities using modify entity commands. You can also use the Properties pane to change settings for divisions, infill, mullion and frame. And when you select a curtain wall, grips display allowing you to resize and reorient the curtain wall in real-time.

To draw a curtain wall

- 1 Do one of the following to choose Curtain Wall ():
 - On the ribbon, choose AEC > Walls > Curtain Wall (in Draw) or AEC > Windows > Curtain Wall (in Draw).
 - On the menu, choose AEC > Walls > Curtain Wall or AEC > Windows > Curtain Wall.
 - On the Walls toolbar, click a wall style.
 - Type *curtainwall* and then press Enter.
- 2 If desired, choose Height to enter the height of the curtain wall.
- 3 If desired, choose Style and enter the name of the curtain wall style. To view the list of all styles, choose ?.
- 4 Specify the start point of the curtain wall.
- 5 Specify the endpoint of each segment.

- 6 If you want to add an arc segment, do the following:
 - Choose Arc.
 - Specify the midpoint of the arc segment.
 - Specify the endpoint of the arc segment.
 - Choose Line to continue adding line segments or choose Close to add the last endpoint at the start point.
- 7 After specifying the last endpoint, press Enter.


To convert two-dimensional entities to a curtain wall

- 1 Do one of the following to choose Curtain Wall (- On the ribbon, choose AEC > Walls > Curtain Wall (in Draw) or AEC > Windows > Curtain Wall (in Draw).
- On the menu, choose AEC > Walls > Curtain Wall or AEC > Windows > Curtain Wall.
- On the Walls toolbar, click a wall style.
- Type *curtainwall* and then press Enter.
- 2 Choose Convert.
- 3 Select lines, arcs, circles, or polylines to convert to curtain walls, then press Enter.
- 4 Choose Yes to delete the original two-dimensional entities, or choose No to retain the original two-dimensional entities.

Drawing steel

Steel entities can be drawn in many different shapes.

To draw steel

- 1 Do one of the following to choose Steel (- On the ribbon, choose AEC > Steel and then the steel style (in Draw).
- On the menu, choose AEC > Steel and then the steel shape.
- On the Steel toolbar, click a steel shape.
- Type *steel* and then press Enter.
- 2 Select the steel dimensions from the list.
- 3 Click OK.
- 4 To mirror the shape about its x-axis, choose XFlip. To mirror the shape about its y-axis, choose YFlip.
- 5 Specify the insertion point.
- 6 Specify the angle of rotation.
- 7 Specify the length. If the length is 0, a two-dimensional steel shape is created.

Drawing a stud, joist, or track

Studs, joists, and track can be drawn in many different shapes and styles.

To draw studs, joists, or track

- 1 Do one of the following to choose Studs, Joists, Track:
 - On the ribbon, choose AEC > Steel and then Stud, S-Stud, Joist, or Track (in Draw).
 - On the menu, choose AEC > Steel and then Stud, S-Stud, Joist, or Track.
 - On the Steel toolbar, click Stud, S-Stud, Joist, or Track.
 - Type *sjt* and then press Enter.
- 2 Select the desired dimensions of the shape from the list.
- 3 Click OK.
- 4 To mirror the shape about its x-axis, choose XFlip. To mirror the shape about its y-axis, choose YFlip.
- 5 Specify the insertion point.
- 6 Specify the angle of rotation.
- 7 Specify the length. If the length is 0, a two-dimensional shape is created.

Drawing a beam

Beams are architectural elements placed between columns, walls and other beams.

After you draw a beam, you can modify it as you would other entities using modify entity commands. For example, beams can be easily moved and resized relative to a wall. You can also use the Properties pane to change the thickness, style, etc. And when you select a beam, grips display allowing you to resize and reorient the beam in real-time.

To draw a beam

- 1 Type *beam* and then press Enter.
- 2 Specify the start point.
- 3 Specify the end point.
- 4 After specifying the last endpoint, press Enter.

You can also set custom options for beams.

At the prompt, choose Style to specify a beam style, Start Offset or End Offset to enter the distance that you want the beam segment to offset from the start or end point, Justify to specify the beam position relative to the baseline, or Roll to rotate the beam segment around the baseline.


To create a beam using an existing entity as the beam path

- 1 Type *beam* and then press Enter.
- 2 Choose Convert.
- 3 Select the lines, arcs, or polylines to convert to beam paths, then press Enter.
- 4 If you selected a polyline to convert, choose Yes to create a separate beam for each polyline segment, or choose No to convert the selected polyline to a single beam.
- 5 Choose Yes to delete the selected entities or No to retain the selected entities.

Drawing wood

Wood details can be drawn as lumber and TJI joists shapes in many different standard sizes.

To draw wood


- 1 Do one of the following to choose Wood (- On the ribbon, choose AEC > Wood and then Lumber or TJI Joists (in Draw).
- On the menu, choose AEC > Wood and then Lumber or TJI Joists.
- Type *wood* and then press Enter.
- 2 Select the desired dimensions from the list.
- 3 Click OK.
- 4 To mirror the shape about its x-axis, choose XFlip. To mirror the shape about its y-axis, choose YFlip.
- 5 Specify the insertion point.
- 6 Specify the angle of rotation.
- 7 Specify the length. If the length is 0, a two-dimensional shape is created.

Drawing a slab

Slabs are foundations that are easy to draw at the beginning of an architectural project or at any time later relative to walls.

After you draw a slab, you can modify it as you would other entities using modify entity commands. For example, slabs can be easily moved and resized relative to a wall. You can also use the Properties pane to change the thickness, style, etc. And when you select a slab, grips display allowing you to resize and reorient the slab in real-time.

To draw a slab


- 1 Do one of the following to choose Slab (- On the ribbon, choose AEC > Slab (in Draw).
- On the menu, choose AEC > Slab.
- Type *slab* and then press Enter.
- 2 Specify the start point of the slab.
- 3 Specify the endpoint of each segment.
- 4 After specifying the last endpoint, press Enter.

Drawing a roof

Architectural roofs can have a slope, gables, hips, overhangs, etc. As you draw the roof, PTCAD automatically calculates the dimensions, slopes, etc. according to the segment points you select.

After you draw a roof, you can modify it as you would other entities using modify entity commands. For example, roofs can be easily moved, resized, scaled, etc. You can also use the Properties pane to change the thickness, slope, overhang, etc. And when you select a roof, grips display allowing you to change its shape from gable to hip and resize and reorient the roof in real-time.

To draw a roof


- 1 Do one of the following to choose Roof ():
 - On the ribbon, choose AEC > Roof (in Draw).
 - On the menu, choose AEC > Roof.
 - Type *roof* and then press Enter.
- 2 Specify the start point of the roof.
- 3 Specify the endpoint of each segment.
- 4 After specifying the last endpoint, press Enter.

Drawing a roof slab

A roof slab is a flat roof that is similar to a slab except it is drawn above walls instead of below.

After you draw a roof slab, you can modify it as you would other entities using modify entity commands. For example, roof slabs can be easily moved and resized relative to a wall. You can also use the Properties pane to change the thickness, style, etc. And when you select a roof slab, grips display allowing you to resize and reorient the roof slab in real-time.


To draw a roof slab

- 1 Do one of the following to choose Roof Slab ():
 - On the ribbon, choose AEC > Roof Slab (in Draw).
 - On the menu, choose AEC > Roof Slab.
 - Type *roofslab* and then press Enter.
- 2 Specify the start point of the roof slab.
- 3 Specify the endpoint of each segment.
- 4 After specifying the last endpoint, press Enter.

Converting 2D entities to walls and slabs

You can convert several 2D entity types to walls or slabs: lines, circles, arcs, and polylines.

To convert entities to a wall

- 1 Do one of the following to choose Convert to Wall (

To convert entities to a slab


- 1 Do one of the following to choose Convert to Slab (

Drawing a section line

A section line can be drawn in any three-dimensional drawing to determine a two-dimensional projection of the area you want to view. The section line is a polyline that can consist of continuous segments that define the area you want to view.

After you draw a section line in your drawing, you can modify its segments and length, then update the two-dimensional projection view.

To draw a selection line

- 1 Do one of the following to choose Section Line ():
 - On the ribbon, choose AEC > Section Line (in Sections).
 - On the menu, choose AEC > Section Line.
 - Type *sectionline* and then press Enter.
- 2 Specify the start point of the section line.
- 3 Specify the end point of each segment.
- 4 Specify the end point of the section line, then press Enter.
- 5 To turn on the visibility of hidden lines, choose No. To keep hidden lines invisible, choose Yes.
- 6 Specify the insertion point of the 2D projection.

To add a new 2D projection from an existing section line

- 1 Type *sectionlinegenerate* and then press Enter.
- 2 Select the section line, then press Enter.
- 3 Specify an insertion point for the new 2D projection view.

Use a shortcut.

Right-click the section line you want to generate a new projection for, then choose Generate 2D Section.

To modify the position of an existing section line

- 1 Select the section line.
- 2 Move its start point, segments, end point, or length.

If Use Model Extents for Height is set to Yes in the Properties pane, the section line height and lower extension are matched automatically. If set to No, you can change the height and lower extension separately.
- 3 If automatic updating is turned off, right-click the 2D projection, then choose Refresh.

To modify the properties of a section line symbol

- 1 Select the symbol of a section line.
- 2 In the Properties pane, select new properties, including the label text. Note that the symbol is annotative and can be scaled.

Use the AECAUTOUPDATE system variable to control automatic updating.


If AECAUTOUPDATE is set to 1, two-dimensional projection views update automatically after a section line or elevation line changes. If set to 0, you need to manually refresh projection views after changes.

Drawing an elevation line

An elevation line can be drawn in any three-dimensional drawing to create a two-dimensional projection view of the drawing.

You can modify existing elevation lines and therefore their related projection views. You can also modify the elevation symbol that marks the elevation line.

To draw an elevation line

- 1 Do one of the following to choose Elevation Line ():
 - On the ribbon, choose AEC > Elevation Line (in Sections).
 - On the menu, choose AEC > Elevation Line.
 - Type *elevationline* and then press Enter.
- 2 Specify the start point of the elevation line.
- 3 Specify the end point of the elevation line.

If you want to include all entities in the 2D projection view, make sure the start and end points are beyond the start and end of the model.
- 4 To turn on the visibility of hidden lines, choose No. To keep hidden lines invisible, choose Yes.
- 5 Specify the insertion point of the 2D projection view.

To add a new 2D projection from an existing elevation line

- 1 Type *elevationlinegenerate* and then press Enter.
- 2 Select the elevation line, then press Enter.
- 3 Specify an insertion point for the new 2D projection view.

Use a shortcut.

Right-click the elevation line you want to generate a new projection for, then choose Generate 2D Elevation.

To reverse the direction of an existing elevation line

- 1 Type *elevationlinereverse* and then press Enter.
- 2 Select the elevation line, then press Enter.
- 3 If automatic updating is turned off, right-click the 2D projection view, then choose Refresh.

Use a shortcut.

Right-click the elevation line you want to reverse, then choose Reverse.

To modify the position of an existing elevation line

- 1 Select the elevation line.
- 2 Move its start point, end point, or length.
If Use Model Extents for Height is set to Yes in the Properties pane, the elevation line height and lower extension are matched automatically. If set to No, you can change the height and lower extension separately.
- 3 If automatic updating is turned off, right-click the 2D projection view, then choose Refresh.

To modify the properties of an elevation line symbol

- 1 Select the symbol of an elevation line.
- 2 In the Properties pane, select new properties, including the label text. Note that the symbol is annotative and can be scaled.

Use the AECAUTOUPDATE system variable to control automatic updating.

If AECAUTOUPDATE is set to 1, two-dimensional projection views update automatically after a section line or elevation line changes. If set to 0, you need to manually refresh projection views after changes.



Drawing tables

Tables contain data in columns and rows. When you draw a table, you assign it a table style which affects its layout, formatting, and color, and specify whether the table content is empty initially or filled automatically with data from a spreadsheet. After you draw a table—empty or filled with linked data, you can add text and modify various cell and table settings.

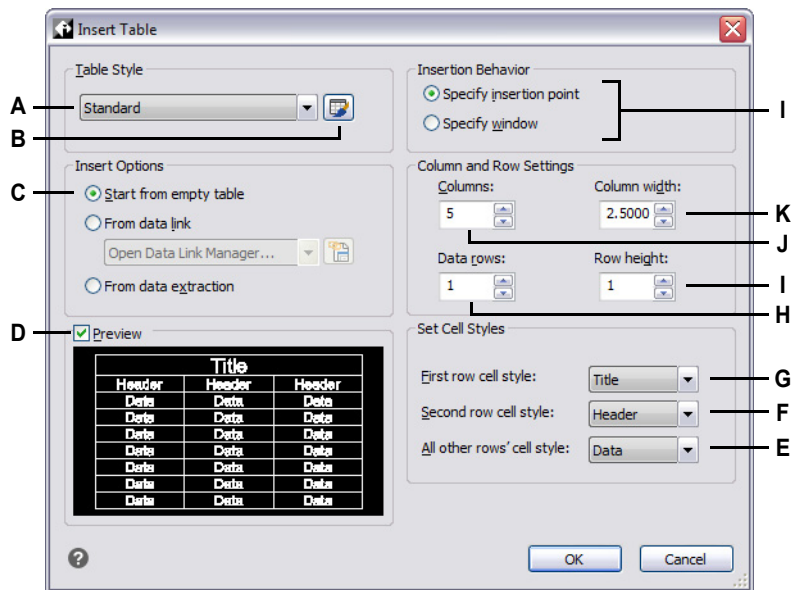
Drawing an empty table

When you draw a table, you assign a table style, determine the number of columns and rows, and other options. Tables are inserted in the drawing by either specifying the upper-left corner of the table or by specifying a window that specifies its total width and height.

To draw an empty table

- 1 Do one of the following to choose Table (- On the ribbon, choose Home > Table (in Annotation) or Annotate > Table (in Tables).
- On the menu, choose Draw > Table.
- Type *table* and then press Enter.
- 2 In Table Style, choose a table style for the table. If necessary, click () to create or modify table styles.
- 3 In Insert Options, select Start from Empty Table.
- 4 Determine how you want to insert the table in the drawing by choosing one of the following:
 - **Specify insertion point** Specify the location of the upper-left corner of the table when you insert it. You can specify the insertion point directly in the drawing or enter coordinate values in the command bar. If the table reads from bottom to top (as specified for the table direction in the table style), the insertion point specifies the location of the lower-left corner of the table.
 - **Specify window** Specify a location and size for the table when you insert it. Specify two points of the window directly in the drawing or specify their coordinates in the command bar.
- 5 Specify column settings:
 - **Columns** Enter the number of columns. If Specify Window is also selected, the column width is determined automatically by the width of the table.
 - **Column Width** Enter the width of the columns. If Specify Window is also selected, the number of columns is determined automatically by the width of the table.

- 6 Specify data row settings:
 - **Data Rows** Enter the number of data rows. If Specify Window is also selected, the row height is determined automatically by the height of the table.
 - **Row Height** Enter the height of data rows. If Specify Window is also selected, the number of data rows is determined automatically by the height of the table.
- 7 Specify cell settings:
 - **First row cell style** Choose the cell style for the first row of the table.
 - **Second row cell style** Choose the cell style for the second row of the table.
 - **All other rows' cell style** Choose the cell style for all other rows.
- 8 When finished, click OK.
- 9 In the drawing, follow the prompts to insert the table.



- A** Select a table style for the table.
- B** Click to create or modify table styles.
- C** Select to create an empty table.
- D** Click to turn the preview display on or off.
- E** Choose the cell style for all rows other than the first and second rows.
- F** Choose the cell style for the second row of the table.
- G** Choose the cell style for the first row of the table.
- H** Specify the number of data rows. If Specify Window is also selected, the row height is determined automatically.
- I** Specify the height of rows. If Specify Window is also selected, the number of data rows is determined automatically.
- J** Specify the number of columns. If Specify Window is also selected, the column width is determined automatically.
- K** Specify the width of columns. If Specify Window is also selected, the number of columns is determined automatically.
- L** Choose whether to insert the table by specifying its upper-left corner as the insertion point or by specifying a window for its overall size.




Drawing a table using content from a spreadsheet

Tables can contain content from a linked .xls or .xlsx file, allowing you to easily share and update data between CAD drawings and external spreadsheets.

When you insert the table, specify the data link that references the content you want to include. The number and size of rows and columns of the new table are determined automatically based on the content in the spreadsheet.

For details about data links, see “Working with data links” on page 591.

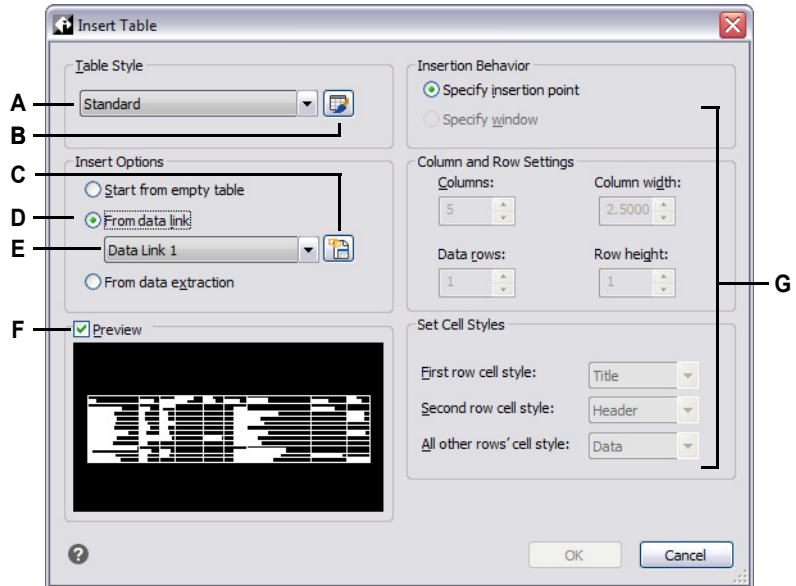
To draw a table that contains content from a spreadsheet

- 1 Do one of the following to choose Table (- On the ribbon, choose Home > Table (in Annotation) or Annotate > Table (in Tables).
- On the menu, choose Draw > Table.
- Type *table* and then press Enter.
- 2 In Table Style, choose a table style for the table. If necessary, click () to create or modify table styles.
- 3 In Insert Options, select From Data Link.
- 4 In the data link list, choose the data link that contains the content you want to include in the table. If necessary, select Open Data Link Manager () to create a data link or modify an existing one.

Data links are references to external files.

If you share your drawing with other users, make sure you include any referenced .xls/.xlsx files.

- 5 Click OK.
- 6 In the drawing, follow the prompts to insert the table.



- A** Select a table style for the table.
- B** Click to create or modify table styles.
- C** Click to create or modify data links.
- D** Select to use content from an external file for the new table.
- E** Select the data link that contains the content you want to include.
- F** Click to turn the preview display on or off.
- G** Disabled because this information is supplied by the linked spreadsheet.


Drawing a table using content from report data

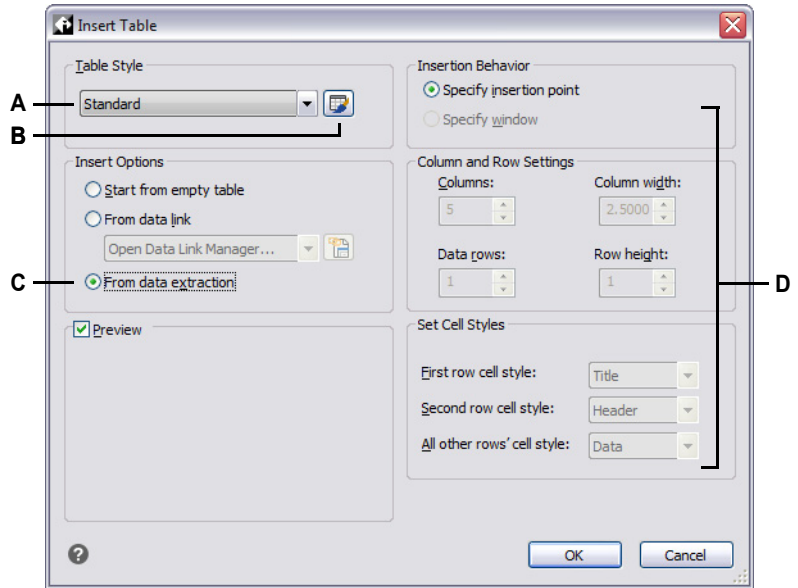
Tables can be created with content from linked report data. When you insert the table, you specify the options for extracting data from entities located in one or more drawings. The number and size of rows and columns of the new table are determined automatically based on the report data.

Later, the table contents can be updated automatically to include the latest report data.

For more details about extracting data and creating reports, see “Reporting information about entities” on page 383.

To draw a table that contains content from report data

- 1 Do one of the following to choose Table () to create or modify table styles.
- 3 In Insert Options, select From Data Extraction.
- 4 Click OK.
- 5 Make your selections for extracting data to include in the table. Click [?] for details about each option.



- A** Select a table style for the table.
B Click to create or modify table styles.

- C** Select to create table content with data extracted from one or more drawings.
D Disabled because this information is supplied by the linked spreadsheet.

Editing table text

You can edit table text just as you would edit multiline text in the drawing.

To edit table text













- 1 Double-click a table cell directly in the drawing, or do one of the following to choose Edit Table (

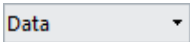
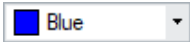






Modifying a table

Use the Properties pane to change a table's style, direction, table breaks, and more. Tables can also be modified by inserting and deleting rows and columns, merging cells, locking cells, and specifying the data format.

To modify a table

- 1 Select a table.
- 2 Use the Properties pane to change settings for the table.
- 3 To modify table cells, do the following:
 - Select the table cells you want to change. To select multiple cells, press Shift while clicking the first and last cell in the range. To select an entire row or column, click the row or column name.
 - Right-click the table cells or use the Table Cell tools to make your selections for highlighted cells.

Tool	Name	Description
	Insert Above	Click to insert a row above the current row.
	Insert Below	Click to insert a row below the current row.
	Delete Row(s)	Click to delete the current row.
	Insert Left	Click to insert a column to the left of the current column.
	Insert Right	Click to insert a column to the right of the current column.
	Delete Column(s)	Click to delete the current column.
	Merge Cells	Click to combine selected cells together. Choose Merge All, Merge by Row, or Merge by Column. (Available only if more than one cell is selected.)
	Unmerge Cells	Click to separate selected cells. (Available only if selected cells are merged.)
	Match Cell	Click to copy the properties of a source table cell and apply them to selected cells.
	Remove Overrides	Click to undo all the changes made to the selected table cell or range of cells.
	Justification	Select the justification for selected cells.
	Edit Borders	Click to modify border properties of the selected cell or range of cells. For more details, click Help in the dialog box that displays.

Tool	Name	Description
	Cell Style	Select a style to apply to selected cells.
	Fill Color	Select a color to fill selected cells.
	Cell Locking	Click to select lock options for selected cells: Unlocked, Content Locked, Format Locked, Content and Format Locked.
	Data Format	Click to select the format for the data in selected cells: Angle, Currency, Date, Decimal Number, General, Percentage, Point, Text, Whole Number. (Not available if the format is locked.)
	Block	Click to insert a block into the selected cell or range of cells. For more details, click Help in the dialog box that displays.
	Field	Click to insert a field in a selected cell. For more details, click Help in the dialog box that displays.
	Formula	Click to insert a Sum, Average, Count, Cell, or Equation formula, and then follow the prompts.
	Manage Cell Contents	Click to specify how text and blocks are positioned relative to each other within a selected cell. For more details, click Help in the dialog box that displays. Available only when the selected table cell contains multiple text entities and/or blocks.




Working with table styles

When you draw a table in a drawing, the table uses the current table style by default. A table style determines the table direction, various cell styles, formatting, and other characteristics.



Every table has a default table style, named Standard. You cannot delete the Standard style, but you can rename it or modify it. You also can create and use an unlimited number of additional table styles.

Table styles can also be managed using the Explorer. For more details, see “Working with table styles” on page 324.

To create a table style

- 1 Do one of the following to choose Table Styles Manager (- On the ribbon, choose Annotate > Table Styles Manager (in Tables).
- On the menu, choose Format > Table Styles Manager.
- Type *tablestyle* and then press Enter.
- 2 Click New.
- 3 In New Style Name, type the name of the new table style.
- 4 In Start With, select an existing table style on which to base the new table style.
- 5 Click Continue.
- 6 To use an existing table as a base for the new table style, click () and select a table in the drawing.
- 7 In Table Direction, choose Down to create a table that reads from top to bottom, or choose Up to create a table that reads from bottom to top.
- 8 In Cell Styles, do the following:
 - Select Data from the list, then select options for data cells.
 - Select Header from the list, then select options for header cells.
 - Select Title from the list, then select options for title cells.
 Click [?] in the dialog box for more details about each option.
- 9 If you need to create a new cell style, click () enter a name for the cell style, then click Continue. When you return to the New Table Style dialog box, make your selections.

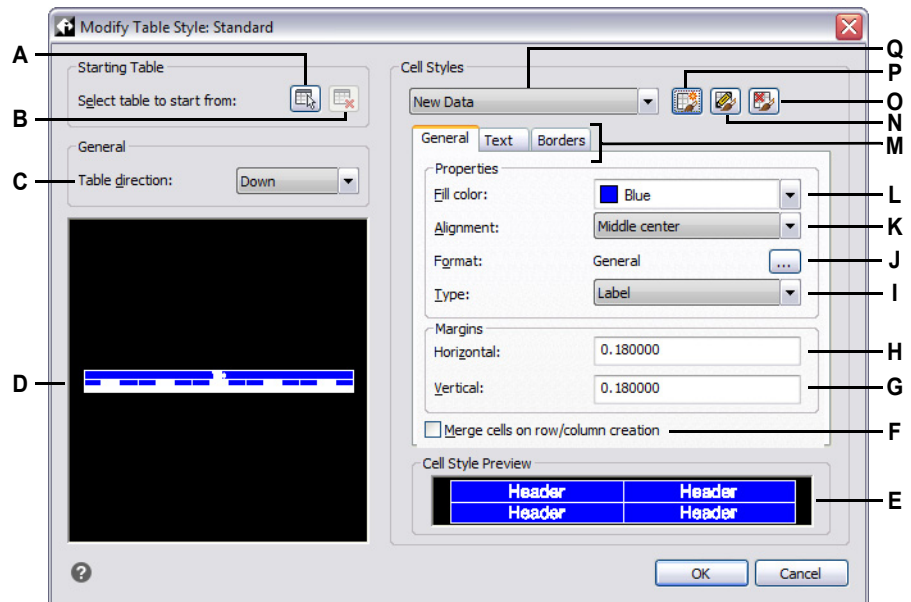
Cell styles can be renamed and deleted.

In the list of cell styles, select the one you want to rename, click () enter a new name, then press Enter. To delete a cell style, select it from the list, then click () Data, Header, and Title cell styles cannot be renamed or deleted.

- 10 When finished, click OK.

To modify a table style

- Do one of the following to choose Table Styles Manager (📊):
 - On the ribbon, choose Annotate > Table Styles Manager (in Tables).
 - On the menu, choose Format > Table Styles Manager.
 - Type *tablestyle* and then press Enter.
- Select the table style you want to modify, then click Modify.
- In the Modify Table Style dialog box, make your selections for the table style as described previously for creating a new table style.
- When finished, click OK.



- | | |
|---|--|
| <p>A Click to select a table in the drawing to use as a base for the table style.</p> <p>B Click to remove a start table from the table style.</p> <p>C Choose Down to create a table that reads from top to bottom. Choose Up to create a table that reads from bottom to top.</p> <p>D Displays a preview of a table with the style.</p> <p>E Displays a preview of the table cell with current settings.</p> <p>F Mark to combine new rows or columns created with the current cell style into one cell.</p> <p>G Specify the spacing between the top and bottom cell borders and the contents of the cell.</p> | <p>H Specify the spacing between the left and right cell borders and the contents of the cell.</p> <p>I Choose whether the cell style is for data or a label.</p> <p>J Click [...] to set the data format of the cell style.</p> <p>K Choose how you want the text to be justified in the cell.</p> <p>L Select the background color of the cell.</p> <p>M Click which options you want to change: general, text, or border options.</p> <p>N Click to rename the selected cell style.</p> <p>O Click to delete the selected cell style.</p> <p>P Click to create a new cell style.</p> <p>Q Select the cell style you want to modify.</p> |
|---|--|

To assign a table style to an existing table

- 1 Select a table in the drawing.
- 2 On the Properties pane in Table Style, change the table style.

Drawing wipeouts

Wipeouts are unique entities that can help you hide areas of your drawing. They display with the current background color, so the details behind the wipeout do not display or print.

Wipeouts are similar to other entities — you can copy, mirror, array, erase, rotate, and scale them, and they can be used in both model space and paper space.

If you want to print the wipeout entities located in a drawing, you must print to a raster-capable printer. Note that in some cases you may have unexpected results when printing drawings that contain wipeout entities, for example, if printing on colored paper.


Wipeouts are created using existing polygons, closed zero-width polylines made up of only line segments, or new polylines that you draw while using the Wipeout command.

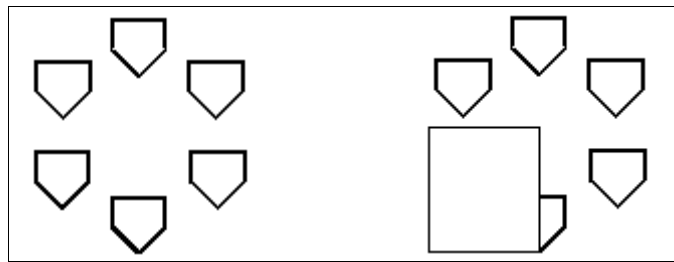
The display of wipeouts varies depending on your version of PTCAD.

If your PTCAD version does not include raster image capability, wipeout entities display, but the details behind the wipeouts also display.

Drawing a wipeout

To draw a wipeout

- 1 Do one of the following to choose Wipeout :
 - On the ribbon, choose Home > Wipeout (in Annotation) or Annotate > Wipeout (in Markup).
 - On the menu, choose Draw > Wipeout.
 - On the Draw toolbar, click the Wipeout tool.
 - Type *wipeout* and then press Enter.
- 2 Specify the start point.
- 3 Specify the endpoint of each segment.
- 4 After specifying the last endpoint, choose Done or press Enter.




Existing drawing.

Drawing with wipeout.

Creating a wipeout using existing polygons and polylines


To create a wipeout using an existing polygon or polyline

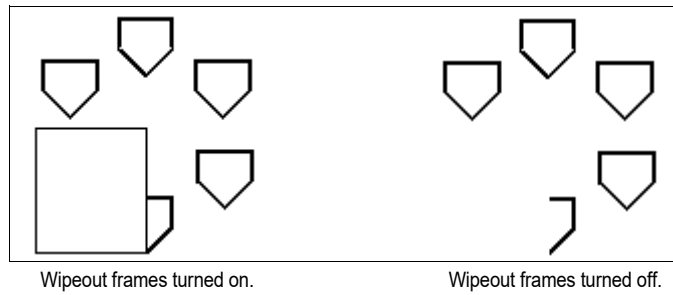
- 1 Do one of the following to choose Wipeout ():
 - On the ribbon, choose Home > Wipeout (in Annotation) or Annotate > Wipeout (in Markup).
 - On the menu, choose Draw > Wipeout.
 - On the Draw toolbar, click the Wipeout tool.
 - Type *wipeout* and then press Enter.
- 2 Choose Polyline.
- 3 Select the closed polyline to use for the wipeout.
- 4 Choose one of the following:
 - **Yes** — Creates the wipeout and deletes the polyline used to create the wipeout.
 - **No** — Creates the wipeout and keeps the polyline used to create the wipeout.

Turning wipeout frames on or off

Each wipeout has a frame along its boundary. Wipeout frames can be turned on or off for any drawing. When wipeout frames are on, you can select and modify wipeouts. You may want to turn off wipeout frames when it's time to print.

To turn wipeout frames on or off

- 1 Do one of the following to choose Wipeout ():
 - On the ribbon, choose Home > Wipeout (in Annotation) or Annotate > Wipeout (in Markup).
 - On the menu, choose Draw > Wipeout.
 - On the Draw toolbar, click the Wipeout tool.
 - Type *wipeout* and then press Enter.
- 2 Choose Frames.
- 3 Choose On or Off.



You can also turn wipeout and image frames on or off using a single command.
Type tframes to turn all wipeout and image frames in the drawing on or off. Or use the IMAGEFRAME and WIPEOUTFRAME system variables.

Drawing revision clouds

Revision clouds are cloud shapes that mark areas of a drawing that require further attention. They are especially helpful when revising a drawing; add a revision cloud to each modified area so reviewers can find changes easily.

Revision clouds are polylines, so you can work with and modify them in the same way you would a polyline: move, copy, mirror, and scale the entire revision cloud or select and move individual vertices to adjust the arcs that make up the revision cloud.

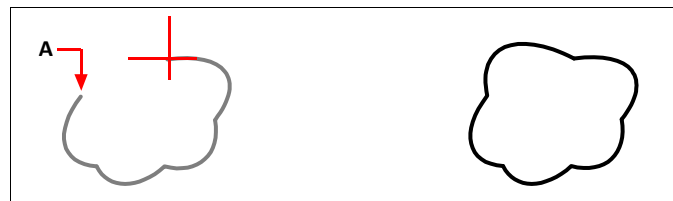
Drawing a revision cloud

Drawing a revision cloud is easy: simply select a start point and move the mouse.

To draw a revision cloud

- 1 Do one of the following to choose Revision Cloud (☁):
 - On the ribbon, choose Home > Revision Cloud (in Annotation) or Annotate > Revision Cloud (in Markup).
 - On the menu, choose Draw > Revision Cloud.
 - On the Draw toolbar, click the Revision Cloud tool.
 - Type *revcloud* and then press Enter.
- 2 Specify the start point.
- 3 Move the mouse, encircling the desired area.

When you return to the starting point, the revision cloud command finishes automatically.



Select start point (A), then move the mouse around the desired area and back towards the start point (A).

Resulting revision cloud.

Edit revision clouds just as you would a polyline.

Select the revision cloud and move its vertices or use the Edit Polyline tool (🔗) on the Modify toolbar.

Creating a revision cloud using existing entities

In addition to drawing new revision clouds, you can also convert existing entities — lines, arcs, circles, 2D polylines, and splines — into revision clouds.

To create a revision cloud using an existing entity

- 1 Do one of the following to choose Revision Cloud (☁):
 - On the ribbon, choose Home > Revision Cloud (in Annotation) or Annotate > Revision Cloud (in Markup).
 - On the menu, choose Draw > Revision Cloud.
 - On the Draw toolbar, click the Revision Cloud tool.
 - Type *revcloud* and then press Enter.
- 2 Choose Entity.
- 3 Select the existing line, arc, circle, 2D polyline, or spline to be converted.
- 4 Choose whether to reverse the direction of the individual arcs of the revision cloud. Choose No to keep the arcs pointing inward. Choose Yes to flip the arcs to point outward.

A revision cloud is created, and the selected entity remains or is deleted depending on the DELOBJ system variable setting.


Customizing default revision cloud settings

All revision clouds are drawn using default settings that can be customized:

- Minimum arc length — The individual arcs that make up a revision cloud can be made smaller or larger by setting the minimum arc length.
- Maximum arc length — The individual arcs that make up a revision cloud can be made smaller or larger by setting the maximum arc length.
- Style — Revision clouds can appear as if they had been drawn with a regular pen or a calligraphy pen.

Only new revision clouds will be drawn with the new settings. Existing revision clouds are not affected.

To customize revision cloud settings

- 1 Do one of the following to choose Revision Cloud (- On the ribbon, choose Home > Revision Cloud (in Annotation) or Annotate > Revision Cloud (in Markup).
- On the menu, choose Draw > Revision Cloud.
- On the Draw toolbar, click the Revision Cloud tool.
- Type *revcloud* and then press Enter.
- 2 Choose Arc Length.
- 3 Enter the minimum length of the individual arcs that make up the revision cloud, then press Enter.
- 4 Enter the maximum length of the individual arcs that make up the revision cloud, then press Enter. The value cannot be set to more than three times the minimum arc length.

Arc lengths can also be scaled.

The setting of the DIMSCALE system variable also affects arc length.

- 5 Choose Style and select one of the following:
 - **Normal** — Draws new revision clouds as if they were drawn with a regular pen.
 - **Calligraphy** — Draws new revision clouds as if they were drawn with a calligraphy pen.

Adding multileaders


Multileaders consist of line segments or a spline curve that connects a feature in a drawing to multiline text or a block. Generally, you place an arrowhead at the first point, and the multiline text or block is placed immediately adjacent to the last point. Landing lines can also be placed horizontally between the leader and content (the multiline text or block).

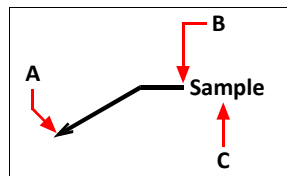
Creating multileaders

By default when you create a multileader, the multileader includes an arrow, line segments, and multiline text at the endpoint. However, you can choose a spline instead of line segments and a block (or nothing) instead of multiline text.

There are more options too. Use the Multileader prompts once you start the Multileader command. For more control over format, use multileader styles. For more details, see “Working with multileader styles” on page 198 in this chapter.


To create a multileader that includes multiline text

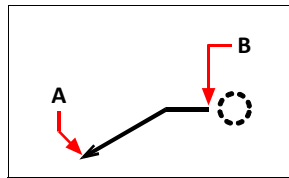
- 1 Do one of the following to choose Multileader ():
 - On the ribbon, choose Home > Multileader (in Annotation) or Annotate > Multileader (in Leaders).
 - On the menu, choose Dimensions > Multileader.
 - On the Multileader toolbar, click the Multileader tool. Or on the Dimensioning toolbar, click the Multileader tool.
 - Type *mleader* and then press Enter.
- 2 Specify the starting point of the multileader.
- 3 Specify the endpoint of the multileader line segment.
- 4 Specify additional multileader line segment endpoints, if the maximum number of points is set to greater than two.
- 5 After you specify the last endpoint, press Enter.
- 6 In the Multiline Text editor, type the annotation.
- 7 When finished, click outside the Multiline Text editor.



Specify the starting point of the multileader (A) and the endpoint of the multileader (B). Then type multiline text (C).


To create a multileader that includes a block

- 1 Do one of the following to choose Multileader ():
 - On the ribbon, choose Home > Multileader (in Annotation) or Annotate > Multileader (in Leaders).
 - On the menu, choose Dimensions > Multileader.
 - On the Multileader toolbar, click the Multileader tool. Or on the Dimensioning toolbar, click the Multileader tool.
 - Type *mleader* and then press Enter.
- 2 Choose Options, choose Content Type, then choose Block.
- 3 Type the name of the block to include at the end of the multileader, then press Enter.
- 4 Choose Exit.
- 5 Specify the starting point of the multileader.
- 6 Specify the endpoint of the multileader line segment.
- 7 Specify additional multileader line segment endpoints, if the maximum number of points is set to greater than two.
- 8 After you specify the last endpoint, press Enter.

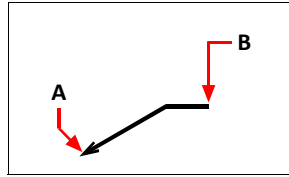


Specify the starting point of the multileader (A) and the endpoint of the multileader (B). The specified block is drawn automatically.

To create a multileader that does not include multiline text or a block


- 1 Do one of the following to choose Multileader ():
 - On the ribbon, choose Home > Multileader (in Annotation) or Annotate > Multileader (in Leaders).
 - On the menu, choose Dimensions > Multileader.
 - On the Multileader toolbar, click the Multileader tool. Or on the Dimensioning toolbar, click the Multileader tool.
 - Type *mleader* and then press Enter.
- 2 Choose Options, choose Content Type, then choose None.
- 3 Choose Exit.
- 4 Specify the starting point of the multileader.
- 5 Specify the endpoint of the multileader line segment.

- 6 Specify additional multileader line segment endpoints, if the maximum number of points is set to greater than two.
- 7 After you specify the last endpoint, press Enter.



Specify the starting point of the multileader (A) and the endpoint of the multileader (B).

To specify options for a multileader before creating it

- 1 Do one of the following to choose Multileader ():
 - On the ribbon, choose Home > Multileader (in Annotation) or Annotate > Multileader (in Leaders).
 - On the menu, choose Dimensions > Multileader.
 - On the Multileader toolbar, click the Multileader tool. Or on the Dimensioning toolbar, click the Multileader tool.
 - Type *mleader* and then press Enter.
- 2 Choose Options, then choose from the following:
 - **Leader type** Select whether the leader line type is straight, a spline, or has no leader line.
 - **Leader landing** Select whether to place a landing line horizontally between the leader and text or block content.
 - **Content type** Select whether a multileader with this style contains multiline text, a block, or neither.
 - **Maxpoints** Select to enter the number of maximum points for the leader line.
 - **First angle** Select to specify the constraint angle of the first line segment.
 - **Second angle** Select to specify the constraint angle of the second line segment.
- 3 When finished, choose Exit, then continue creating the multileader.


You can specify more multileader options using multileader styles.

For more details, see “Working with multileader styles” on page 198 in this chapter.


Adding and removing leader lines from multileaders

Each multileader can have more than one leader line attached to it.

To add leader lines to a multileader

- 1 Do one of the following to choose Add Leader Lines (- On the ribbon, choose Home > Add Leader Lines (in Annotation) or Annotate > Add Leader Lines (in Leaders).
- On the Multileader toolbar, click the Add Leader Lines tool.
- Type *aimmleadeditadd* and then press Enter.
- 2 Select the desired multileader.
- 3 Select an arrowhead location for a new leader.
- 4 Continue selecting points to add additional leader lines, then press Enter when finished.


To remove leader lines from a multileader

- 1 Do one of the following to choose Remove Leader Lines (- On the ribbon, choose Home > Remove Leader Lines (in Annotation) or Annotate > Remove Leader Lines (in Leaders).
- On the Multileader toolbar, click the Remove Leader Lines tool.
- Type *aimmleadeditremove* and then press Enter.
- 2 Select the entities to include in the selection set for leader removal.
- 3 Select a leader line to remove.
- 4 Continue selecting leader lines to remove, then press Enter when finished.

Aligning multileaders

It's easy to align and distribute multileaders. You can also collect multileaders and place them in one location. Note that multileaders with block content can be collected, not multileaders with text content.


To align multileaders

- 1 Do one of the following to choose Align Multileader (- On the ribbon, choose Home > Align Multileader (in Annotation) or Annotate > Align Multileader (in Leaders).
- On the Multileader toolbar, click the Align Multileader tool.
- Type *mleaderalign* and then press Enter.
- 2 Select the multileaders to align, then press Enter.
- 3 Choose options if you want to customize the alignment, then choose from the following:
 - **Distribute** Places multileader block or text content between two points that you select.
 - **Make Leader Segments Parallel** Places multileader block or text content where the last leader line segments will be parallel.
 - **Specify Spacing** Places multileader block or text content according to the spacing that you specify.
 - **Use Current Spacing** Places multileader block or text content according to the current spacing that was previously specified.

The following prompts display for the Distribute option, which is the default.

- 4 Select the start point to use for distributing the multileader block or text content.
- 5 Select the end point to use for distributing the multileader block or text content.
Content will be aligned and distributed between the two points.

To collect multileaders that include blocks into a row or column

- 1 Do one of the following to choose Collect Multileader (- On the ribbon, choose Home > Align Multileader (in Annotation) or Annotate > Collect Multileader (in Leaders).
- On the Multileader toolbar, click the Collect Multileader tool.
- Type *mleadercollect* and then press Enter.
- 2 Select the multileaders to collect, then press Enter.
- 3 Choose how you want to collect and align multileader content:
 - **Horizontal** Places collected multileaders horizontally.
 - **Vertical** Places collected multileaders vertically.
 - **Wrap** Wraps the collected multileaders according to a width and number per row that you specify.
- 4 Specify the upper left corner of the new location for the selected multileaders.

Working with multileader styles

When you create a multileader, the program draws it on the current layer, using the current multileader style. Each multileader has a corresponding multileader style, which controls the appearance of the multileader, such as the arrowhead type, text style, colors, and more.


You can create, save, modify, and delete named multileader styles. You can also set any multileader style to be the current one that is assigned to new multileaders as you add them to your drawing.

Multiline styles can also be managed using the Explorer. For more details, see “Working with multileader styles” on page 330.

Creating a multileader style

Multileader styles provide a way for you to change various settings that control the appearance of multileaders. You can then save those settings for reuse. If you don't define a multileader style before creating multileaders, the program uses the Standard multileader style, which stores the default multileader settings.

To create a multileader style


- 1 Do one of the following to choose Multileader Styles Manager ():
 - On the ribbon, choose Home > Multileader Styles Manager (in Annotation) or Annotate > Multileader Styles Manager (in Leaders).
 - On the menu, choose Format > Multileader Styles Manager.
 - On the Multileader or Styles toolbar, click the Multileader Styles Manager tool. Or on the Format toolbar, click the Multileader tool.
 - Type *mleaderstyle* and then press Enter.
- 2 Click New.
- 3 In New Style Name, type the name of the new multileader style.
- 4 In Start With, select an existing multileader style on which to base the new multileader style.
- 5 In Annotative, select to make multileaders with this style affected by annotation scaling.
- 6 Click Continue.
- 7 In the Create New Multileader Styles dialog box, make your selections for the multileader style as necessary. For details about the available options, click [?] in the dialog box.
- 8 When finished, click OK.

Setting the current multileader style

When you create a multileader, it is automatically assigned the multileader style that is set to current. By default, the program uses the Standard multileader style, which stores the default multileader settings. Or you can choose a different multileader style to be the current one.

If you choose a different multileader style as current, that style is assigned to multileaders that you create in the future, not existing multileaders.

To set a multileader style as current

- 1 Do one of the following to choose Multileader Styles Manager ():
 - On the ribbon, choose Home > Multileader Styles Manager (in Annotation) or Annotate > Multileader Styles Manager (in Leaders).
 - On the menu, choose Format > Multileader Styles Manager.
 - On the Multileader or Styles toolbar, click the Multileader Styles Manager tool. Or on the Format toolbar, click the Multileader tool.
 - Type *mleaderstyle* and then press Enter.
- 2 In the Multileader Styles Manager dialog box, select a multileader style from the list.
- 3 Click Set Current.
- 4 Click Close.


Assigning a style to a multileader

You can assign a different multileader style to a multileader at any time.


To assign a style to a multileader

- 1 Select a multileader in the drawing.
- 2 Do one of the following:
 - On the ribbon, choose Home then in Annotation, select the style in the Multileader Style list. Or choose Annotate then select the style (in Leaders).
 - On the Multileader or Styles toolbar, select the style in the Multileader Style list.
 - Right-click the multileader, choose Properties, then in Style, select the style.

*Renaming a multileader style***To rename a multileader style**

- 1 Do one of the following to choose Multileader Styles Manager (- On the ribbon, choose Home > Multileader Styles Manager (in Annotation) or Annotate > Multileader Styles Manager (in Leaders).
- On the menu, choose Format > Multileader Styles Manager.
- On the Multileader or Styles toolbar, click the Multileader Styles Manager tool. Or on the Format toolbar, click the Multileader tool.
- Type *mleaderstyle* and then press Enter.
- 2 In the Multileader Styles Manager dialog box, select the multileader style to rename.
- 3 Click the name a second time, type the new name, then press Enter.
- 4 Click Close.


*Deleting a multileader style***To delete a named multileader style**

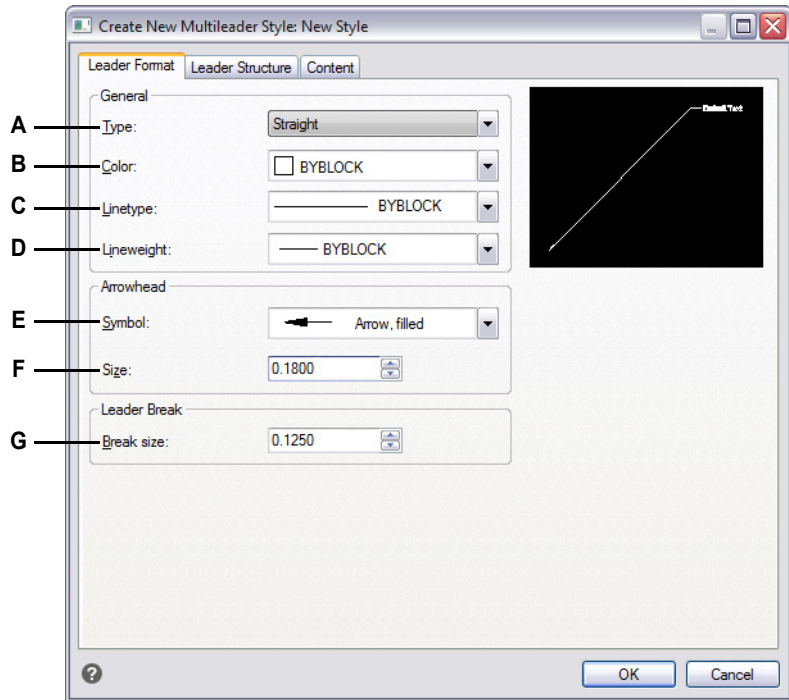
- 1 Do one of the following to choose Multileader Styles Manager (- On the ribbon, choose Home > Multileader Styles Manager (in Annotation) or Annotate > Multileader Styles Manager (in Leaders).
- On the menu, choose Format > Multileader Styles Manager.
- On the Multileader or Styles toolbar, click the Multileader Styles Manager tool. Or on the Format toolbar, click the Multileader tool.
- Type *mleaderstyle* and then press Enter.
- 2 In the Multileader Styles Manager dialog box, select the multileader style to delete.
- 3 Click Delete.
- 4 To confirm the deletion, click OK.
- 5 Click Close.

Controlling leader format settings for multileader styles

You can control leader format settings that affect the type, color, linetype, lineweight, arrowhead, and break size. Any changes you make affect the multileader style you're working with (a new or existing one). The image tile on the right side of the Create New/Modify Multileader Styles dialog box shows the appearance of a multileader based on the current multileader style settings.

To set the leader format for multileader styles

- 1 Do one of the following to choose Multileader Styles Manager ():
 - On the ribbon, choose Home > Multileader Styles Manager (in Annotation) or Annotate > Multileader Styles Manager (in Leaders).
 - On the menu, choose Format > Multileader Styles Manager.
 - On the Multileader or Styles toolbar, click the Multileader Styles Manager tool. Or on the Format toolbar, click the Multileader tool.
 - Type *mleaderstyle* and then press Enter.
- 2 Select a multileader style, then click Modify. Or if you want to create a new multileader style instead, see “Creating a multileader style” on page 198 in this chapter.
- 3 Click the Leader Format tab and make your selections.
- 4 Click OK when finished.




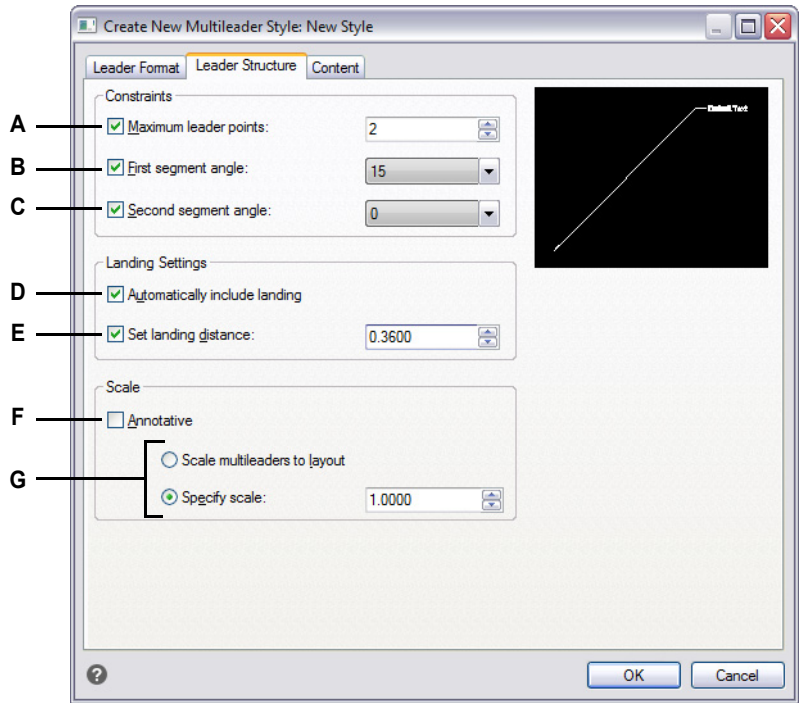
- A** Select the leader line type: straight, spline, or no leader line.
- B** Select the leader line color for multileaders assigned this style.
- C** Select the leader linetype for multileaders assigned this style.
- D** Select the leader lineweight for multileaders assigned this style.
- E** Select the leader line arrowhead for multileaders assigned this style.
- F** Enter or select the leader line arrowhead size for multileaders assigned this style.
- G** Enter or select the leader break size for multileaders assigned this style. The break size is used when the Dimension Break command is used with multileaders that are assigned this style.

Controlling leader structure settings for multileader styles

You can control settings affecting leader constraints, landing settings, and leader scale. Any changes you make affect the multileader style you're working with (a new or existing one). The image tile on the right side of the Create New/Modify Multileader Styles dialog box shows the appearance of a multileader based on the current multileader style settings.

To set the leader structure for multileader styles

- 1 Do one of the following to choose Multileader Styles Manager ():
 - On the ribbon, choose Home > Multileader Styles Manager (in Annotation) or Annotate > Multileader Styles Manager (in Leaders).
 - On the menu, choose Format > Multileader Styles Manager.
 - On the Multileader or Styles toolbar, click the Multileader Styles Manager tool. Or on the Format toolbar, click the Multileader tool.
 - Type *mleaderstyle* and then press Enter.
- 2 Select a multileader style, then click Modify. Or if you want to create a new multileader style instead, see “Creating a multileader style” on page 198 in this chapter.
- 3 Click the Leader Structure tab and make your selections.
- 4 Click OK when finished.



- A** Select to limit the number of points for leader line, then enter or select the maximum number of points.

B Select to specify the angle of the first line segment, then enter or select the angle.

C Select to specify the angle of the second line segment, then enter or select the angle.

D Select to include a landing line horizontally between the leader and text content.
- E** Select to specify the length of the landing line, then enter or select the length of the landing line.


F Select to specify that multileaders with this style are affected by annotation scaling.

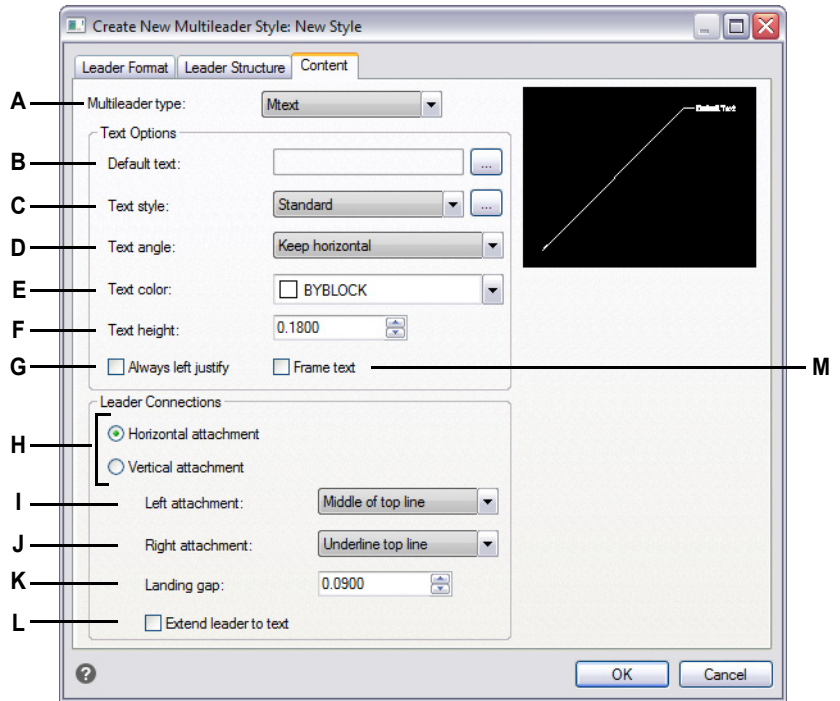
G Select whether to scale multileaders automatically according to layout scaling or to enter a fixed scale. (Available for non-annotative styles.)

Controlling content settings for multileader styles

You can control settings for the content located at the end of multileaders. Specifically, choose whether the content is multiline text, a block, or none. Then you make further selections. The image tile on the right side of the Create New/Modify Multileader Styles dialog box shows the appearance of a multileader based on the current multileader style settings.


To set the content settings for a multileader style with multiline text

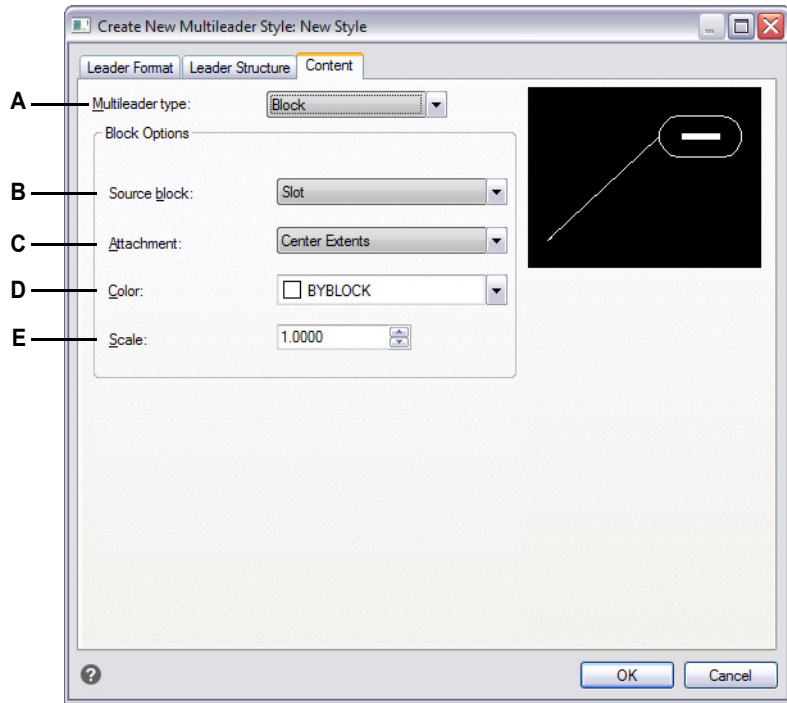
- 1 Do one of the following to choose Multileader Styles Manager ():
 - On the ribbon, choose Home > Multileader Styles Manager (in Annotation) or Annotate > Multileader Styles Manager (in Leaders).
 - On the menu, choose Format > Multileader Styles Manager.
 - On the Multileader or Styles toolbar, click the Multileader Styles Manager tool. Or on the Format toolbar, click the Multileader tool.
 - Type *mleaderstyle* and then press Enter.
- 2 Select a multileader style, then click Modify. Or if you want to create a new multileader style instead, see “Creating a multileader style” on page 198 in this chapter.
- 3 Click the Content Settings tab.
- 4 In Multileader Type, choose Mtext.
- 5 Continue making your selections.
- 6 Click OK when finished.



- A** Select whether a multileader with this style contains multiline text or a block, or none. The options in the dialog box change depending on your selection.
- B** Displays the default text for the multiline text content. Click [...] to specify the default text.
- C** Select a text style for the multiline text. Click [...] to manage text styles.
- D** Select whether the rotation angle of multiline text is as-inserted, always right-reading, or always horizontal.
- E** Select a color for the multiline text.
- F** Enter or select the height of the multiline text.
- G** Select to always left justify the multiline text.
- H** Select to attach multiline text content to the multileader horizontally or vertically. The options that display depend on your selection.
- I** Select the location of multiline text when it is attached to the left (for horizontal) or top (for vertical) of the leader line.
- J** Select the location of multiline text when it is attached to the right (for horizontal) or bottom (for vertical) of the leader line.
- K** Enter or select the distance between the landing line and multiline text.
- L** Select to extend the multileader to the text when needed. (Available only for horizontal attachments.)
- M** Select to add a frame around the multiline text.

To set the content settings for a multileader style with a block

- 1 Do one of the following to choose Multileader Styles Manager ():
 - On the ribbon, choose Home > Multileader Styles Manager (in Annotation) or Annotate > Multileader Styles Manager (in Leaders).
 - On the menu, choose Format > Multileader Styles Manager.
 - On the Multileader or Styles toolbar, click the Multileader Styles Manager tool. Or on the Format toolbar, click the Multileader tool.
 - Type *mleaderstyle* and then press Enter.
- 2 Select a multileader style, then click Modify. Or if you want to create a new multileader style instead, see “Creating a multileader style” on page 198 in this chapter.
- 3 Click the Content Settings tab.
- 4 In Multileader Type, choose Block.
- 5 Continue making your selections.
- 6 Click OK when finished.



- A** Select whether a multileader with this style contains multiline text or a block, or none. The options in the dialog box change depending on your selection.
- B** Select the block to use at the end of the multileader.
- C** Select which point to use for attaching the block to the multileader: insertion point that you specify or center point.
- D** Select the color of the block, which is applied if the block entity color is ByBlock.
- E** Enter or select the scale of the block.

Defining boundaries

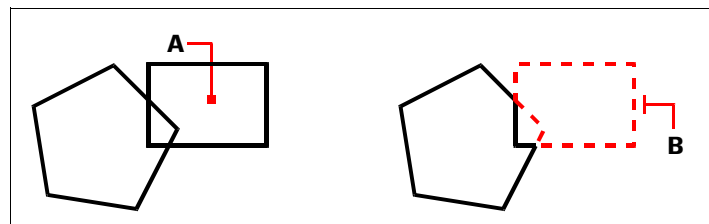
A bounded polyline is an area bound by a single closed entity or by multiple entities that intersect and is used for hatching.

For details about adding hatching after you create a boundary, see “Adding hatching and gradients” on page 212 in this chapter.

Understanding boundaries

With the Boundary command, you can designate a specific area of a drawing for operations such as hatching and dimensioning. You create a boundary by selecting an area inside a closed loop. The area you select can be bounded by a single closed entity or by multiple entities that intersect. The resulting boundary can be a polyline or region, if your program version supports working with regions.

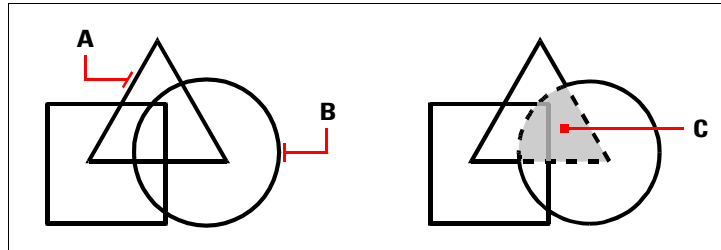
In cases where entities intersect, PTCAD interprets the boundary as the closed loop closest to the point specifying the area. In the following figure, for example, the area point selected in the rectangle results in a boundary consisting of the closed loop nearest the point selection, as opposed to the closed loop formed by the rectangle itself.



Selected point (A) and resulting boundary (B).

To make boundaries more specific, you can create a boundary set. A boundary set specifies which entities are considered in determining the boundary path. This can make creating the boundary faster if you are working with a complex drawing.

In the following figure, the circle and triangle are the selected entities. If you select an area anywhere inside the circle or the triangle, the result is a polyline (or region if your program version supports regions) that bounds the shaded area.




Selected entities (A and B) and the point specified in the selected area (C), which results in a new boundary around the shaded area.

Defining a boundary

When you create a boundary, you select an existing entity or multiple entities to define the boundary.

To define boundaries from enclosed areas

Advanced experience level

- 1 Do one of the following to choose Boundary ():
 - On the ribbon, choose Home > Boundary (in Draw) or choose Draw > Boundary (in Draw).
 - On the menu, choose Draw > Hatch > Boundary.
 - On the Draw toolbar, click the Boundary tool.
 - Type *boundary* and then press Enter.
- 2 In Boundary Set, optionally specify a subset of entities to consider when creating the boundary:
 - Click Select Boundary Set.
 - In the drawing, select the entities to consider when creating the boundary, and press Enter when finished.

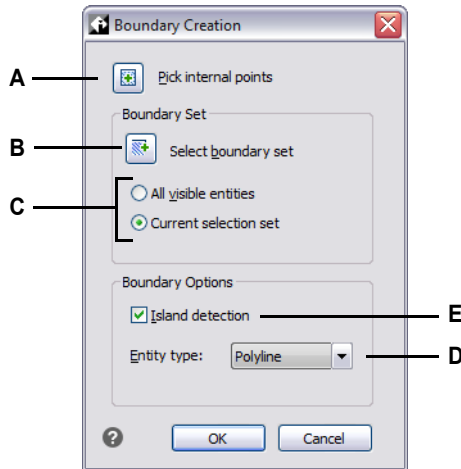
The Boundary Creation dialog box displays again and the Current Selection Set option becomes selected automatically, which indicates that the entities you selected will be considered when creating the boundary.

You don't have to select entities again using Select Boundary Set.

The Current Selection Set option uses the last set of entities you selected with Select Boundary Set.

- 3 Mark Island Detection if you want to consider all internal, closed boundaries, called islands, for defining a boundary.

- 4 In Entity Type, choose whether to create the resulting boundaries as polylines or regions. Regions are available only for program versions that support working with regions.
- 5 Click Pick Internal Points.
- 6 In the drawing, click inside the area whose closed perimeter forms the boundary, not on the entity itself. If desired, continue clicking inside additional closed perimeters.
- 7 To complete the selection and create the boundaries, press Enter.



- | | |
|---|---|
| <p>A Click to pick enclosed areas used to create new boundaries.</p> <p>B Click to customize the selection set of entities in the drawing to be considered when creating boundaries.</p> <p>C If you clicked Select Boundary Set, choose whether to consider all visible entities when creating boundaries or only the entities in the custom selection set.</p> | <p>D Click to consider all internal, closed boundaries when creating boundaries.</p> <p>E Select whether to create the boundary as a polyline or region (for program versions that support working with regions).</p> |
|---|---|

To define a boundary around a hatch

- 1 Type `hatchgenerateboundary` and then press Enter.
- 2 Select the hatch entities around which you want to create boundaries, then press Enter.

The boundaries are created as polylines and are not associated with the hatch entities.

Adding hatching and gradients

When you add hatching or gradients to a drawing, PTCAD fills entities or enclosed areas with a pattern.

Adding hatching or gradients is a four step process:

- 1 Specify a hatch or gradient pattern.
- 2 Specify entities or areas.
- 3 Optionally specify island detection options.
- 4 Specify additional options.

Hatch patterns and gradients are memory intensive. Because hatch and gradient patterns can take a considerable amount of time to draw and display, you may want to add hatching and gradients during the last steps of drawing creation or insert them on a separate layer that you can freeze as you continue to work on your drawing. For example, instead of using a small-scaled line pattern to create a solid fill, use the SOLID hatch pattern instead as this will print and display much faster.

Specifying a hatch or gradient pattern

A hatch pattern consists of a repeating pattern of lines, dashes, and dots. You can select one from a set of predefined patterns, define a pattern of your own, or choose a pattern from a custom hatch library.

A gradient pattern consists of a gradual or smooth change from one color to another color. You simply select colors and a pattern.

Existing hatches and gradients can be modified.


Type `hatchedit` to modify an existing hatch or gradient, double-click it in the drawing, or change its settings in the Properties pane.


Specifying a predefined hatch pattern

Predefined hatch patterns are stored in the following hatch pattern library files:

- *icad.pat* — American National Standards Institute (ANSI)-compliant patterns.
- *icadiso.pat* — International Standards Organization (ISO)-compliant patterns.

To specify a predefined hatch pattern

- 1 Do one of the following to choose Hatch ():
 - On the ribbon, choose Home > Hatch (in Draw) or Draw > Hatch (in Draw).
 - On the menu, choose Draw > Hatch > Hatch.
 - On the Draw toolbar, click the Hatch tool.
 - Type *bhatch* and then press Enter.


- 2 Click the Hatch tab.
- 3 In Type, select Predefined.
- 4 In Pattern, select the desired pattern. You can also click [...] or click the swatch to open the Hatch Patterns dialog box and select a pattern from the ANSI, ISO, or Other Predefined tabs.
- 5 Specify any of the following options:
 - **Color** In the first (left) color list, select the color of the hatch lines. In the second (right) color list, select the color of the hatch background.
 - **Angle** Enter the angle of the hatch pattern relative to the x-axis in degrees (1-360). The default angle is clockwise; you can change the angle by entering a numerical value.
 - **Scale** Enter the scale factor as a percentage of the default. Changing the scale factor makes the hatch pattern larger or smaller than the default size. If you selected an ISO standard pattern, the scale affects the ISO pen width.
 - **Related to paper space** Select to scale the hatch pattern relative to the units defined for paper space. (Available for named layouts.)
 - **ISO pen width** Enter the pen width if you selected an ISO standard pattern.
- 6 In Hatch Origin, select whether to draw the hatch pattern from the default origin or a new origin. If you choose to specify a new origin, do one of the following:
 - Click () to pick the new origin directly in the drawing.
 - Select Default to Boundary Extents to set the origin to the top left, top right, bottom left, bottom right, or center of the boundary extents.


A new origin can be saved as the new default.

To store the new hatch origin as the default, select Store as Default Origin. The hatch origin will be saved in the HPORIGIN system variable.

- 7 To continue, specify entities or areas for hatching. For details, see “Specifying entities or areas for hatching and gradients” on page 220 in this chapter.

To specify a user-defined hatch pattern

- 1 Do one of the following to choose Hatch ():
 - On the ribbon, choose Home > Hatch (in Draw) or Draw > Hatch (in Draw).
 - On the menu, choose Draw > Hatch > Hatch.
 - On the Draw toolbar, click the Hatch tool.
 - Type *bhatch* and then press Enter.
- 2 Click the Hatch tab.

- 3 In the Type list, select User Defined.
- 4 Specify any of the following options:
 - **Color** In the first (left) color list, select the color of the hatch lines. In the second (right) color list, select the color of the hatch background.
 - **Angle** Enter the angle of the hatch pattern relative to the x-axis in degrees (1-360). The default angle is clockwise; you can change the angle by entering a numerical value.
 - **Double** Select to cross-hatch the pattern, which places another copy of the pattern at a 90-degree angle over the original.
 - **Related to paper space** Select to scale the hatch pattern relative to the units defined for paper space. (Available for named layouts.)
 - **Spacing** Enter the line spacing for the hatch pattern.
- 5 In Hatch Origin, select whether to draw the hatch pattern from the default origin or a new origin. If you choose to specify a new origin, do one of the following:
 - Click () to pick the new origin directly in the drawing.
 - Select Default to Boundary Extents to set the origin to the top left, top right, bottom left, bottom right, or center of the boundary extents.

A new origin can be saved as the new default.

To store the new hatch origin as the default, select Store as Default Origin. The hatch origin will be saved in the HPORIGIN system variable.

- 6 To continue, specify entities or areas for hatching. For details, see “Specifying entities or areas for hatching and gradients” on page 220 in this chapter.

Specifying a custom library hatch pattern



You can use custom external hatch pattern libraries (.pat files), such as a standard library used in your office, customized patterns, and libraries available from vendors or standards organizations. The .pat files can be in any location.

Each .pat file contains one hatch pattern, and the filename must match the name of the hatch pattern. The hatch pattern definition is as follows:

```
*[Name], [Description]
angle, x-origin, y-origin, delta-x, delta-y [, dash1, dash2, ...]
```

Alternatively, you can add a custom hatch pattern to icad.pat and use predefined hatch patterns described previously instead of a custom library pattern below.

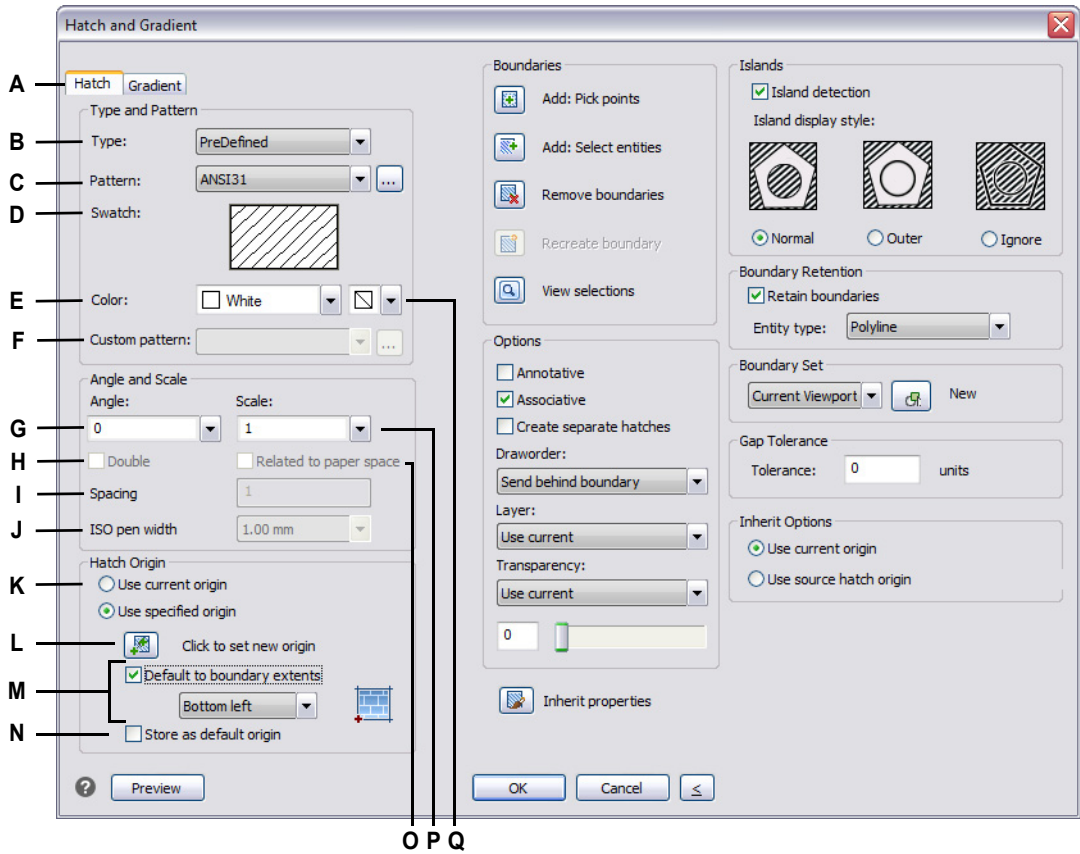
To use a custom library pattern

- 1 Do one of the following:
 - Copy the custom pattern file (.pat file) to the Patterns folder where you installed PTCAD. To check where PTCAD searches for pattern files, choose Tools > Options, click Paths/Files, and check the folders listed for Hatch Patterns.
 - Choose Tools > Options, click Paths/Files, and add the location of the custom pattern file (.pat file) to the folders listed for Hatch Patterns.
- 2 Do one of the following to choose Hatch ():
 - On the ribbon, choose Home > Hatch (in Draw) or Draw > Hatch (in Draw).
 - On the menu, choose Draw > Hatch > Hatch.
 - On the Draw toolbar, click the Hatch tool.
 - Type *bhatch* and then press Enter.
- 3 Click the Hatch tab.
- 4 In Type, choose Custom.
- 5 In Custom Pattern, select a pattern from the list. You can also click [...] to open the Hatch Patterns dialog box and select a pattern from the Custom tab.
- 6 Specify any of the following options:
 - **Color** In the first (left) color list, select the color of the hatch lines. In the second (right) color list, select the color of the hatch background.
 - **Angle** Enter the angle of the hatch pattern relative to the x-axis in degrees (1-360). The default angle is clockwise; you can change the angle by entering a numerical value.
 - **Scale** Enter the scale factor as a percentage of the default. Changing the scale factor makes the hatch pattern larger or smaller than the default size.
 - **Related to paper space** Select to scale the hatch pattern relative to the units defined for paper space. (Available for named layouts.)
- 7 In Hatch Origin, select whether to draw the hatch pattern from the default origin or a new origin. If you choose to specify a new origin, do one of the following:
 - Click () to pick the new origin directly in the drawing.
 - Select Default to Boundary Extents to set the origin to the top left, top right, bottom left, bottom right, or center of the boundary extents.

A new origin can be saved as the new default.

To store the new hatch origin as the default, select Store as Default Origin. The hatch origin will be saved in the HPORIGIN system variable.

- 8 To continue, specify entities or areas for hatching. For details, see “Specifying entities or areas for hatching and gradients” on page 220 in this chapter.




- A Select to create a hatch pattern.
- B Select the type of hatch pattern.
- C Select a predefined hatch pattern from the list or click [...] to select it in the Hatch Patterns dialog box.
- D Click to select a hatch pattern.
- E Select the color of the hatch lines.
- F Select a custom hatch pattern from the list or click [...] to select it in the Hatch Patterns dialog box.
- G Enter the angle of the hatch pattern in relation to the x-axis.
- H Select to cross-hatch a user defined pattern.
- I Enter the line spacing for a user defined pattern.
- J Enter the pen width for an ISO-standard predefined pattern.
- K Select to use the default hatch origin or to specify a new origin.
- L Click to pick the new hatch origin in the drawing.
- M Select to set the new hatch origin to the top left, top right, bottom left, bottom right, or center of the boundary extents.
- N Select to save the new hatch origin as the new default.
- O Select to scale the hatch pattern relative to the units defined for paper space. (Available for named layouts.)
- P Enter the scale factor as a percentage of the default.
- Q Select the color of the hatch background.

Specifying a super hatch pattern

In addition to standard hatch patterns, you can use the Super Hatch command to apply a hatch pattern created by automatically repeating an image, texture, block, external reference, or wipeout.

Different than a standard hatch pattern, after you create a super hatch pattern, it is not associative with the boundary used to create it.

To use a super hatch pattern

- 1 Do one of the following to choose Super Hatch (- On the ribbon, choose Home > Super Hatch (in Draw), Draw > Super Hatch (in Draw), or Express Tools > Super Hatch (in Draw).
- On the menu, choose Draw > Hatch > Super Hatch or Express Tools > Draw > Super Hatch.
- Type *superhatch* and then press Enter.
- 2 In Curve Error Tolerance, optionally enter a tolerance value when approximating vectors of super-hatch boundaries that are curved.
- 3 Select which type of entity you want to use as a hatch pattern:
 - **Image** Click to select an image file to use as a hatch pattern. In the Attach Image dialog box, specify additional settings for inserting the image file in the drawing.
 - **Texture** Click to select a texture file to use as a hatch pattern. Textures are image files with a seamless design and are easy to use with super hatches. In the Attach Image dialog box, specify additional settings for inserting the texture file in the drawing. Several textures are available with the program.
 - **Block** Click to open the Insert Block dialog box and choose the drawing file or existing block to use as a hatch pattern.
 - **Xref** Click to open the Attach External Reference dialog box and choose an external reference file to use as a hatch pattern.
 - **Wipeout** Click to create a hatch using a new wipeout entity.

Alternatively, if the image, block, external reference, or wipeout you want to use as a hatch already exists in the drawing, click Select Pattern to select it in the drawing, then follow the prompts.
- 4 Click OK.
- 5 Optionally choose Advanced to specify more options for defining the area boundary.
- 6 Pick a point inside the enclosed area you want to hatch.


You may want to turn off image and wipeout frames after you create a super hatch.

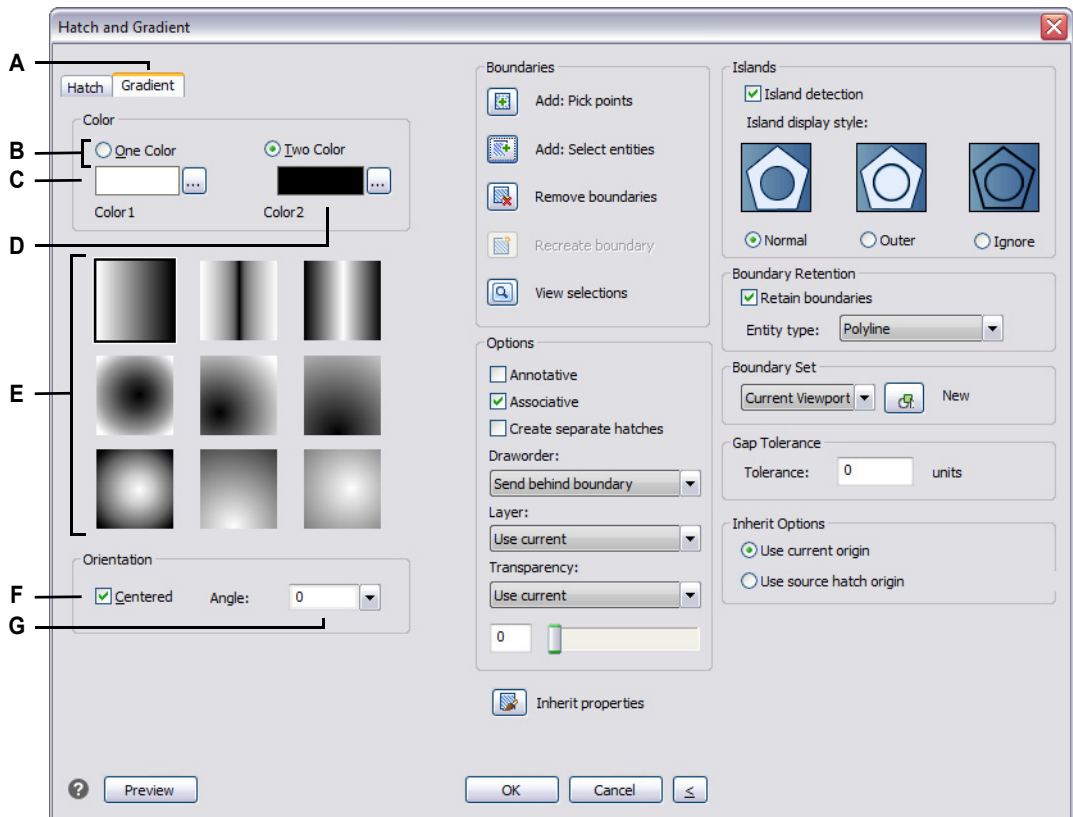
Type tframes to turn image and wipeout frames off (and on).

Specifying a gradient pattern

A gradient pattern consists of a gradual or smooth change from one color to another color. Simply select the colors, a pattern, and orientation settings.

To specify a gradient pattern

- 1 Do one of the following to choose Gradient (- On the ribbon, choose Home > Gradient (in Draw) or Draw > Gradient (in Draw).
- On the menu, choose Draw > Hatch > Gradient.
- On the Draw toolbar, click the Gradient tool.
- Type *gradient* and then press Enter.
- 2 In Color, choose one of the following:
 - **One Color** Specify one color for the gradient pattern, then specify the color in Color1. The color you specify will be smoothed with the color white or black, depending on the GFCLRSTATE system variable.
 - **Two Colors** Specify two colors for the gradient pattern, then specify the colors in Color1 and Color2. Color1 will be smoothed with Color2.
- 3 In the 3x3 display of gradient patterns, select a gradient pattern.
- 4 Specify the orientation options:
 - **Centered** Select to center the gradient pattern. If not selected, the gradient pattern appears to have a light source above and to the left (the pattern is shifted up and to the left).
 - **Angle** Enter or select the angle of the gradient pattern.
- 5 To continue, specify entities or areas for gradients. For details, see “Specifying entities or areas for hatching and gradients” on page 220 in this chapter.





- A** Select to create a gradient pattern.
- B** Select to specify one or two colors for the gradient pattern.
- C** Select the first color of the gradient pattern; click [...] for more color options.
- D** Select the second color if creating a two-color gradient pattern; click [...] for more color options.
- E** Select a gradient pattern.
- F** Select to center the gradient pattern; if not selected, the pattern will shift up and to the left.
- G** Enter or select an angle for the gradient pattern.

Specifying entities or areas for hatching and gradients

You can specify either entities or areas in which to apply hatches and gradients:

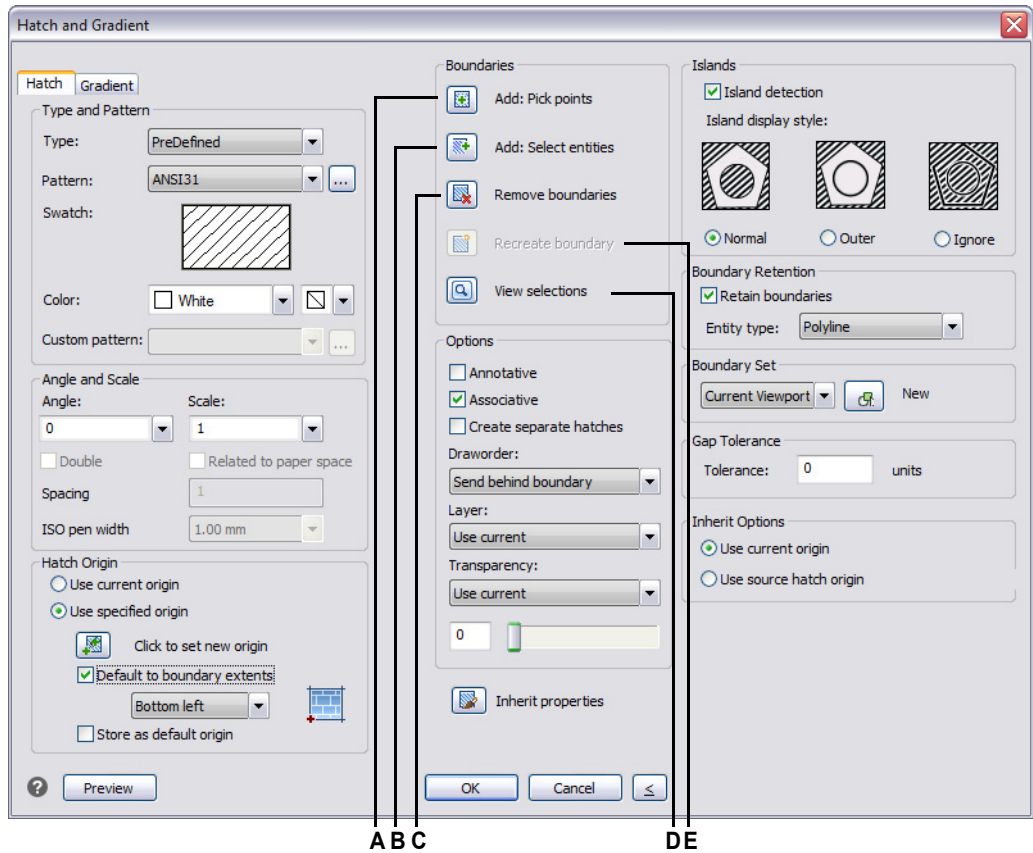
- **Select entities** — Add hatching or gradients to any entity that forms a closed boundary, for example, a circle or rectangle. You can assign hatching or gradients to a single entity or several entities at the same time.
- **Select areas** — Add hatching or gradients to an area enclosed by entities that form a boundary. The hatch or gradient is created in the enclosed area, not the entities themselves. The entire hatch or gradient, once drawn, is treated as a single entity and it is either associative or independent of the boundary entities.

To select entities or areas for hatching and gradients

- 1 Do one of the following to choose Hatch () or Gradient ():
 - On the ribbon, choose Home > Hatch or Home > Gradient (in Draw); or choose Draw > Hatch or Draw > Gradient (in Draw).
 - On the menu, choose Draw > Hatch > Hatch, or Gradient.
 - On the Draw toolbar, click the Hatch tool or Gradient tool.
 - Type *bhatch* or *gradient* and then press Enter.
- 2 Click one of the following:
 - **Add: Pick points** Specifies areas for which to apply hatching and gradients. In the drawing, click inside the closed perimeter of a boundary, not on the boundary itself. If desired, continue clicking inside additional closed perimeters. To complete the selection, press Enter.
 - **Add: Select entities** Specifies entities for which to apply hatching and gradients. In the drawing, click the entities individually or by choosing a selection method from the prompt box, and then press Enter when done.
 - **Remove boundaries** Removes boundaries from the selection set. In the drawing, click the boundaries to remove from the selection set.
 - **Recreate boundary** Creates a polyline or region around a hatch or gradient. In the drawing, click the hatch or gradient for which you want to create a boundary.
 - **View selections** Displays the selection set. In the drawing, press Enter when done viewing the selection set.
- 3 To continue, specify additional options. For details, see “Specifying additional hatch and gradient options” on page 223 in this chapter.

A warning message displays when you create the hatch or gradient if the number of entities selected exceeds the HPOBJWARNING system variable value.

If the warning displays, to improve performance before continuing with hatch creation, select fewer entities. Hatch patterns are memory intensive and can take a considerable amount of time to draw.

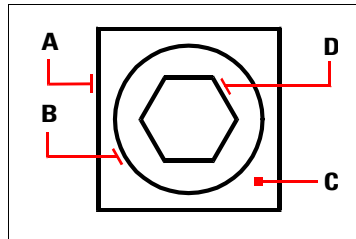


- A** Click to select a point inside one or more entities that create a boundary.
- B** Click to select one or more entities.
- C** Click to remove a boundary from the selection set.
- D** Click to view currently selected entities in the drawing.
- E** Click to select a hatch or gradient for which to surround with a polyline or region.

Specifying islands and island detection

Islands are closed loops that reside inside other closed loops. PTCAD provides island-detection options so that you can specify which islands, if any, should be considered in the area selection for a boundary.

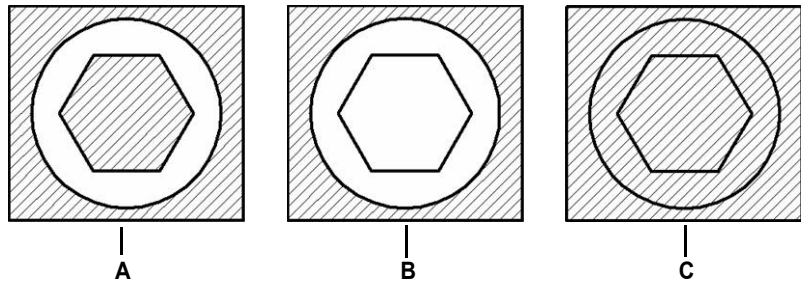
The following figure shows a rectangular polyline with two islands. The circle is referred to as the outer island, and the octagon is referred to as a nested island.



Rectangular polyline (A) with outer island (B), with point (C) specifying the area selection, and nested island (D).



You can choose from three island-detection methods.

- **Nested Islands** The outer entity and all its islands are considered for the polyline.
- **Outer Only** Only the outer entity and its outer island are considered for the polyline.
- **Ignore Islands** Only the outer entity is considered for the polyline.



Nested islands (A), with outer island (B), and with ignore islands (C).

To specify island detection options

- 1 Do one of the following to choose Hatch () or Gradient ():
 - On the ribbon, choose Home > Hatch or Home > Gradient (in Draw); or choose Draw > Hatch or Draw > Gradient (in Draw).
 - On the menu, choose Draw > Hatch > Hatch, or Gradient.
 - On the Draw toolbar, click the Hatch tool or Gradient tool.
 - Type *bhatch* or *gradient* and then press Enter.
- 2 In Islands, select Island Detection and then choose one of the following:
 - **Normal** The outer entity and all its islands are considered for hatching.
 - **Outer** Only the outer entity and its outer island are considered for hatching.
 - **Ignore** Only the outer entity is considered for hatching.



Specifying additional hatch and gradient options

There are many options that can be specified for hatches and gradients, including formatting options and specific options related to boundaries.

Existing hatches and gradients can be modified.

Type hatchedit to modify an existing hatch or gradient, double-click it in the drawing, or change its settings in the Properties pane.

To specify additional hatch and gradient options

- 1 Do one of the following to choose Hatch () or Gradient ():
 - On the ribbon, choose Home > Hatch or Home > Gradient (in Draw); or choose Draw > Hatch or Draw > Gradient (in Draw).
 - On the menu, choose Draw > Hatch > Hatch, or Gradient.
 - On the Draw toolbar, click the Hatch tool or Gradient tool.
 - Type *bhatch* or *gradient* and then press Enter.
- 2 Select any of the following formatting options:
 - **Annotative** Makes the display and printing of the hatch or gradient pattern affected by annotation scaling.
 - **Associative** Associates the hatch or gradient pattern with its boundary entities. An associative pattern updates automatically if you move any of its boundaries.

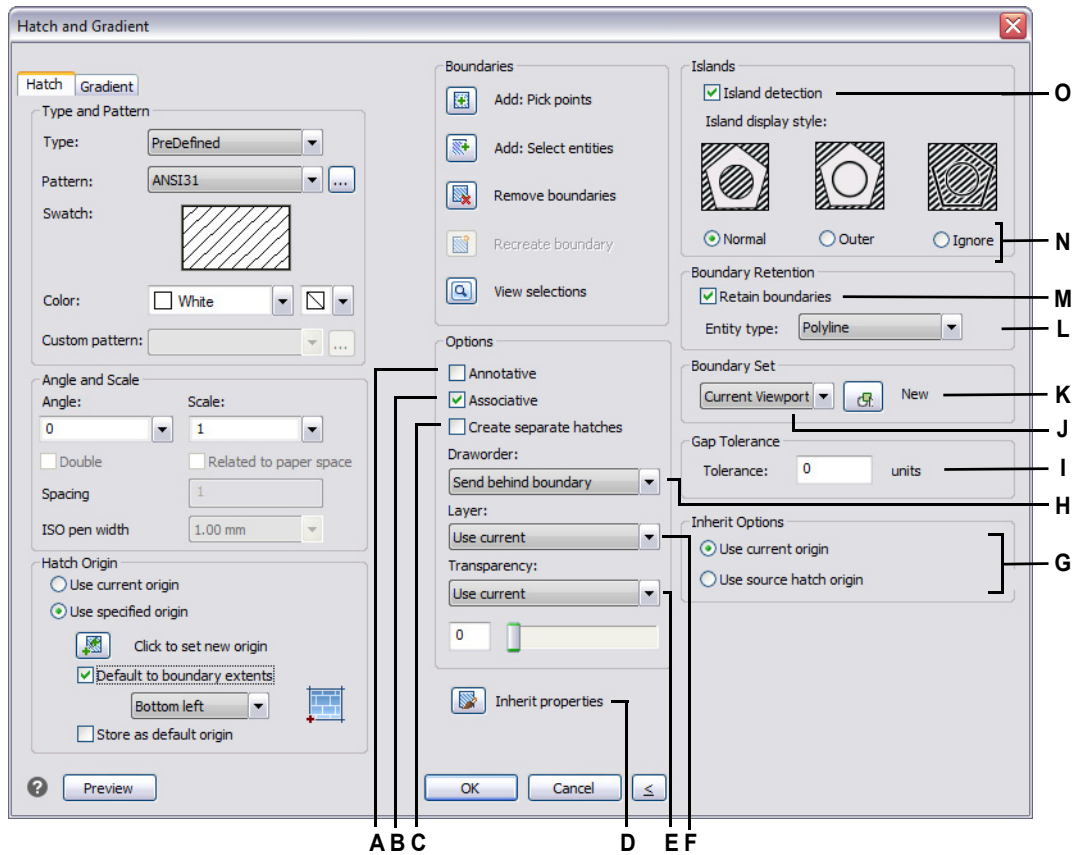
An associative hatch can be selected automatically when its boundary entities are selected.

Use the Hatch Selection command to turn the setting on or off. On the ribbon if your program version includes one, choose Home > Hatch Selection (in Draw) or Draw > Hatch Selection (in Modify), or on the menu choose Draw > Hatch > Hatch Selection.

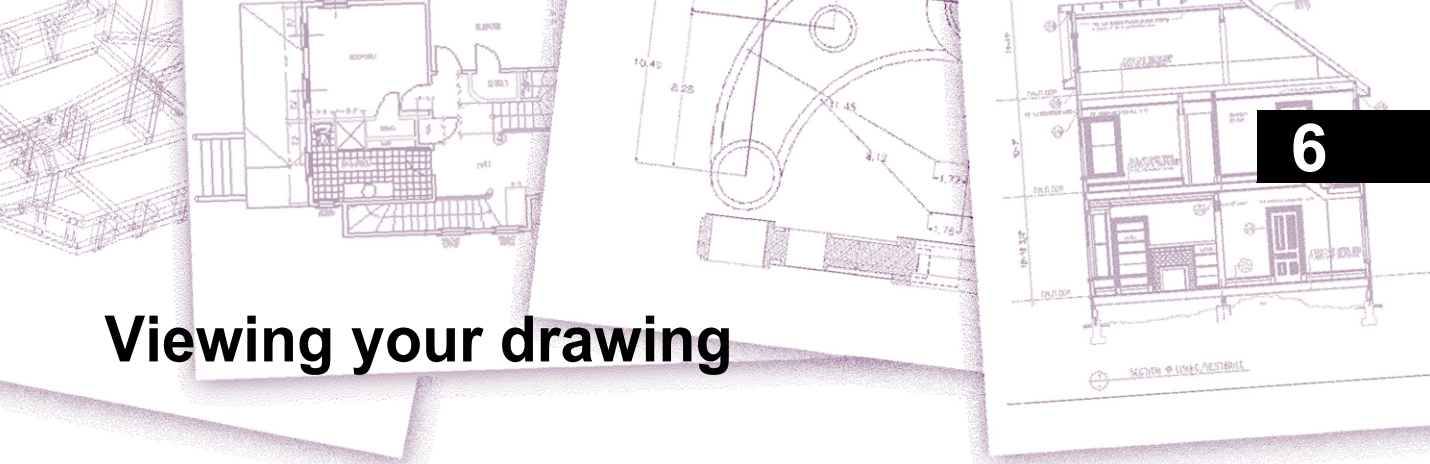
- **Create Separate Hatches** Creates multiple hatches or gradients when more than one closed boundary is selected.
 - **Draw Order** Determines the order in which the hatch or gradient pattern is drawn relative to its boundary.
 - **Layer** Determines the layer on which the hatch or gradient pattern is assigned.
 - **Transparent** Determines the transparency of the hatch or gradient. If you select Specify Value, enter the transparency value or use the slider.
 - **Inherit Properties** Allows selection of a hatch or gradient from which to copy settings and load those settings in the Hatch and Gradient dialog box.
- 3 From the Hatch and Gradient dialog box, click > if necessary to expand the dialog box.
 - 4 Select any of the following boundary-related options:
 - **Boundary Retention** Keeps any new entities that are created for drawing the hatch or gradient. If turned on, select whether to create new entities as polylines or regions. Existing entities are always retained.
 - **Boundary Set** Determines the area of entities that is considered for creating a boundary when using the Add: Pick Points option. Choose Current Viewport to make available all entities within the current viewport. Choose Existing Set to make available only those entities within the area specified with the New option. Click New to specify a new area.
 - **Gap Tolerance** Determines the tolerance that non-touching entities can be within and still be used to create a boundary. For example, if two lines do not meet at the same point but the distance between their endpoints falls within the hatch tolerance, they will be considered closed and can be used to create a boundary. PTCAD calculates the gap tolerance automatically, depending on the size of the PTCAD window.
 - **Inherit Options** Determines whether to use the current origin or the original hatch origin when you use Inherit Properties.
 - 5 Click OK.

Hatch patterns are memory intensive.

Because hatch patterns can take a considerable amount of time to draw and display, you may want to add hatching during the last steps of drawing creation or insert hatches on a separate layer that you can freeze as you continue to work on your drawing. In addition, be sure to use the appropriate hatch scale and pattern. For example, instead of using a small-scaled line pattern to create a solid fill, use the SOLID hatch pattern instead as this will print and display much faster.



- A Select to make the hatch or gradient annotative.
- B Select to update the hatch or gradient automatically if you move any of its boundaries.
- C Select to create a separate hatch or gradient when more than one closed boundary is selected.
- D Click to select a hatch or gradient in the drawing from which to copy settings.
- E Select the transparency of the hatches or gradients; if Specify Value, enter the transparency value or use the slider.
- F Select the layer of the hatches or gradients.
- G Choose the hatch or gradient origin to apply when using Inherit Properties.
- H Select the order of the hatch or gradient relative to the boundary.
- I Enter the tolerance that non-touching entities can be within and still be used to create the hatch pattern boundary.
- J Select the area of entities to consider for creating a boundary when using the Add: Pick Points option.
- K Click to specify a new area for boundary selection when using the Add: Pick Points option.
- L If Retain Boundaries is turned on, select whether to create boundaries as polylines or regions.
- M Select to keep any new entities that are created for drawing the hatch or gradient.
- N Determines how hatching and gradients interact with islands.
- O Select to detect islands when determining boundaries.



Viewing your drawing

PTCAD provides many ways to display and view your drawing. You can also change various display settings to speed up the display or printing of a drawing. This section explains how to:

- Navigate within a drawing by scrolling, panning, and rotating the view.
- Change the magnification of a drawing by zooming in and out.
- View a drawing with annotation scales.
- Work with multiple windows or views of a drawing.
- Control the display of elements to optimize performance when working with large or complex drawings.

When working with three-dimensional drawings, see “Viewing entities in three dimensions” on page 688. To hide, shade, or render a drawing, see “Hiding, shading, and rendering” on page 767.


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Redrawing and regenerating a drawing

As you work on a drawing, visual elements may remain after the completion of a command. You can remove these elements by refreshing, or redrawing, the display.

To redraw (refresh) the current window display

Do one of the following to choose Redraw (

- On the ribbon, choose View > Redraw (in Navigate).
- On the menu, choose View > Redraw.
- On the Zoom toolbar, click the Redraw tool.
- Type *redraw* and then press Enter.

Information about drawing entities is stored in a database as floating point values, ensuring a high level of precision. Sometimes a drawing must be recalculated, or regenerated, from the floating-point database to convert those values to the appropriate screen coordinates. This occurs automatically. You can also manually initiate a regeneration. When the drawing is regenerated, it is also redrawn.

To regenerate the current window, type *regen* in the command bar. If more than one window is displayed, type *regenall* to regenerate all the windows.

Moving around within a drawing

You can move the view of a drawing displayed in the current viewport by scrolling, panning, or rotating the view. Doing this changes the portion of the drawing you are viewing without changing the current magnification. Scrolling lets you move around in the drawing horizontally and vertically. Panning lets you move the drawing in any direction. Rotating lets you view your drawing from any angle.

You can also move to a different view using the following:

- Model and Layout tabs. For more details, see “Viewing drawings in paper space and model space” on page 618.
- Preset views. For more details, see “Switching to a preset viewing direction” on page 688.
- Named views. For more details, see “Creating and switching to named views” on page 690.
- Cameras. For more details, see “Using a camera to view the drawing” on page 704.

Using scroll bars


To assist you in navigating within a drawing, horizontal and vertical scroll bars are available in each drawing window. The size of the scroll box in relation to the scroll bar indicates the current level of drawing magnification. The position of the scroll box in relation to the scroll bar indicates the location of the center of the drawing in relation to the extents of the drawing (the smallest rectangle containing all the entities in the drawing).

To turn scroll bars on or off

Do one of the following to choose Scroll Bars:

- On the ribbon, choose View > Scroll Bars (in Display).
- On the menu, choose View > Display > Scroll Bars.
- Choose Tools > Options > Display tab, and select Show Scroll Bars.
- Type *scrollbar*, press Enter, and then select On, Off, or Toggle.

Panning a drawing

You can move the drawing in any direction using the Pan tool () on the View toolbar. Panning shifts or slides the view of the drawing horizontally, vertically, or diagonally. The magnification of the drawing remains the same, as does its orientation in space. The only change is the portion of the drawing displayed.

If you often pan (and zoom) to a certain area of a drawing, you can create and re-use a view using the View Manager.

For details, see “Working with named views” on page 333.


Panning (and zooming) can be affected by smooth view transitions.

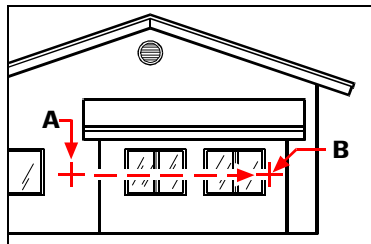
Type vtoptions to specify whether smooth view transitions are used when panning and zooming, and to specify speed and performance settings.

Panning by specifying two points

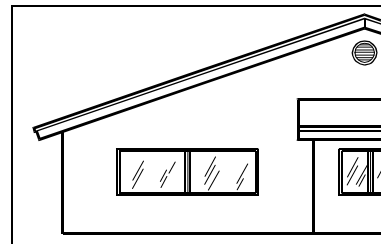
For precise panning, specify two points defining the magnitude and direction of the pan. The first point, or base point, indicates the starting point of the pan. The second point indicates the amount of pan displacement relative to the first point.

To pan by specifying two points

- 1 Do one of the following to choose Pan ():
 - On the ribbon, choose View > Pan (in Navigate).
 - On the menu, choose View > Pan > Pan.
 - On the View toolbar, click the Pan tool.
 - Type *pan* and then press Enter.
- 2 Specify the pan base point either by typing the coordinates or by specifying a point in the drawing window.
- 3 Specify the pan displacement point either by typing the coordinates or by specifying a point in the drawing window.



Select the pan base point (A), and then select a second point (B) to specify the pan displacement.




Result.

Panning in real time

By panning in real time, you control the pan at the same time you move your mouse.

To pan in real time

- 1 Do one of the following to choose Real-Time Pan ():
 - On the ribbon, choose View > Real-Time Pan (in Navigate).
 - Choose View > Pan > Real-Time Pan.
 - On the Zoom toolbar, click the Real-Time Pan tool.
 - Type *rtpan* and then press Enter.
- 2 Click and hold the left mouse button.
- 3 Move the cursor in the direction you want to pan.
- 4 To stop panning, release the mouse button.

Panning using a mouse with a wheel

You can use the wheel of your mouse to help move around in a drawing.

The MBUTTONPAN system variable controls this feature.

To pan using a mouse with a wheel

- Press and hold the wheel, and then move the mouse in the direction you want to pan.

Panning using the arrow keys

To pan in small increments, use the arrow keys.

You can pan using the arrow keys if Use Up/Down Arrows for Command History Navigation is not marked on the Display tab in Tools > Options.

To pan using the arrow keys

- Press the up, down, right, or left arrow keys.


Orbiting the drawing in real time

PTCAD allows you to orbit the drawing, or rotate the view, in real time. This allows you to view your model from any angle while in model space. You cannot rotate the view while in paper space.

Using 3D Orbit commands in large drawing files can be memory intensive.

It may be helpful to set the ZOOMDETAIL system variable to a higher value to reduce the number of entities that display during the command. For example, if the value is set to 10, entities with a pixel size less than or equal to 10 become invisible, and when finished orbiting in real time, the entities become visible again.


To orbit the drawing

- 1 Do one of the following to choose Constrained Orbit (- On the ribbon, choose View > Constrained Orbit (in Navigate).
- On the menu, choose View > 3D Orbit > Constrained Orbit.
- On the 3D Orbit toolbar, click the Constrained Orbit tool.
- Type *3dorbit* and then press Enter.
- 2 Do one of the following:
 - Click and drag the left mouse button to orbit the drawing.
 - Choose Set to pick a different point on which to orbit, then click and drag the mouse to orbit the drawing.
- 3 To stop orbiting, release the mouse button.

Use a shortcut.

Press and hold Shift while viewing a drawing, then click and drag the middle mouse button (wheel) to orbit the drawing.

To orbit the drawing using continuous motion

- 1 Do one of the following to choose Continuous Orbit (- On the ribbon, choose View > Continuous Orbit (in Navigate).
- On the menu, choose View > 3D Orbit > Continuous Orbit.
- On the 3D Orbit toolbar, click the Continuous Orbit tool.
- Type *3dcorbit* and then press Enter.
- 2 Do one of the following:
 - Click and drag the left mouse button to orbit the drawing.
 - Choose Set to pick a different point on which to orbit, then click and drag the mouse to orbit the drawing.
- 3 Release the mouse button. The view continues to orbit.
- 4 When finished, press Enter or Esc, or right-click the drawing.

You can orbit without locking any axis or choose a different axis to lock.

Use the Free Orbit command to orbit the drawing without any axis locked. Use the Constrained X Orbit, Constrained Y Orbit, and Constrained Z Orbit commands to orbit the drawing while maintaining the chosen axis. You can also press Ctrl and use the right mouse button to rotate the view about the z-axis.

Changing the magnification of your drawing

You can change the magnification of your drawing at any time by zooming. The cursor changes to a magnifying glass (🔍) when a zoom tool is active. Zoom out to reduce the magnification so you can see more of the drawing, or zoom in to increase the magnification so you can see a portion of the drawing in greater detail. Changing the magnification of the drawing affects only the way the drawing is displayed; it has no effect on the dimensions of the entities in your drawing.

If you often zoom (and pan) to a certain area of a drawing, you can create and re-use a view using the View Manager.

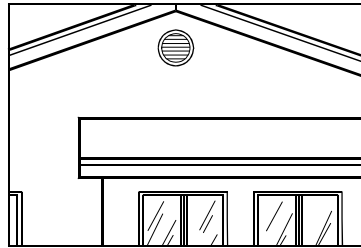
For details, see “Working with named views” on page 333.

If you cannot zoom in a layout viewport, the layout viewport may be locked.

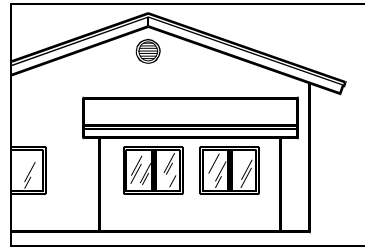
The scale and view do not change in model space while panning or zooming in a locked layout viewport. For more details, see “Modifying layout viewports” on page 628.

Understanding zoom

One of the easiest ways to change the magnification of the drawing is to zoom in or out by a preset increment. On the Zoom toolbar, the Zoom In tool (🔍) doubles the current magnification of the drawing. The Zoom Out tool (🔍) reduces the magnification of the drawing by half. The portion of the drawing located at the center of the current viewport remains centered on the screen as you zoom in and out.



Zoom in.



Zoom out.


Zooming (and panning) can be affected by smooth view transitions.

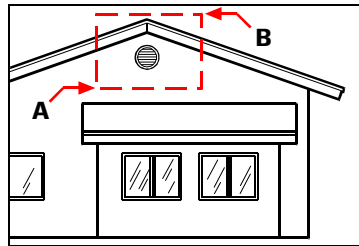
Type vtoptions to specify whether smooth view transitions are used when zooming and panning, and to specify speed and performance settings.

Zooming in to an area using a window

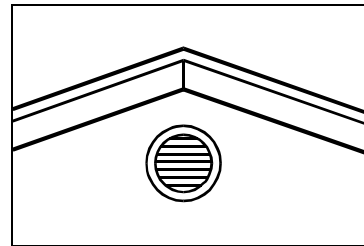
You can create a window that defines the portion of the drawing to which you want to zoom.

To zoom in to an area using a window

- 1 Do one of the following to choose Zoom Window ():
 - On the ribbon, choose View > Zoom Window (in Navigate).
 - On the menu, choose View > Zoom > Window.
 - On the Zoom toolbar, click the Zoom Window tool.
 - Type *zoom* and then press Enter.
- 2 Select one corner of the window around the area you want to magnify.
- 3 Specify the opposite corner of the window around the area you want to magnify.



To specify a rectangular window around the area you want to magnify, select first one corner (A), and then select the opposite corner (B).




Result.

Zooming in to one or more entities

You can zoom in to specific entities that you select. The window fills with the entities that you select.

To zoom in to one or more entities

- 1 Select one or more entities.
- 2 Do one of the following to choose Zoom Entity ():
 - On the ribbon, choose View > Zoom Entity (in Navigate).
 - On the menu, choose View > Zoom > Entity.
 - On the Zoom toolbar, click the Zoom Entity tool.


Zooming in real time

By zooming in real time, you control the zoom at the same time you move your mouse.

Using real-time zoom in large drawing files can be memory intensive.

It may be helpful to set the ZOOMDETAIL system variable to a higher value to reduce the number of entities that display during the command. For example, if the value is set to 10, entities with a pixel size less than or equal to 10 become invisible, and when finished zooming in real time, the entities become visible again.

To zoom in real time

- 1 Do one of the following to choose Real-Time Zoom ():
 - On the ribbon, choose View > Real-Time Zoom (in Navigate).
 - On the menu, choose View > Zoom > Real-Time Zoom.
 - On the Zoom toolbar, click the Real-Time Zoom tool.
 - Type *rtzoom* and then press Enter.
 - Simultaneously press and hold Ctrl + Shift.
- 2 Click and hold the left mouse button.
- 3 To zoom in, move the cursor up the screen; to zoom out, move the cursor down the screen.
- 4 To stop zooming, release the mouse button.

To zoom in real time using a mouse with a wheel

- Rotate the wheel away from you to zoom in or toward you to zoom out.


Customize the mouse wheel.

Settings for the mouse wheel can be customized to accommodate your work style using the ZOOMWHEEL (wheel direction), ZOOMPERCENT (display accuracy for curved entities), and ZOOMFACTOR (zoom factor for the wheel) system variables.

Displaying the previous view of a drawing

After you zoom in or pan to view a portion of your drawing in greater detail, you may want to restore the previous view to see the entire drawing.

To display the previous view of a drawing

- 1 Do one of the following to choose Zoom Previous (- On the ribbon, choose View > Zoom Previous (in Navigate).
- On the menu, choose View > Zoom > Previous.
- On the Zoom toolbar, click the Zoom Previous tool.

Selecting this tool repeatedly steps back through up to 25 successive zoomed or panned views.


Zooming to a specific scale

You can increase or decrease the magnification of your view by a precise scale factor measured relative to the overall size of the drawing or in relation to the current display. When you change the magnification factor, the portion of the drawing located at the center of the current viewport remains centered on the screen.

To change the magnification of the view relative to the overall size of the drawing, type a number representing the magnification scale factor. For example, if you type a scale factor of 2, the drawing appears at twice its original size. If you type a magnification factor of .5, the drawing appears at half its original size.

You can also change the magnification of the drawing relative to its current magnification by adding an *x* after the magnification scale factor. For example, if you type a scale factor of 2*x*, the drawing changes to twice its current size. If you type a magnification factor of .5*x*, the drawing changes to half its current size.

To zoom to a specific scale relative to the current display

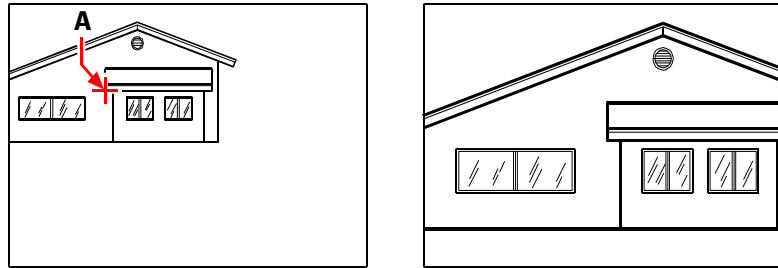
- 1 Do one of the following to choose Zoom In (- On the ribbon, choose View > Zoom In (in Navigate).
- On the menu, choose View > Zoom > Zoom In.
- On the Zoom toolbar, click the Zoom In tool.
- Type *zoom* and then press Enter.
- 2 Type the scale factor, followed by an *x* (such as 2*x*).
- 3 Press Enter.

Combining zooming and panning

You can specify the point you want at the center of the view when you change the drawing magnification. You can specify the point you want at the lower left of the view when you change the magnification of the drawing with the Zoom Left tool (

To change the center of the current view

- 1 Do one of the following to choose Zoom Center (🔍):
 - On the ribbon, choose View > Zoom Center (in Navigate).
 - On the menu, choose View > Zoom > Center.
 - On the Zoom toolbar, click the Zoom Center tool.
 - Type *zoom*, press Enter, and then in the prompt box, choose Center.
- 2 Select the point you want located at the center of the new view.
- 3 Specify the zoom scale factor or the height of the drawing in drawing units.



Current view showing the point to be centered in the new view (A), and the new view zoomed using a scale factor of 2x.

Displaying the entire drawing

There are two main ways you can zoom to display the entire drawing:

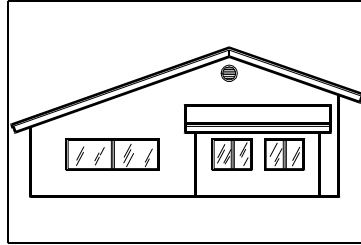
- Zoom all — Displays the entire drawing. If you have drawn any entities outside the defined limits of the drawing, the extents of the drawing are displayed. If all entities are within the limits of the drawing, the drawing is displayed all the way to the drawing limits.
- Zoom extents — Displays the drawing to include all entities (to its extents), making the image fill the display to the greatest possible magnification.

To display the entire drawing

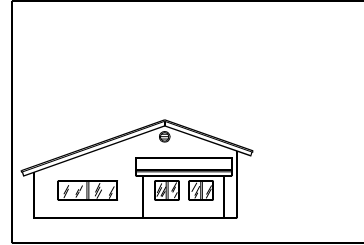
- 1 To display the entire drawing, do one of the following to choose Zoom All (🔍):
 - On the ribbon, choose View > Zoom All (in Navigate).
 - On the menu, choose View > Zoom > All.
 - On the Zoom toolbar, click the Zoom All tool.

Or, to display the drawing to its extents, do one of the following to choose Zoom Extents (🔍):

- On the ribbon, choose View > Zoom Extents (in Navigate).
- On the menu, choose View > Zoom > Extents.
- On the Zoom toolbar, click the Zoom Extents tool.



Zoom extents (displays all entities).



Zoom all (displays to drawing limits).

Changing the view of annotative entities

If your drawing contains annotative entities, such as text and dimensions, you can change the scale, or size, of these entities by setting the annotation scale.

Entities that can be annotative include text, multiline text, tolerances, dimensions, leaders, multileaders, attributes, hatches, and blocks. If Annotative is set to Yes for one of these types of entities and you change the annotation scale, the entity will display at a different scale than other entities in the drawing. For example, if you set the annotation scale to 1:2, all annotative entities will display at that scale (if Automatic Annotation is turned on) or only those annotative entities that support the 1:2 scale will display at that scale (if Automatic Annotation is turned off).


Text styles, dimension styles, and multileader styles also can be annotative, so text, dimensions, or multileaders assigned an annotative style can also be affected.

Turning on scaling of annotative entities

Annotation scaling allows you to control certain entities so their size will consistently display when a drawing is displayed or printed at different scales. You don't have to use annotation scaling, but it is a convenient way to control the scaling of the following entities: text, tolerances, dimensions, leaders, multileaders, attributes, hatches, and blocks.

Text styles, dimension styles, and multileader styles can also use annotation scaling. Text, dimension, and multileader entities that are assigned an annotative style will have annotation scaling turned on by default. For details about text styles, see "Working with text styles" on page 474. For details about dimensions styles, see "Using dimension styles and variables" on page 523. PTCAD supports the display of multileaders and their styles, but not editing.)

To turn annotation scaling on or off for one or more entities

- 1 Do one of the following to choose Properties ():
 - On the ribbon, choose View > Properties (in Panes).
 - On the menu, choose View > Display > Properties or Modify > Properties.
 - On the Modify toolbar, click the Properties tool.
 - Type *entprop* and then press Enter.
 - Press Ctrl +1.

The Properties pane displays.


- 2 Select the desired entities.
- 3 In Annotative, select one of the following:
 - **Yes** Entities display according to the currently set annotation scale.
 - **No** Entities are not affected by the currently set annotation scale.

To assign a scale to all annotative entities automatically


- 1 On the status bar, turn on automatic annotation by double-clicking Automatic Annotation On/Off.
- 2 On the status bar, click Annotations Scales List.
- 3 Choose the annotation scale.

All annotative entities (all entities with Annotative set to Yes) will be assigned the selected annotation scale.


To add the current annotation scale to one more entities

- 1 Do one of the following to choose Add Current Scale ():
 - On the ribbon, choose Annotate > Add Current Scale (in Annotation Scaling).
 - On the menu, choose Modify > Annotative Scale > Add Current Scale.
 - Type *aiobjectscaleadd* and then press Enter.
- 2 Select the desired entities.

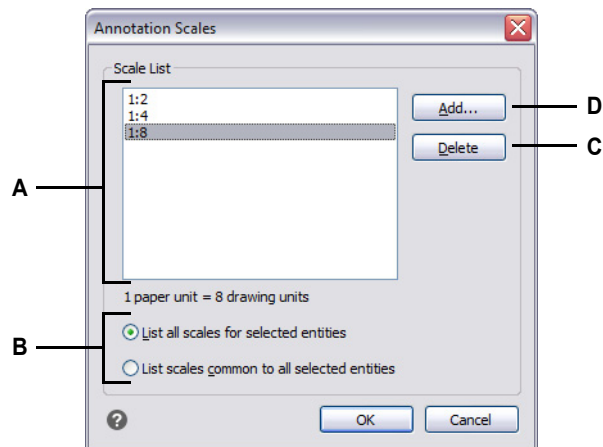
To remove the current annotation scale from one more entities

- 1 Do one of the following to choose Delete Current Scale ():
 - On the ribbon, choose Annotate > Delete Current Scale (in Annotation Scaling).
 - On the menu, choose Modify > Annotative Scale > Delete Current Scale.
 - Type *aiobjectscaleremove* and then press Enter.
- 2 Select the desired entities.

To add or remove annotation scales for one or more entities

- 1 Do one of the following to choose Add/Delete Scales ():
 - On the ribbon, choose Annotate > Add/Delete Scales (in Annotation Scaling).
 - On the menu, choose Modify > Annotative Scale > Add/Delete Scales.
 - Type *objectscale* and then press Enter.
- 2 Select the desired entities.

The Annotation Scales dialog box displays a list of annotation scales that are assigned to the entities.
- 3 Do one of the following:
 - Click Add to select a scale and it to all selected entities.
 - Select a scale and click Delete to remove it from all selected entities.
- 4 Click OK.



- | | |
|---|---|
| <p>A Displays the annotation scales assigned to selected entities.</p> <p>B Select whether to display all annotation scales assigned to selected entities or only those that are in common.</p> | <p>C Click to delete the selected annotation scale from selected entities.</p> <p>D Click to select an annotation scale to assign to selected entities.</p> |
|---|---|

Changing the scale of annotative entities

The annotation scale of a drawing determines the size of annotative entities without changing the scale of other, non-annotative entities.

If automatic annotation is turned on, changing the annotation scale changes the scale, or size, of all entities that have annotation scaling turned on. If automatic annotation is turned off, the size is changed only for annotative entities that are assigned the selected annotation scale.

To set the annotation scale

- 1 On the status bar, click Annotations Scales List.
- 2 Choose the annotation scale.

All enabled annotative entities that have the selected scale assigned to them will display at the new scale.

To assign and set the scale automatically for all annotative entities

- 1 On the status bar, turn on automatic annotation by double-clicking Automatic Annotation On/Off.
- 2 On the status bar, click Annotations Scales List.
- 3 Choose the annotation scale.

All enabled annotative entities (all entities with Annotative set to Yes) will be assigned the selected annotation scale and display at that scale.

Use a system variable.

Automatic annotation can also be set using the ANNOAUTOSCALE system variable.

Displaying and hiding certain annotative entities

An annotative entity can be assigned numerous scales that are used for viewing and printing. By default, if an annotative entity is not assigned the current annotation scale, it still displays but at the default scale. Alternately, you can hide annotative entities that are not assigned the current annotation scale.

You can set the display status for the Model tab and for each Layout tab.


To display or hide annotative entities

- 1 Click the desired Model tab or Layout tab.
- 2 On the status bar, double-click Annotation Visibility On/Off.

Returning scale views of annotative entities to their default positions

Each scale view of an enabled annotative entity can be moved to different locations using grip editing. If various scale views of an annotative entity have been repositioned, you can easily return those scale views to their original basepoint.

To return scale views of annotative entities to their default positions

- 1 Do one of the following to choose Synchronize Multiple-Scale Positions ():
 - On the ribbon, choose Annotate > Sync Scale Positions (in Annotation Scaling).
 - On the menu, choose Modify > Annotative Scale > Synchronize Multiple-Scale Positions.
 - Type *annoreset* and then press Enter.
- 2 Select the desired entities.

Displaying multiple windows

When you begin a new drawing, it is displayed in a single window. You can view the drawing in a second window, or you can divide one window into multiple windows. You can also open and display multiple drawings.

Working with multiple windows of a single drawing

You can open and work with several views of the same drawing simultaneously. There are two methods for dividing the current drawing into multiple views:

- Open a new window of the open drawing.
- Divide the current window into multiple windows.

After you divide a single window into multiple windows, you can control each window separately. For example, you can zoom or pan in one window without affecting the display in any of the other windows. You can control the grid, snap, and view orientation separately for each window. You can restore named views in individual windows, draw from one window to another, and name window configurations individually so you can reuse them later.

As you draw, any changes you make in one window are immediately visible in the others. You can switch from one window to another at any time, even in the middle of a command, by clicking the window's title bar.

Opening a new window of the same drawing

You can open additional windows to create more than one view of a drawing. To open a new window, choose Window > New Window. After you open a new window, you can change its display without affecting any of the other windows.

When you open more than one window for a single drawing, each window is assigned a unique number (for example, mydrawing:1, mydrawing:2, and so on). If your current window is maximized, you can switch to another open window by selecting its name from the bottom of the Window menu.

The names of other open drawings appear at the bottom of the Window menu. You can also use the Cascade, Tile Horizontally, and Tile Vertically commands to arrange all the open windows and drawings. To arrange all the windows and drawings into a stack of identically sized windows, choose Window > Cascade. Arranging windows and drawings in this way makes it easy to see the title bar for each window.

To arrange all the windows and drawings horizontally so they are placed in order from top to bottom, choose Window > Tile Horizontally. Arranging windows and drawings in this way displays each open window. The windows are resized to fit within the available space.

To arrange all the windows and drawings vertically so they are placed side by side, choose Window > Tile Vertically. Arranging windows and drawings in this way displays each open window. The windows are resized to fit within the available space.

To manage all the windows from one dialog box, choose Window > Windows to open the Windows dialog box.

PTCAD uses the commands in the following table to control its windows.

PTCAD window-control commands

Command	Result
<i>vports</i>	Splits the current window into two, three, or four tiled windows.
<i>wcascade</i>	Cascades (overlaps) all open windows.
<i>close</i>	Closes the current window.
<i>closeall</i>	Closes all windows; also closes all drawings.
<i>closeallother</i>	Closes all windows except the current drawing window.
<i>wcloseall</i>	Closes all windows; also closes all drawings.
<i>wiarrange</i>	Arranges window icons.
<i>wopen</i>	Opens another window of the current drawing.
<i>wvtile</i>	Tiles all windows vertically.

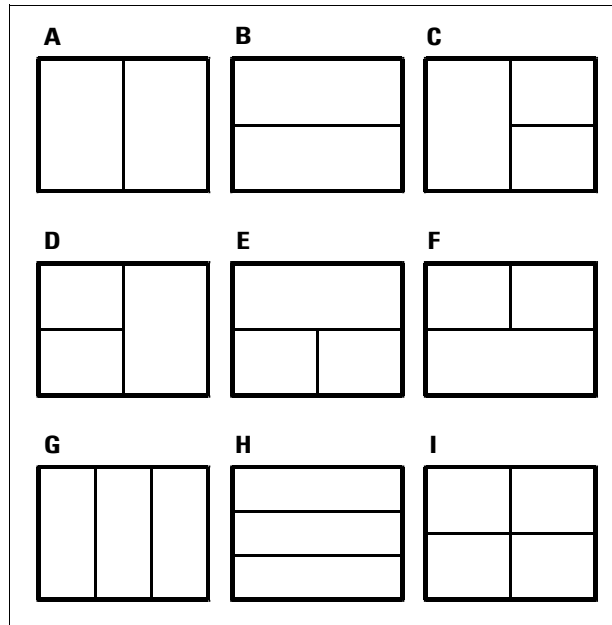
Dividing the current window into multiple windows

You can divide a single drawing window into multiple tiled windows (called viewports) on the Model tab. You can control the number of windows created and the arrangement of the windows. You can also save and restore named window configurations and display a list of the current and saved window configurations.

While working in a viewport, use the Maximize Viewport command to enlarge the view to full size, allowing you to easily work on the geometry in that view. When done, use the Minimize Viewport command to switch back to the original scale and center point of the viewport.

To create multiple viewports

- 1 Do one of the following to choose Viewports (




You can divide a drawing window into two windows arranged vertically (A) or horizontally (B); three windows arranged left (C), right (D), above (E), below (F), vertically (G), or horizontally (H); or four tiled windows (I).

To switch to a different viewport


- 1 Do one of the following:
 - Click the edge of the viewport you want to make active.
 - Press Ctrl+R to make the next viewport active.

To maximize the current viewport

- 1 Do one of the following:
 - On the ribbon, choose View > Maximize Viewport (in Viewports).
 - On the menu, choose View > Viewports > Maximize Viewport.
 - On the status bar, click Maximize Viewport (.
 - Type *vpmax* and then press Enter.

The viewport is enlarged.

To minimize the current viewport (if it is maximized)


- 1 Do one of the following:
 - On the ribbon, choose View > Minimize Viewport (in Viewports).
 - On the menu, choose View > Viewports > Minimize Viewport.
 - On the status bar, click Minimize Viewport ().
 - Type *vpmin* and then press Enter.

The viewport returns to its original scale and center point.

Saving window configurations


If you have divided the drawing window into multiple views, you can save the current window arrangement so that you can recall it to the screen later. The number and placement of the windows are saved exactly as they are currently displayed. The settings for each window are also saved.

To name and save a window configuration

- 1 Do one of the following to choose Viewports ():
 - On the ribbon, choose View > Viewports (in Viewports).
 - On the menu, choose View > Viewports.
 - On the View toolbar, click the Viewports tool.
 - Type *viewports* and then press Enter.
- 2 In the prompt box, choose Save.
- 3 Type a configuration name, and then press Enter.

The name can be up to 255 characters in length and can contain letters, numbers, the dollar sign (\$), hyphen (-), and underscore (_), or any combination.

To restore a named window configuration

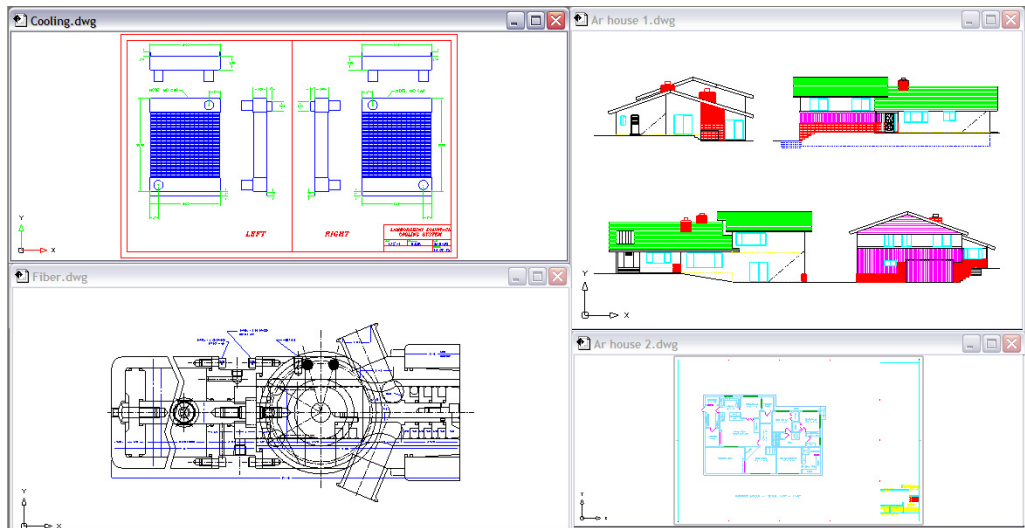
- 1 Do one of the following to choose Viewports ():
 - On the ribbon, choose View > Viewports (in Viewports).
 - On the menu, choose View > Viewports.
 - On the View toolbar, click the Viewports tool.
 - Type *viewports* and then press Enter.
- 2 In the prompt box, choose Restore.
- 3 Type the name of the window configuration you want to restore.

Working with multiple drawings

With the multiple-document interface (MDI) feature, you can open more than one drawing inside of PTCAD. Because you can open and work on several drawings at one time, you can copy, cut, or paste an entity from one drawing to another.

Each drawing appears in a drawing window, which has the following advantages:

- You can see two or more drawings side by side.
- You can easily copy entities from one drawing to another.
- Using PTCAD's Explorer, you can copy such elements as layers, linetypes, and text styles from one drawing to another.
- Like viewports on the Model tab, you can tile or overlap drawing windows; unlike viewports on the Model tab, drawing windows maximize or reduce to an icon.



One session of PTCAD with four drawings open.

Each drawing window that you open and work on retains in the Prompt History log all the commands that you perform, but the command line does not indicate when you have switched windows.

When you work with more than one drawing open in its own window, you can easily move, cut, copy, and paste in between drawings. If you move an entity from one window to another and then want to undo this action, you must undo it in both drawings for it to take effect. If you copy an entity from one window to another and then want to undo that action, you must undo it from the drawing into which you copied the entity. If you cut and paste an entity and then want to undo that action, you must undo it in both drawings.

In the upper right corner of each window are buttons that help manage the display of open drawing windows. The buttons that are available depend on the state of the window. Minimize reduces a window to an icon, Restore expands a window to its normal size, Maximize expands and fills a window in the drawing area, and Close closes a drawing window. Each window can also be resized by dragging its edges and positioned by dragging its titlebar.

PTCAD's primary window type (or document type) is for .dwg drawings. The Start Page is another document type. You can customize PTCAD to display commands according to the document type using the Visibility option of the Customize UI command. For details about how to set this option for menus, see "Setting properties of menu items" on page 874.

Controlling visual elements

The number of entities in your drawing and the complexity of the drawing affect how quickly PTCAD can process commands and display your drawing. You can improve overall program performance by turning off the display of certain visual elements, such as solid fills and text, while you work on the drawing. When you are ready to print your drawing, turn on the display of these elements so your drawing prints the way you want.


You can also improve performance by turning off entity-selection highlighting, turning off the display of marker blips created when you select locations in the drawing, and by turning off the display of lineweights.

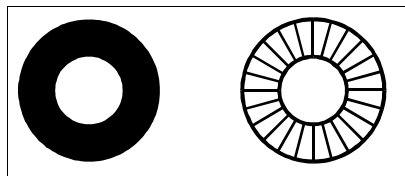
Displaying solid fills

You can reduce the time it takes to display or print a drawing by turning off the display of solid fill. When solid fills are turned off, all filled entities, such as wide polylines and planes, display and print as outlines. When you turn solid fills on or off, you must redraw the drawing before the change is displayed.

A check mark appears next to the Fill command on the Settings menu when it is turned on.

To turn the display of solid fills on or off

- 1 Do one of the following to choose Fill ():
 - On the menu, choose View > Display > Fill.
 - On the Settings toolbar, click the Fill tool.
 - Type *fill* and then press Enter.
- 2 Choose View > Redraw.





Fill on.

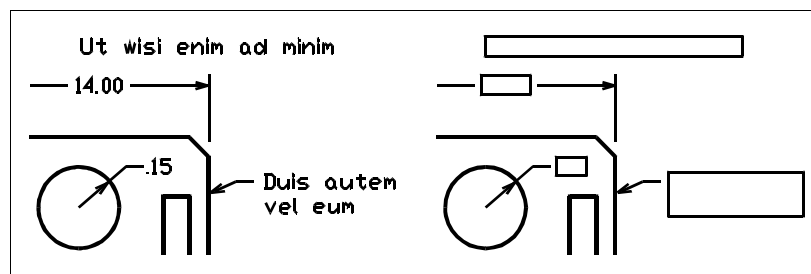
Fill off.

Displaying quick text

Text entities require a considerable amount of time to display and print. You can reduce the time it takes to display or print a drawing by enabling quick text. For example, if you're doing a preliminary check print of a drawing, you may want to turn quick text on to speed up printing. When quick text is enabled, text entities are replaced by rectangular boxes that indicate the outline of the area occupied by the text. When you turn quick text on or off, you must regenerate the drawing before the change is displayed.

To turn the display of quick text on and off

- 1 Do one of the following to choose Drawing Settings ():
 - On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
 - On the menu, choose Tools > Drawing Settings.
 - On the Tools toolbar, click the Drawing Settings tool.
 - Type *qtext*, press Enter, and then in the prompt box, choose On or Off.
- 2 Click the Display tab.
- 3 Click the secondary Display tab.
- 4 Select or clear the Enable Quick Text check box.
- 5 Click OK.
- 6 To regenerate your drawing, do one of the following to choose Regen ():
 - On the ribbon, choose View > Regen.
 - On the menu, choose View > Regen.
 - On the View toolbar, click the Regen tool.
 - Type *regen* and then press Enter.



Quick Text off.


Quick Text on.

Displaying highlighting

When you select entities to modify, the program can highlight them using a dashed line pattern or a glow effect. This highlight disappears when you finish modifying the entities or when the entities are cleared.

Sometimes highlighting entities can take a considerable amount of time, and you may want to improve overall program performance by turning highlighting off.


To turn highlighting on and off

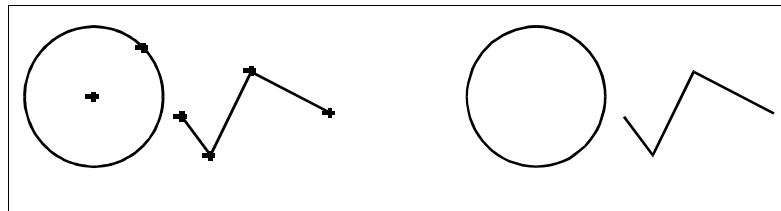
- 1 Do one of the following to choose Drawing Settings ():
 - On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
 - On the menu, choose Tools > Drawing Settings.
 - On the Tools toolbar, click the Drawing Settings tool.
 - Type *highlight*, press Enter, and then in the prompt box, choose On or Off.
- 2 Click the Display tab.
- 3 Click the secondary Display tab.
- 4 Choose from the following options:
 - **Highlight selected entities** Displays selected entities highlighted in a dashed line. When unchecked, selected entities remain unchanged.
 - **Selection effect** Displays selected entities with a glowing highlight instead of a dashed line. This option only applies if Highlight Selected Entities is turned on. The glowing highlight effect applies only when the graphics device is set to OpenGL ES.
- 5 Click OK.

Displaying blips

You can turn blips off. They are the temporary markers that appear on the screen when you select an entity or location. Blips are visible only until you redraw the drawing. You cannot select blips; they are used only for reference and never print.

To turn the display of blips on and off

- 1 Do one of the following to choose Drawing Settings ():
 - On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
 - On the menu, choose Tools > Drawing Settings.
 - On the Tools toolbar, click the Drawing Settings tool.
 - Type *blipmode*, press Enter, and then in the prompt box, choose On or Off.
- 2 Click the Display tab.
- 3 Click the secondary Display tab.
- 4 Select or clear the Show Marker Blips check box.
- 5 Click OK.



Blips on.

Blips off.

Displaying lineweights

You can reduce the time it takes to display a drawing by turning off the display of lineweights. When you turn off lineweights, all entities display as outlines.


You can also specify a lineweight scale. Specify a smaller scale to display thinner lines; specify a larger scale to display thicker lines. For example, a scale factor of 0.5 would display a .80 millimeter lineweight as .40 millimeter; a scale factor of 2 would increase the same lineweight to display at 1.6 millimeters. This can help you differentiate various lineweights that display in a drawing. Adjusting the lineweight display scale affects how the lineweights appear on your screen, not how they appear when printed.

Lineweight scale can affect performance.

Setting the lineweight scale too high may result in slower system performance.

You can also adjust the units for measuring lineweights, and the default lineweight.

To control the display of lineweights

- 1 Do one of the following to choose Drawing Settings ():
 - On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
 - On the menu, choose Tools > Drawing Settings.
 - On the Tools toolbar, click the Drawing Settings tool.
 - Type *dsettings* and then press Enter.
 - Type *lweight* and then press Enter.
- 2 Click the Display tab.
- 3 Click the Lineweights tab.
- 4 Select or clear Display Lineweights.
- 5 In Units for Listing, choose Millimeters or Inches.
- 6 In Default, select the lineweight assigned to layers and entities that use the Default lineweight.
- 7 In Adjust Display Scale, move the slider to the scale you want. By default, the slider begins at 1.00.
- 8 Click OK.

Use a shortcut.

To turn the display of lineweights on or off, use the LWDISPLAY system variable or double-click the word LWT on the status bar.

You can turn lineweights on or off when you print. For details, see “Specifying print options specifically for layouts” on page 640.



Working with coordinates

For accuracy in a drawing, you can locate specific points by entering coordinates as you draw or modify entities. When you create two-dimensional entities, you enter two-dimensional coordinates; for three-dimensional entities, you specify three-dimensional coordinates.

You can also specify coordinates in relation to other known locations or entities in a drawing. In particular, when you work in three-dimensional drawings, it is often easier to specify coordinates in relation to a two-dimensional working plane, called a user coordinate system (UCS).

This section explains how to work with coordinates, including how to:

- Use two-dimensional and three-dimensional coordinate systems.
- Specify absolute and relative coordinates.
- Specify polar, spherical, and cylindrical coordinates.
- Define and manipulate user coordinate systems.

Topics in this chapter

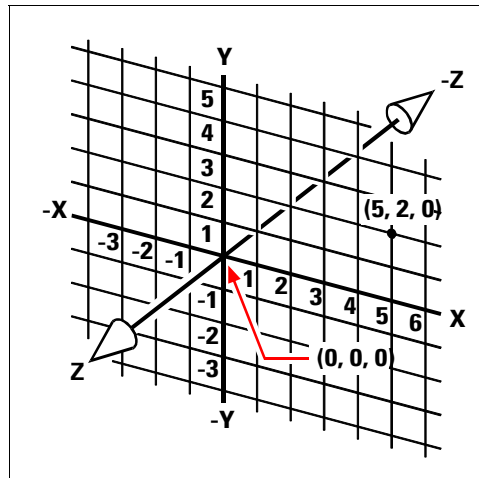
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Using Cartesian coordinates

Many commands in PTCAD require that you specify points as you draw or modify entities. You can do so by selecting points with the mouse or by typing coordinate values in the command bar. The program locates points in a drawing using a Cartesian coordinate system.

Understanding how coordinate systems work

The Cartesian coordinate system uses three perpendicular axes — x , y , and z — to specify points in three-dimensional space. Every location in a drawing can be represented as a point relative to a $0,0,0$ coordinate point, referred to as the origin. To draw a two-dimensional entity, you specify horizontal coordinate positions along the x -axis and vertical coordinate positions along the y -axis. Thus, every point on a plane can be represented as a coordinate pair composed of an x -coordinate and a y -coordinate. Positive coordinates are located above and to the right of the origin; negative coordinates are located to the left and below the origin.



The three perpendicular axes of the Cartesian coordinate system. When you work in two dimensions, you need enter only the x - and y -coordinates; the program assumes that the z -axis value is always the current elevation. When you work in three dimensions, however, you must also specify the z -axis value. When you look at a plan view of your drawing (a view from above, looking down), the z -axis extends straight up out of the screen at a 90-degree angle to the xy plane. Positive coordinates are located above the xy plane, and negative coordinates are below the plane.

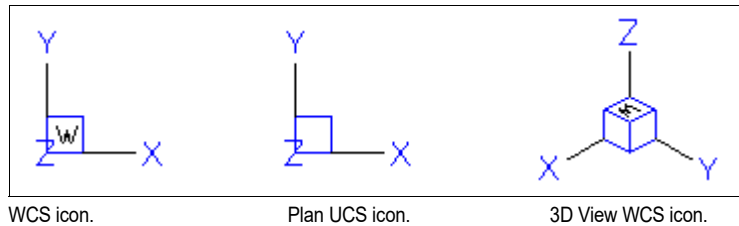
All PTCAD drawings use a fixed coordinate system, called the World Coordinate System (WCS), and every point in a drawing has a specific x,y,z -coordinate in the WCS. You can also define arbitrary coordinate systems located anywhere in three-dimensional space. These are called user coordinate systems and can be located anywhere in the WCS and oriented in any direction.

You can create as many user coordinate systems as you want, saving or redefining them to help you construct three-dimensional entities. By defining a UCS within the WCS, you can simplify the creation of most three-dimensional entities into combinations of two-dimensional entities.

To help you keep your bearings in the current coordinate system, the program displays a coordinate system icon. When you begin a new drawing, you are automatically in the WCS, indicated by the letter *W* in the icon. When you display a drawing in plan view, you see the coordinate system icon from the top, with the z -axis directed straight toward you. When you display a three-dimensional drawing in a view other than plan view, the coordinate system icon changes to reflect your new viewpoint.

The visible portions of the axes are the positive directions.

The invisible portions are the negative directions.



The PTCAD UCS icon looks different from the UCS icon in AutoCAD, because it presents more information. Three colors represent the three axes, making it easier for you to recognize the orientation in three-dimensional space:

- x -axis: red
- y -axis: green
- z -axis: blue


If you prefer a single color for the cursor and UCS icon, you can make that change with the *config* or *options* command.

Changing how coordinates display

The current position of the cursor is displayed as x,y,z-coordinates in the status bar and, by default, updates dynamically as you move the cursor. There are several display modes to choose from in PTCAD:

- Absolute for points — Updates the display with absolute coordinates only when you specify a point.
- Absolute continuous — Updates the display with absolute coordinates as you move the cursor and make selections. This is the default.
- Relative continuous — The same as absolute always when moving the cursor and making selections, with the addition of displaying relative distances and angles (rather than x,y,z-coordinates) when you have previously selected a point or entity and the program displays a rubber-band line. If no command is active, absolute coordinates display.
- Geographic continuous — Updates the display with geographic coordinates as you move the cursor and make selections.

To change how coordinates display in the status bar

- 1 Do one of the following to choose Drawing Settings ():
 - On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
 - On the menu, choose Tools > Drawing Settings.
 - On the Tools toolbar, click the Drawing Settings tool.
 - Type *dsettings* and then press Enter.
- 2 Click the Display tab.

Use a shortcut.

In the status bar, right-click Drawing Coordinates and choose Coordinate Settings to open the Drawing Setting > Display tab.

- 3 Choose one of the following:
 - Absolute coordinates update only when specifying points
 - Absolute coordinates update continuously
 - Relative polar coordinates update continuously
 - Geographic coordinates update continuously
- 4 Click OK.


Use a shortcut.

*On the status bar, click Drawing Coordinates to switch to the next display mode, or press **Ctrl+I**.*

Finding the coordinates of a point

To find the x,y,z-coordinates for a point on an entity, such as the endpoint of a line, select an appropriate entity snap (such as Endpoint) before selecting the entity. If you have no entity snaps set, the x,y-coordinates of the point you specified is displayed, with the z-coordinate equal to the current elevation.

To find the coordinate of a point in the drawing

- 1 Do one of the following to choose ID Coordinates ():
 - On the ribbon, choose Tools > ID Coordinate (in Inquiry).
 - On the menu, choose Tools > Inquiry > ID Coordinates.
 - On the Inquiry toolbar, click the ID Coordinates tool.
 - Type *idpoint* and then press Enter.
- 2 Select the point for which you want to find the coordinates.

If the command bar is activated, the x,y,z-coordinates for the point you selected display in the command bar.

If the command bar is not activated, the Prompt History window displays, showing the x,y,z-coordinates for the point you selected.

Using two-dimensional coordinates

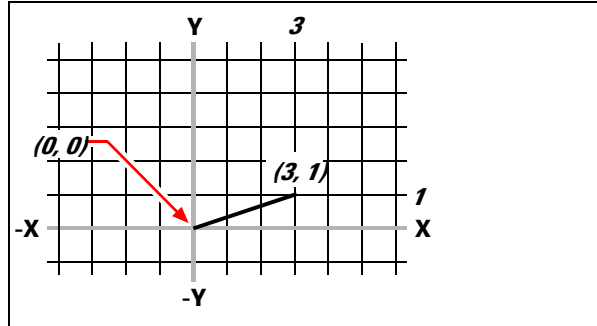
When working in two dimensions, you specify points on the xy plane. You can specify any point as an absolute coordinate (or Cartesian coordinate), using the exact x-coordinate and y-coordinate locations in relation to the origin (the 0,0 coordinate point at which the two axes intersect), or as a relative coordinate in relation to the previous point. You can also specify points using relative or absolute polar coordinates, which locate a point using a distance and an angle.

Entering absolute Cartesian coordinates

To enter absolute Cartesian coordinates, type the coordinate location of the point in the command bar. For example, to use absolute Cartesian coordinates to draw a line from the origin (0,0) to a point 3 units to the right and 1 unit above the origin, start the Line command and respond to the prompts as follows:

```
Start of line: 0,0
```

```
Angle • Length • <Endpoint>: 3,1
```



Drawing a line using the absolute Cartesian coordinate method.

When using absolute Cartesian coordinates, you need to know the exact point locations for anything you draw. For instance, to use absolute Cartesian coordinates to draw an 8.5-unit square with its lower left corner at 4,5, you must determine that the upper left corner is at coordinate 4,13.5, the upper right corner at 12.5,13.5, and the lower right corner at 12.5,5.

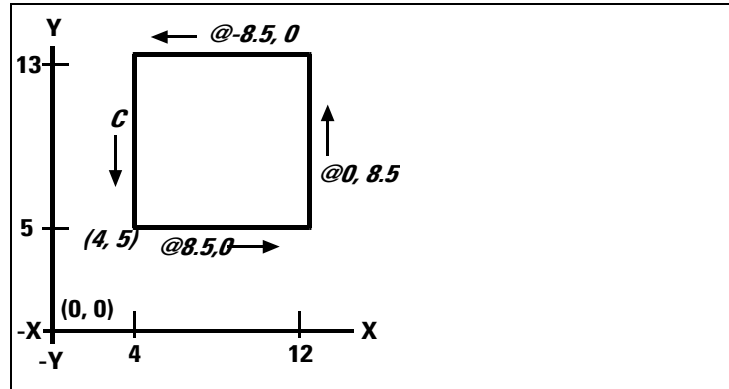
Entering relative Cartesian coordinates

Another, simpler method is to use relative Cartesian coordinates: you specify a location in the drawing by determining its position relative to the last coordinate you specified. To use relative Cartesian coordinates, type the coordinate values in the command bar, preceded by the *at* symbol (@). The coordinate pair following the @ symbol represents the distance along the x-axis and the y-axis to the next point. For example, to draw an 8.5-unit square with its lower left corner at 4,5 using relative Cartesian coordinates, start the Line command, and then respond to the prompts as follows:

```

Start of line: 4,5
Angle • Length • <Endpoint>: @8.5,0
Angle • Length • Follow • Undo • <Endpoint>: @0,8.5
Angle • Length • Follow • Close • Undo • <Endpoint>: @-8.5,0
Angle • Length • Follow • Close • Undo • <Endpoint>: C

```



Drawing a square using the relative Cartesian coordinates method; enter C to close.

The first relative coordinate ($@8.5,0$) locates the new point 8.5 units to the right (along the x-axis) from the previous point of 4,5; the second relative coordinate ($@0,8.5$) locates the next point 8.5 units above (along the y-axis) the previous point, and so on. Entering C (for Close) draws the final line segment back to the first point specified when you started the Line command.

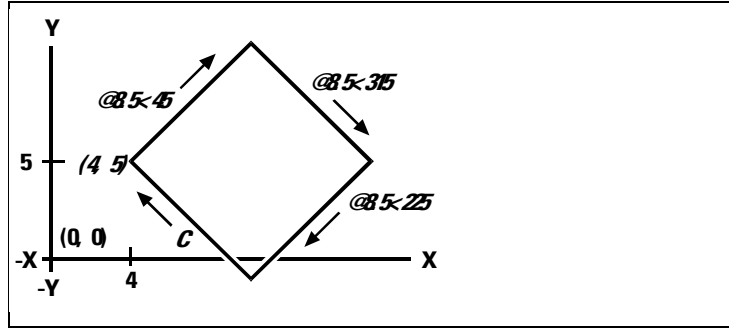
Entering polar coordinates

Using relative polar coordinates makes drawing a square tilted at a 45-degree angle a simple task. Polar coordinates base the location of a point on a distance and angle from either the origin (absolute coordinate) or from the previous point (relative coordinate).

To specify polar coordinates, type a distance and an angle, separated by the open angle bracket (<). For example, to use relative polar coordinates to specify a point 1 unit away from the previous point and at an angle of 45 degrees, type $@1<45$.

To draw the square from the example in the previous section, “Entering relative Cartesian coordinates,” this time tilted at a 45-degree angle, start the Line command, and then respond to the prompts as follows:

```
Start of line: 4,5
Angle • Length • <Endpoint>: @8.5<45
Angle • Length • Follow • Undo • <Endpoint>: @8.5<315
Angle • Length • Follow • Close • Undo • <Endpoint>: @8.5<225
Angle • Length • Follow • Close • Undo • <Endpoint>: C
```



Drawing a tilted square using the relative polar coordinates method; enter C to close.

This example assumes the program's default settings.

Like all examples in this guide, the example assumes default settings: Angles increase counterclockwise and decrease clockwise. An angle of 315 degrees is the same as -45 degrees.

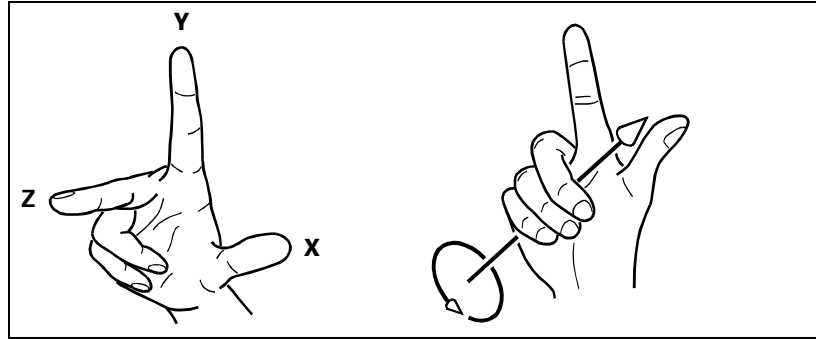
Using three-dimensional coordinates

Specifying coordinates in three-dimensional space is similar to working in two dimensions, except that you also use the z-axis to locate coordinates. Three-dimensional coordinates are represented in the format x,y,z (for example, 2,3,6).

Using the right-hand rule

To visualize how PTCAD works with three-dimensional space, use a technique known as the right-hand rule. Hold up your right hand in a loose fist with your palm facing you. Extend your thumb in the direction of the positive x-axis and your index finger upward in the direction of the positive y-axis. Then extend your middle finger straight toward you in the direction of the z-axis. These three fingers are now pointing in the positive x, y, and z directions, respectively.

You can also use the right-hand rule to determine the positive rotation direction. Point your thumb in the positive direction of the axis about which you want to rotate, and then curl the rest of your fingers toward your palm. These fingers are curling in the positive rotation direction.



The right-hand rule helps you determine the positive direction of the x-, y-, and z-axes and the positive rotation direction.

Entering x,y,z-coordinates

When working in three dimensions, you can specify x,y,z-coordinates as absolute distances in relation to the origin (the 0,0,0 coordinate point at which the three axes intersect) or as relative coordinates based on the last point selected. For example, to specify a point 3 units along the positive x-axis, 4 units along the positive y-axis, and 2 units along the positive z-axis, specify the coordinate 3,4,2.

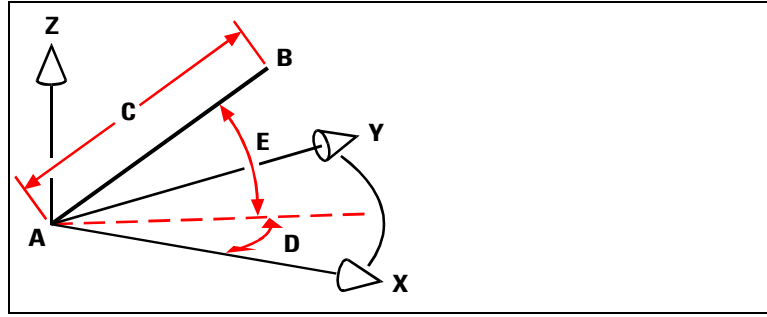
Entering spherical coordinates

When working in three-dimensional space, you can use spherical coordinates to specify a three-dimensional point by entering its distance from either the origin (absolute distance) or the last point (relative distance), along with its angle in the xy plane and its angle up from the xy plane. In spherical format, you separate each angle with the open angle bracket (<).

Thus, to draw a line from the origin to a point 10.2500 drawing units away, at an angle of 45 degrees from the x-axis and 35 degrees from the xy plane, start the Line command, and then respond to the prompts as follows:

Start of line: 0,0,0

Angle • Length • <Endpoint>: 10.2500<45<35



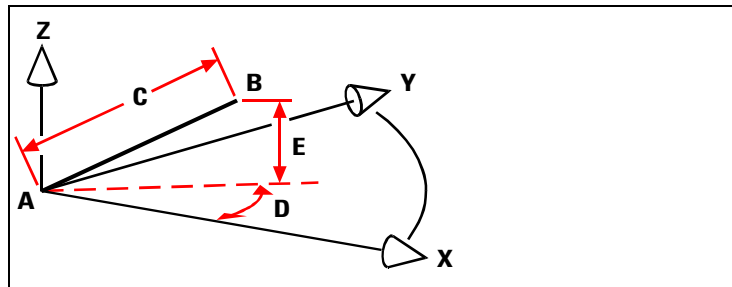
When you draw a line from a start point (A) to an endpoint (B) using spherical coordinates, you specify its length (C, in this case 10.2500 units), the angle in the xy plane (D, in this case 45 degrees), and the angle from the xy plane (E, in this case 35 degrees).

Entering cylindrical coordinates

When working in three-dimensional space, you can also use cylindrical coordinates to specify a three-dimensional point. You specify a point by entering its distance from either the origin (absolute distance) or the last point (relative distance), its angle in the xy plane, and its z-coordinate value.

In cylindrical format, you separate the distance and angle with the open angle bracket (<) and separate the angle and z value with a comma. For example, to draw a line from the last point to a point 7.4750 units away, at an angle of 27 degrees from the x-axis in the xy plane and 3 units up in the z direction, start the Line command, and then respond to the prompts as follows:

```
Start of line: (select point A)
Angle • Length • <Endpoint>: @7.4750<27,3
```



When you draw a line from a start point (A) to an endpoint (B) using cylindrical coordinates, you specify its length (C, in this case 7.4750), the angle in the xy plane (D, in this case 27 degrees), and the distance in the z direction (E, in this case 3 units).

Using xyz point filters

Point filters provide a method of locating a point in a drawing relative to another point without specifying the entire coordinate. Using a point filter, you can enter partial coordinates, and then the program prompts you for the remaining coordinate information. To use xyz point filters, respond to the prompt for a coordinate with a filter in the following form:

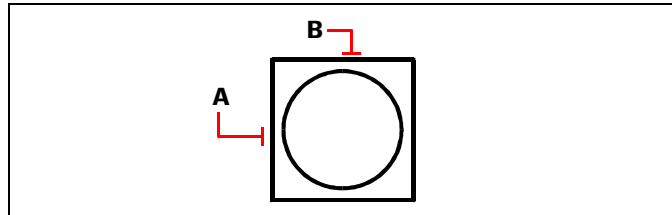
.coordinate

where *coordinate* is one or more of the letters x, y, and z. The program then prompts you for the filtered coordinate(s). For example, if you type *.xy*, the program prompts you to select a point whose xy-coordinate you want, and then prompts you for the z-coordinate. The filters *.x*, *.y*, *.z*, *.xy*, *.xz*, and *.yz* are all valid filters.

Using point filters in two dimensions

You can use point filters when you work in two dimensions to locate points in relation to existing entities. For example, to draw a circle centered in a rectangle, start the Circle command, and then respond to the prompts as follows:

```
2Point • 3Point • RadTanTan • Arc • Multiple • <Center of circle>: .y
Select Y of: mid
Snap to midpoint of: (select the left side of the rectangle)
Still need XZ of: mid
Snap to midpoint of: (select top of the rectangle)
Diameter • <Radius>: (specify radius of circle)
```

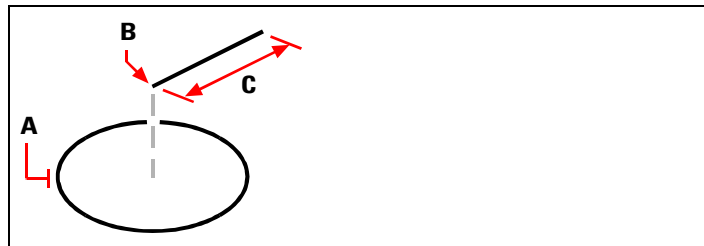


You can use point filters to center the circle by separately selecting the midpoints of two sides of the rectangle (A and B) and then specifying its radius.

Using point filters in three dimensions

You can use point filters when you work in three-dimensional space to locate points in two dimensions and then specify the z-coordinate as the elevation above the xy plane. For example, to begin drawing a line from a point with a z-coordinate 3 units above the center of a circle, insert the circle, and then start the Line command and respond to the prompts as follows:

```
ENTER to use last point • Follow • <Start of line>: .xy
Select XY of: cen
Snap to centerpoint of: (select a point on the circle)
Still need Z of: 3 (locates the starting point 3 units above the center of
the circle)
Length of line: (specify the length of the line)
```



You can use point filters to draw a line by first selecting a point in the xy plane (A), specifying the z-coordinate (B), and then specifying the length of the line (C).

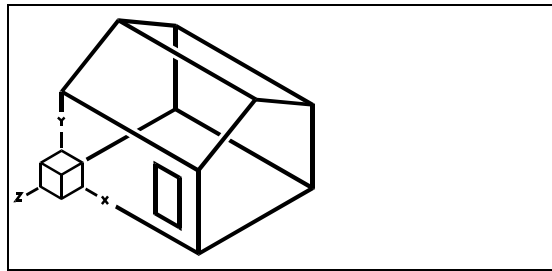
Defining user coordinate systems

A user coordinate system (UCS) is a Cartesian coordinate system with origins and orientation defined by the user.

Understanding user coordinate systems

When working in three-dimensional space, you can define a UCS with its own 0,0,0 origin and orientation separate from the WCS. You can create as many user coordinate systems as you want, and then save and recall them as you need them to simplify construction of three-dimensional entities.

For example, you can create a separate UCS for each side of a building. Then, by switching to the UCS for the east side of the building, you can draw the windows on that side by specifying only their x- and y-coordinates. When you create one or more user coordinate systems, the coordinate entry is based on the current UCS.



UCS aligned with the front wall of the house.



Defining a user coordinate system

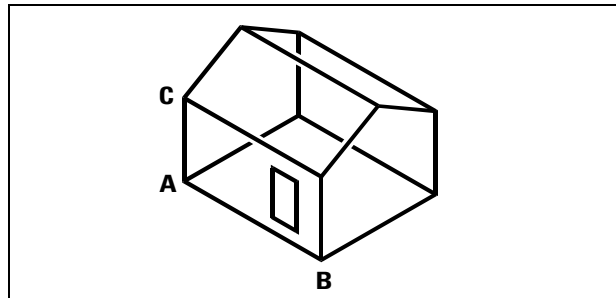
To define a UCS, you can use any of the following methods:

- Specify a new origin and points on the positive x- and y-axes.
- Specify a new origin and point on the positive z-axis.
- Align the UCS with an existing entity.
- Rotate the current UCS around any of its axes.
- Align the UCS with its z-axis parallel to the current viewing direction.
- Align the UCS xy plane perpendicular to the current view.

When you define a new UCS, the UCS icon changes to indicate the origin and orientation of the new UCS.

To define a UCS by specifying a new origin and points on the positive x- and y-axes

- 1 Do one of the following to choose User Coordinate Systems ():
 - On the ribbon, choose Draw > User Coordinate Systems or choose View > User Coordinate Systems.
 - On the menu, choose Tools > User Coordinate Systems > Select a Predefined UCS.
 - On the UCS toolbar, click the Select a Predefined UCS tool.
 - Type *setucs* and then press Enter.
- 2 In the User Coordinate Systems dialog box, click Explore UCSs.
- 3 In the Explorer, be sure that Coordinate Systems is selected, and click the New Item tool ().
- 4 In the prompt box, choose 3 Point.
- 5 Select the new origin.
- 6 Select a point on the positive x-axis.
- 7 Select a point in the positive y direction.
- 8 In the Explorer - Coordinate Systems element, type the name for the new UCS, and then close the dialog box.



Define the new UCS by selecting the origin (A), a point on the positive x-axis (B), and a point in the positive y direction (C).


Using a preset user coordinate system

PTCAD lets you select a preset UCS. The six planes defined by looking along the x, y, z-axes align the UCS with the top, left, front, bottom, right, or back, based on either the WCS or the current UCS in effect when you select the tool. You can also select the previous UCS, align the UCS to the current view, or select the WCS.

When you select a UCS, the cursor orientation and UCS icon change to reflect the new UCS. The display does not change, however, unless you select the Change View To Plan View Of The Selected UCS check box.

After you align the UCS to a preset UCS, you can use PTCAD's Explorer to save the UCS. To do this, in the Explorer window, choose Edit > New > UCS, and then select Current.

To select a preset UCS

- 1 Do one of the following to choose User Coordinate Systems (- On the ribbon, choose Draw > User Coordinate Systems or choose View > User Coordinate Systems.
- On the menu, choose Tools > User Coordinate Systems > Select a Predefined UCS.
- On the UCS toolbar, click the Select a Predefined UCS tool.
- Type *setucs* and then press Enter.
- 2 Under Set The Selected UCS Relative To, select either Current UCS to change to the new UCS by reorienting relative to the current UCS or World Coordinate System (WCS) to base the new UCS orientation on the WCS.
- 3 Under Select UCS, click the button corresponding to the UCS you want as the new current UCS.

Preset UCSs are also available with a dialog box.

Choose Format > User Coordinate Systems, then select a predefined UCS from the list. The preset UCSs are also available directly on the UCS toolbar.

Working with the Explorer

PTCAD's Explorer provides a powerful and convenient way to maintain and manage many settings of your drawings. You can use the Explorer to work with numerous elements within the current drawing or to copy this information between drawings. This section explains how to use the Explorer to:

- Organize information on layers, manage layers, and work with layer states.
- Create and use linetypes and multiline styles.
- Load text fonts and create text styles.
- Create and modify table styles.
- Work with named views, visual styles, and coordinate systems.
- Copy, cut, and paste dimension styles between drawings.
- Save, insert, and manage blocks, groups, and external references.
- Work with layouts and materials.

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Using the Explorer

PTCAD's Explorer allows you to view, create, copy, and edit layers, layer states, linetypes, text styles, coordinate systems, blocks, dimension styles, and more within your drawings. You can also use the Explorer to copy these elements from one open drawing to another. The Explorer is a great way to manage and maintain your drawings.

Displaying the Explorer

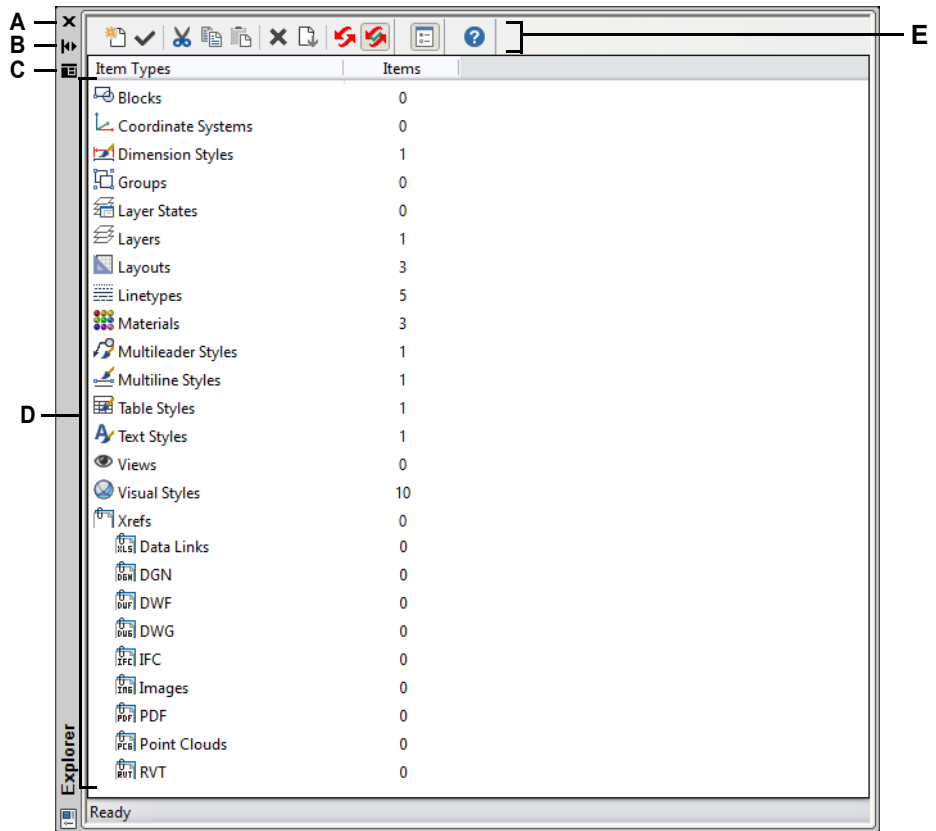
The Explorer opens in its own, separate pane or window, which you can move or resize. The Explorer has its own menu and tools.

To display the Explorer

Do one of the following to choose Explorer ():

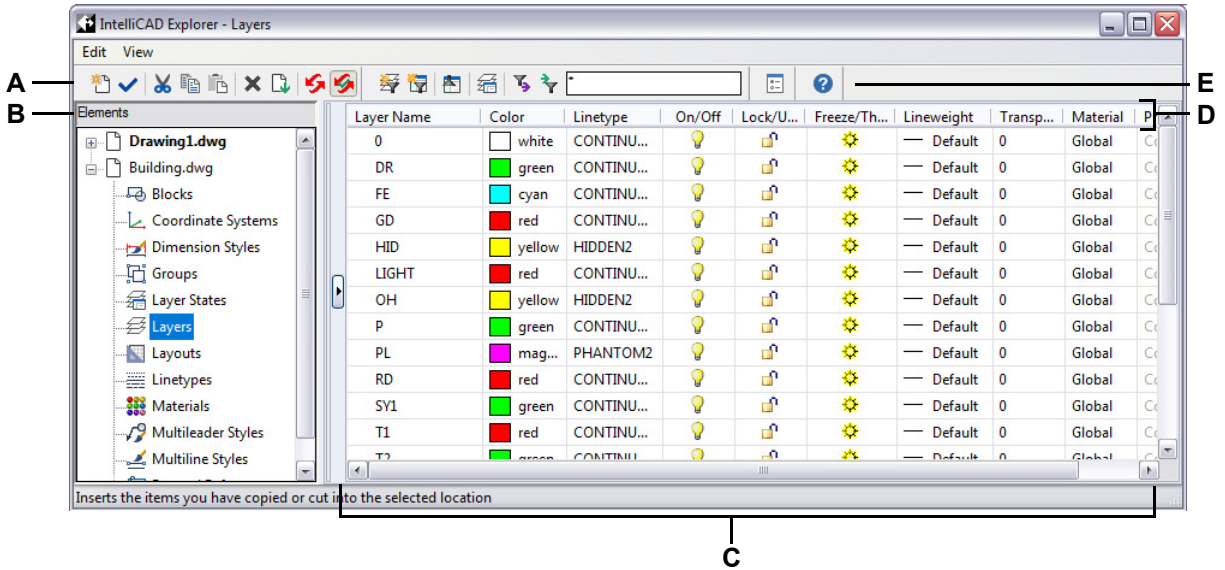
- On the ribbon, choose Tools > Explorer (in Explorer) or View > Explorer (in Panes).
- On the menu, choose View > Display > Explorer or Tools > Explorer > Explore Layers (or any other item).
- On the Tools toolbar, click the Explorer tool.
- Type *explorer* and then press Enter.

By default, the Explorer displays as a pane.



- A** Click to close the Explorer pane.
- B** Click to turn auto-hide on or off for the pane. Available when the Explorer is an undocked pane.
- C** Click to choose the position and transparency of the pane. (Available when an undocked pane.)
- D** Lists the elements you can control. Double-click an item to open its pane.
- E** Tools on the Explorer toolbar provide controls common to all elements.

You can click the Show as Pane tool to display the Explorer as a dialog box instead, which allows you to view the elements for each open drawing. The Explorer displays elements, such as layers, layer states, linetypes, etc., on the left and corresponding drawing settings on the right.







- A Tools on the Explorer toolbar vary depending on the element.
- B The Elements pane shows the name of every drawing currently open and lists the elements you can control in each drawing.
- C Click a setting to change it.
- D The named settings for the selected element. Click a column title to sort by category. Click and drag a column title to change the column order. Right-click a column title to choose which columns display or to restore all column settings to the default settings.
- E Other tools appear, depending on the Elements pane selection.

Using the Explorer, you can create, delete, or modify any of the settings for the currently selected element for a given drawing. You can also copy the contents of any element from one drawing to another. The common tools and menu items in the Explorer provide the following functions:

Common Explorer tools

Tool	Function
New Item	Creates a new layer, linetype, text style, coordinate system, view, block, external reference, or dimension style.
Current	Makes the selection current.
Cut	Cuts the selection to the Clipboard.
Copy	Copies the selection to the Clipboard.
Paste	Pastes the selection from the Clipboard into the appropriate list of a different drawing.
Delete	Deletes the selection from the list.
Purge	Eliminates unreferenced elements from your drawing file.




Common Explorer tools

Tool	Function
 Regen	Recalculates the display for the current window.
 On/Off Regen	Turns on and off the display recalculation.
 Show as Dialog (or Show as Pane)	Displays the Explorer as a pane or a dialog box.
 Help	Displays online help for the Explorer.

Copying settings

A particularly powerful feature of the Explorer is its capability of copying many of the settings — layers, linetypes, text styles, coordinate systems, views, blocks, or dimension styles — from one drawing to another. If you have more than one drawing open, the Explorer makes it easy to reuse information. For example, when you copy layers from one drawing to another, the layer names as well as their linetypes, colors, and other settings are also copied, but not the entities on those layers.

To copy layers from one open drawing to another open drawing

- Do one of the following to choose Explore Layers ():
 - On the ribbon, choose Home > Explore Layers (in Layers) or choose Tools > Layers (in Explorer).
 - On the menu, choose Format > Explore Layers or choose Tools > Explorer > Explore Layers.
 - On the Explorer toolbar, click the Explore Layers tool.
 - Type *explayers* and then press Enter.
- In the Elements pane, select the drawing from which you want to copy layers.
- If necessary, click the plus (+) symbol to expand the Elements list for the drawing, and then click Layers.
- In the Layers Settings In Drawing list (right pane), select the layers you want to copy.
- Choose Edit > Copy, or click the Copy tool ().
- In the Elements pane, select the drawing to which you want to copy the layers.
- If necessary, click the plus (+) symbol to expand the Elements list for the drawing, and then click Layers.
- Choose Edit > Paste, or click the Paste tool ().



Deleting settings

You can use the Explorer to delete many of the items that appear in the Elements list. You can delete a layer, linetype, text style, coordinate system, view, block, or dimension style.

Because you may have already created entities on a particular layer or using a particular linetype or text style, deleting one of these elements requires that you make certain choices from options the program presents. For example, if you attempt to delete a layer, the program prompts you to specify whether you want to move any entities from that layer to another layer. Every drawing has at least one layer, the default layer, named “0.” You cannot delete or rename this layer. Your drawing can also contain an unlimited number of additional layers, each of which you assign a unique name.

If you try to delete a linetype, the program prompts you to specify whether you want to convert all entities drawn using that linetype to a different linetype. If you attempt to delete a text style, the program prompts you to specify whether you want to convert all text entities created using that style to a different style.

To delete a layer and relocate its entities to another layer

- 1 Do one of the following to choose Explore Layers (- On the ribbon, choose Home > Explore Layers (in Layers) or choose Tools > Layers (in Explorer).
- On the menu, choose Format > Explore Layers or choose Tools > Explorer > Explore Layers.
- On the Explorer toolbar, click the Explore Layers tool.
- Type *explayers* and then press Enter.
- 2 From the Layer Name list, select the layer that you want to delete.
If that layer is the current layer, layer 0 automatically becomes the current layer.
- 3 Click the Delete tool (.
- 4 From the dialog box that displays, click the Change option and then select the layer to which you want to relocate entities.



Delete a layer and its entities.

In the command bar, type LAYDEL, then select an entity that is assigned the layer you want to delete. Using this method also deletes all entities assigned the layer.

Purging elements

From within the Explorer, you can eliminate unused blocks, layers, linetypes, text styles, dimension styles, or annotation scales from your drawing file. Purging unused elements can significantly reduce the drawing file size.

To purge an element

- 1 Do one of the following to choose Explorer (- On the ribbon, choose Tools > Explorer (in Explorer).
- On the menu, choose Tools > Explorer.
- On the Tools toolbar, click the Explorer tool.
- Type *explayers* and then press Enter.
- 2 Select the element from which you want to purge unreferenced elements.
- 3 Choose Edit > Purge, or click the Purge tool (.

Organizing information on layers

Understanding layers

Layers in PTCAD are like the transparent overlays you use in manual drafting. You use layers to organize different types of drawing information. In PTCAD, each entity in a drawing exists on a layer. When you draw an entity, it is created on the current layer.

You can control the visibility of layers in individual viewports. When you turn a layer off, entities drawn on that layer are no longer visible, and they do not print. Although a layer may be invisible, you can still select it as the current layer, in which case new entities are also invisible until you turn the layer back on. Entities on invisible layers can also affect the display and printing of entities on other layers. For example, entities on invisible layers can hide other entities when you use the Hide command to remove hidden lines.

You can also freeze and thaw layers, and control whether layers are frozen by default in newly created layout viewports. Entities drawn on frozen layers do not display, do not print, and do not regenerate. When you freeze a layer, its entities do not affect the display or printing of other entities. For example, entities on frozen layers do not hide other entities when you use the Hide command to remove hidden lines. In addition, you cannot draw on a frozen layer until you thaw it, and you cannot make a frozen layer current.

You cannot freeze the current layer. If you attempt to freeze the current layer, a dialog box appears prompting you to specify a different layer. You also cannot freeze or thaw a viewport layer unless you are working in a Layout tab.

You can lock or unlock layers. The entities on a locked layer are still visible and will print, but you cannot edit them. Locking a layer prevents you from accidentally modifying entities. You can also control whether entities on layers can be cut by section planes, which is helpful if you use section planes to visualize models.



You can control whether entities on each layer are included or omitted when you print the drawing.

Each layer has its own properties, such as color, linetype, lineweight, transparency, print visibility, and more. For drawings that use named print styles, layers can also have their own print style. Entities you draw on a particular layer are displayed in the color, linetype, and lineweight associated with that layer unless you override these settings. You control all of the associated settings for layers using the Layers element in the Explorer. You can also access layer settings for entities using the Entity Properties toolbar.

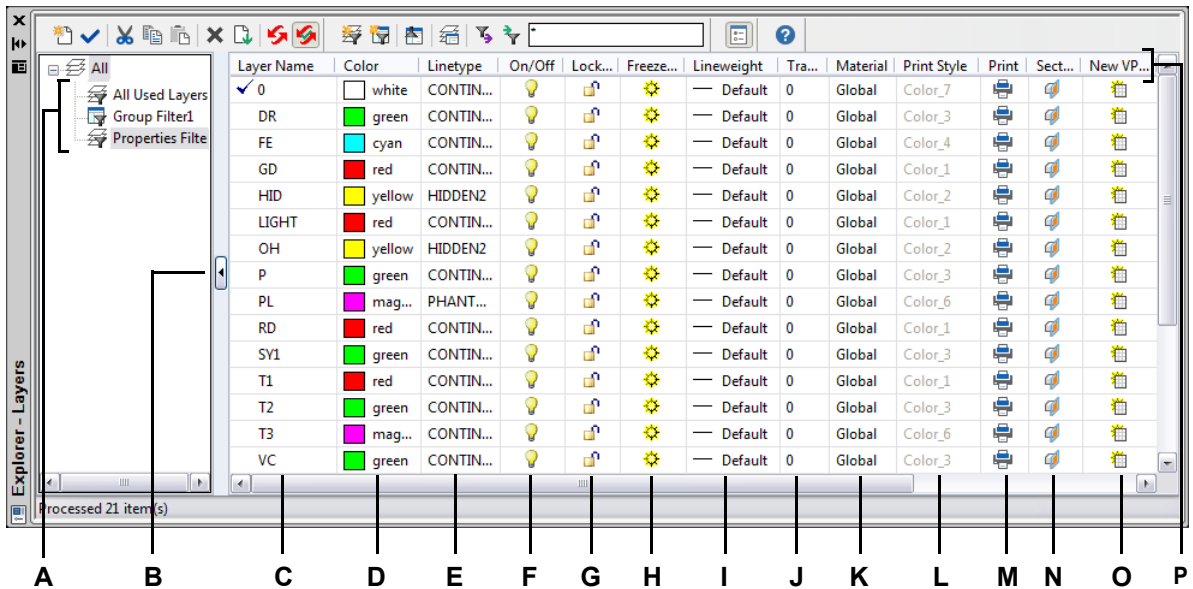
Some drawings contain large lists of layers, in which case you can search for layers by name, or you can organize layers into subsets using layer filters. Layer states are also useful for drawings that contain many layers. With layer states, you can assign properties to individual layers and save them in a layer state, then apply those settings at any time.

Displaying layer information in the Explorer

To display layer information

- 1 Do one of the following to choose Explore Layers (- On the ribbon, choose Home > Explore Layers (in Layers) or choose Tools > Layers (in Explorer).
- On the menu, choose Format > Explore Layers or choose Tools > Explorer > Explore Layers.
- On the Tools toolbar, click the Explorer tool (.
- On the Explorer toolbar, click the Explore Layers tool.
- Type *explayers* and then press Enter.
- 2 If desired, view information for only certain layers by doing one of the following:
 - On the toolbar, type the name of the desired layers, including wildcards, and press Enter.
 - In the list of layer filters, select All Used Layers to display only those layers that are used in the drawing.
 - In the list of layer filters, select an existing group filter or properties filter.

For details about filtering and searching layers, see “Filtering and finding layers” on page 279 in this chapter.





- A** Select to display all layers, all layers that are in use, or a filtered subset.
- B** Click to collapse or expand the display of layer filters.
- C** Lists named layers in the current drawing. The list contains all layers or a subset if viewing by filter or only used layers. A check mark indicates the current layer.
- D** Displays the color assigned to each layer.
- E** Indicates the linetype assigned to each layer.
- F** Indicates the visibility status of each layer.
- G** Indicates the locked or unlocked status of each layer.
- H** Indicates the frozen or thawed status of each layer for all viewports.
- I** Indicates the lineweight assigned to each layer.
- J** Indicates the transparency percentage assigned to each layer.
- K** Indicates the material assigned to each layer.
- L** Indicates the print style assigned to each layer.
- M** Indicates the print status of each layer.
- N** Indicates whether entities on each layer can be cut by section planes.
- O** Indicates whether each layer is frozen by default in newly created layout viewports.
- P** Click a column title to sort by category. Click and drag a column title to change the column order. Right-click a column title to choose which columns display or to restore all column settings to the default settings.

Creating and naming layers


You can create an unlimited number of layers in every drawing and use those layers for organizing information. When you create a new layer, it is initially assigned the color white (or black, depending on your system settings) and the linetype CONTINUOUS. By default, a new layer is also visible. After you create and name a layer, you can change its color, linetype, visibility, and other properties.

To create a new layer

- 1 Do one of the following to choose Explore Layers (- On the ribbon, choose Home > Explore Layers (in Layers) or choose Tools > Layers (in Explorer).
- On the menu, choose Format > Explore Layers or choose Tools > Explorer > Explore Layers.
- On the Explorer toolbar, click the Explore Layers tool.
- Type *explayers* and then press Enter.
- 2 Do one of the following:
 - Choose Edit > New > Layer.
 - Click the New Item tool (.

A new layer is added to the Layer Name list, with the default name NewLayer1.
- 3 Type a name for the new layer over the highlighted default name, and then press Enter.
- 4 To complete the command and return to your drawing, close the window.

To change a layer name in the current drawing

- 1 Do one of the following to choose Explore Layers (- On the ribbon, choose Home > Explore Layers (in Layers) or choose Tools > Layers (in Explorer).
- On the menu, choose Format > Explore Layers or choose Tools > Explorer > Explore Layers.
- On the Explorer toolbar, click the Explore Layers tool.
- Type *explayers* and then press Enter.
- 2 In the Layer Name list, select the layer you want to rename.
- 3 Do one of the following:
 - Choose Edit > Rename, type a new name, and then press Enter.
 - Highlight the layer name you want to change, type a new name, and then press Enter.
 - Right-click the layer name you want to change, and from the shortcut menu, select Rename, type a new name, and then press Enter.
- 4 To complete the command and return to your drawing, close the window.

The layer named “0” is reserved.

You cannot rename it.

Filtering and finding layers

Some drawings contain large lists of layers. To limit the layers that appear in the list, you can search layers by name (including wildcards) and you can also create layer filters.

By default, there are two predefined filters. One displays all layers and the other displays all layers that are used in the drawing.

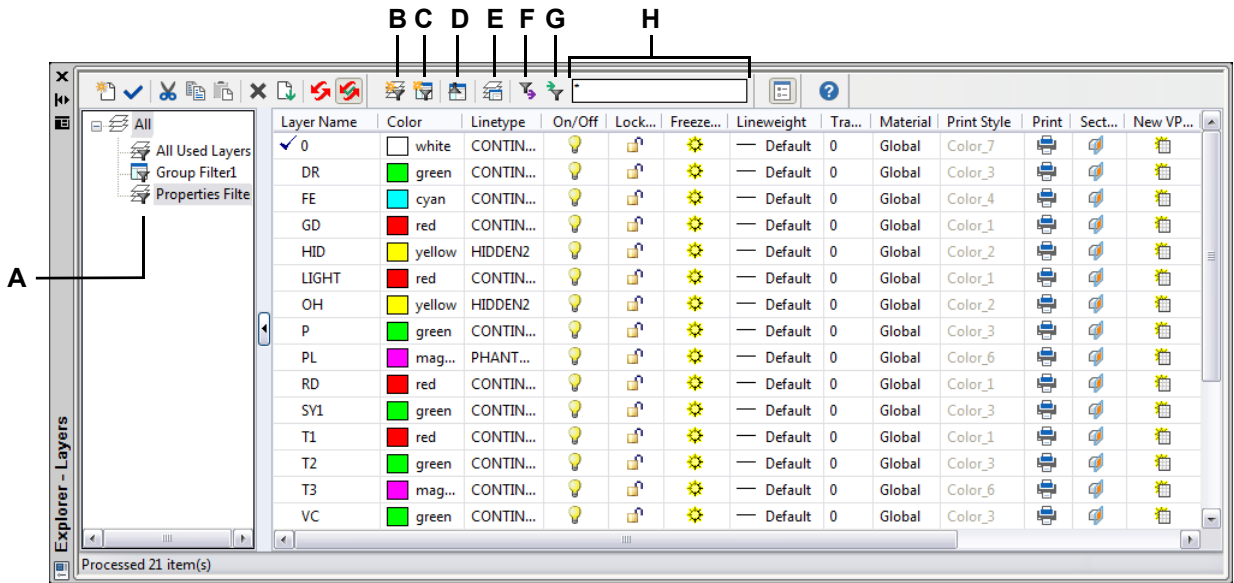
There are two types of layer filters that you can create:

- **Properties filter** — Includes layers according to properties that you specify. This filter is dynamic, that is, if a new layer meets the requirements for the property filter, it is automatically added to the filter.
- **Group filter** — Includes layers that you include in the group. This filter is static, that is, new layers are handled manually.

After you create a layer filter, you can turn all of its layers on or off, thaw or freeze the layers, and lock or unlock the layers. Layer filters can also be inverted, imported, and exported.

Layer filters can affect several lists of layers.

In addition to the list of layers in the Explorer, layer filters can also be applied to the layer controls on the ribbon Home tab and Quick Access toolbar (if the ribbon is available in your program version), Layer Tools toolbar, and Entity Properties toolbar. Click Apply Layer Filter to Layer Control to change whether the current layer filter selected in the Explorer is applied to these layer lists.



A Displays the list of layer filters.

B Click to create a new layer property filter.

C Click to create a new layer group filter.

D Click to invert the currently selected layer filter.

E Click to display the Layer States Manager.


F Click to export the currently selected layer filter to an .lft file.

G Click to import a layer filter (.lft file).

H Type a search term, including wildcards, then press Enter to search layer names.

Searching layers by name



To search for layers by name

- 1 Do one of the following to choose Explore Layers (- On the ribbon, choose Home > Explore Layers (in Layers) or choose Tools > Layers (in Explorer).
- On the menu, choose Format > Explore Layers or choose Tools > Explorer > Explore Layers.
- On the Explorer toolbar, click the Explore Layers tool.
- Type *explayers* and then press Enter.
- 2 In the Explorer toolbar, type the search name, including the following wildcards:
 - * Matches any character or series of characters.
 - ? Matches a single character.
 - # Matches any numerical character.
 - @ Matches any alphabetic character.
 - . Matches any character that is not alphabetic or numeric.
 - [] Matches any of the characters inside the brackets.
 - ~ Matches any characters except for those after the tilde.
 - [~] Matches any of the characters except for those inside the brackets.
 - [-] Matches a range of characters inside the brackets.
 - ' Matches the exact characters located after the quote mark. This is most often used when searching for layer names that include wildcards, for example, @Floor or Field#.
- 3 To view all layers again in the list, delete the search term.
- 4 Click OK.

Filtering layers by property

Layer property filters include layers according to properties that you specify. For example, a property filter can display all layers with a dashed linetype, then you can easily turn those layers on or off.

To create a properties filter

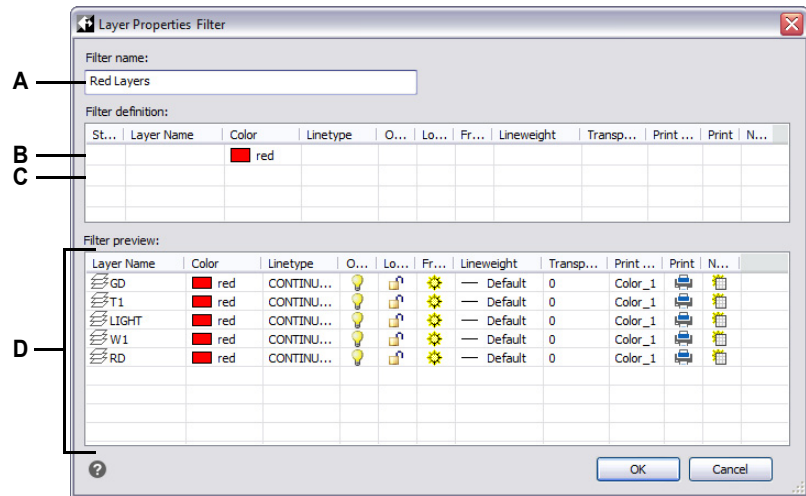
- 1 Do one of the following to choose Explore Layers (- On the ribbon, choose Home > Explore Layers (in Layers) or choose Tools > Layers (in Explorer).
- On the menu, choose Format > Explore Layers or choose Tools > Explorer > Explore Layers.
- On the Explorer toolbar, click the Explore Layers tool.
- Type *explayers* and then press Enter.
- 2 Click the Properties Filter tool () or press Alt + F.

Filters can be nested.

To create a sub-filter of an existing filter, right-click the existing filter and choose New Properties Filter or New Group Filter.

- 3 Type a name for the filter.
- 4 Do any of the following in Filter Definition:
 - Click the column of an empty row to specify the property to match. All layers that match the specified properties of any row will be included.
 - Click the column of an existing row to specify an additional property that a layer must match to be included in the filter. All layer properties in a single row must be matched for the layer to be included.
 - Right-click a row and choose Delete Row to delete a row you no longer need.
 - Right-click a row and choose Duplicate Row to make a copy of an existing row.

The filter preview displays the layers that will be included in the filter.
- 5 Click OK.
- 6 To complete the command and return to your drawing, close the window.



- A** Type a name of the new layer properties filter. **C** Click the column of an empty row to select the property to match.
- B** Click the column of an existing filter to specify an additional property that a layer must match to be included in the filter. **D** Displays the layers that match the settings of the layer properties filter.

To modify a properties filter

- In the Explorer, select Layers, then do one of the following:
 - To rename a properties filter, select it and type a new name.
 - To delete a properties filter, select it and press Delete.
 - To change the definition of a properties filter, double-click it.
 - To change the list order of filters, drag and drop a properties filter to a new location in the list. To sort all filters in alphabetical order, right-click the top item, All, and choose Sort Filters.

Use a shortcut.

Right-click a properties filter to modify it.

To modify the layers in a properties filter

- 1 In the Explorer, select Layers.
- 2 To change the visibility of all layers in a properties filter, right-click the properties filter, choose Visibility, then choose one of the following:
 - On / Off
 - Thaw / Freeze
 - Isolate Freeze



These additional options are available if you're working on a Layout tab:

 - Current Viewport Thaw / Freeze
 - Current Layout All Viewports Thaw / Freeze
 - Isolate Viewport Freeze
- 3 To change the accessibility of all layers in a properties filter, right-click the properties filter, choose Lock, then choose one of the following:
 - Lock
 - Unlock

Filtering layers by group

Layer group filters include any layer that you select for the group. For example, a group filter can display all layers that contain text entities, then you can freeze or thaw those layers as needed.

To create a group filter

- 1 Do one of the following to choose Explore Layers (- On the ribbon, choose Home > Explore Layers (in Layers) or choose Tools > Layers (in Explorer).
- On the menu, choose Format > Explore Layers or choose Tools > Explorer > Explore Layers.
- On the Explorer toolbar, click the Explore Layers tool.
- Type *explayers* and then press Enter.
- 2 Click the Group Filter tool () or press Alt + G.

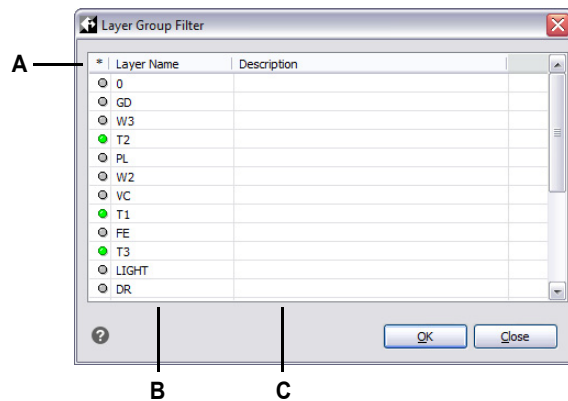
Filters can be nested.

To create a sub-filter of an existing filter, right-click the existing filter and choose *New Properties Filter* or *New Group Filter*.

- 3 Type a name for the new group filter, then press Enter.
- 4 To choose layers for the group filter, do one of the following:
 - Right-click the group filter, choose *Select Layers > Add*, then select entities in your drawing that reside on layers to include in the group. Press Enter when done. Choose *Select Layers > Replace* if the group filter has existing layers that you want to remove before adding new layers.
 - Double-click the group filter, then mark and unmark the desired layers.
- 5 Click OK.
- 6 To complete the command and return to your drawing, close the window.

Create a group filter by converting an existing property filter.

In the *Explorer*, view layer filters, right-click a property filter, and choose *Convert to Group Filter*.



- A** Click the circle green to include its layer in the layer group filter. Click again to exclude it.
- B** Displays layers in the drawing.
- C** Displays the layer description, if available.

To modify a group filter

- 1 In the Explorer, select Layers, then do one of the following:
 - To rename a group filter, select it and type a new name.
 - To delete a group filter, select it and press Delete.
 - To change the layers included in a group filter, double-click it.
 - To change the list order of filters, drag and drop a group filter to a new location in the list. To sort all filters in alphabetical order, right-click the top item, All, and choose Sort Filters.
 - To select new layers for the group filter directly in the drawing, right-click the group filter. Choose Select Layers > Add if you want to keep all existing layers in the group filter and select new ones to add directly in the drawing. Choose Select Layers > Replace if you want to remove all existing layers from the group filter and select new ones directly in the drawing.

Use a shortcut.

Right-click a group filter to modify it.

To modify the layers in a group filter

- 1 In the Explorer, select Layers.
- 2 To change the visibility of all layers in a group filter, right-click the group filter, choose Visibility, then choose one of the following:
 - On / Off
 - Thaw / Freeze
 - Isolate Freeze



These additional options are available if you're working on a Layout tab:

 - Current Viewport Thaw / Freeze
 - Current Layout All Viewports Thaw / Freeze
 - Isolate Viewport Freeze
- 3 To change the accessibility of all layers in a group filter, right-click the group filter, choose Lock, then choose one of the following:
 - Lock
 - Unlock

Inverting layer filters

Layer filters can be inverted, for example, you can invert the All Used Layers filter to display a list of all layers that are unused in the drawing.

To invert a layer filter

- 1 Do one of the following to choose Explore Layers (- On the ribbon, choose Home > Explore Layers (in Layers) or choose Tools > Layers (in Explorer).
- On the menu, choose Format > Explore Layers or choose Tools > Explorer > Explore Layers.
- On the Explorer toolbar, click the Explore Layers tool.
- Type *explayers* and then press Enter.
- 2 Select a layer filter.
- 3 Click the Invert Filter tool ().
- 4 To complete the command and return to your drawing, close the window.



The filter stays inverted until it's turned off.

If the contents of a filter look incorrect, it could be because it is inverted.



Importing and exporting layer properties filters

Layer properties filters can be imported and exported as .lst files.

To import layer properties filters

- 1 Do one of the following to choose Explore Layers (- On the ribbon, choose Home > Explore Layers (in Layers) or choose Tools > Layers (in Explorer).
- On the menu, choose Format > Explore Layers or choose Tools > Explorer > Explore Layers.
- On the Explorer toolbar, click the Explore Layers tool.
- Type *explayers* and then press Enter.
- 2 Click the Import Layer Filter tool ().
- 3 Navigate to where you want to save the layer filter, type a name, then click Save.
- 4 To complete the command and return to your drawing, close the window.



To export layer properties filters

- 1 Do one of the following to choose Explore Layers (- On the ribbon, choose Home > Explore Layers (in Layers) or choose Tools > Layers (in Explorer).
- On the menu, choose Format > Explore Layers or choose Tools > Explorer > Explore Layers.
- On the Explorer toolbar, click the Explore Layers tool.
- Type *explayers* and then press Enter.
- 2 Select the layer filter you want to export.
- 3 Click the Export Layer Filter tool (.
- 4 Locate the layer filter (.lft file) and select it.
- 5 Click Open.
- 6 To complete the command and return to your drawing, close the window.


Setting the current layer

When you create new entities, they are drawn on the current layer. To draw new entities on a different layer, you must first make that layer the current layer.

To make a layer current

- 1 Do one of the following to choose Explore Layers (- On the ribbon, choose Home > Explore Layers (in Layers) or choose Tools > Layers (in Explorer).
- On the menu, choose Format > Explore Layers or choose Tools > Explorer > Explore Layers.
- On the Explorer toolbar, click the Explore Layers tool.
- Type *explayers* and then press Enter.
- 2 In the Layer Name list, select the layer you want to make current.
- 3 Do one of the following:
 - Choose Edit > Current.
 - In the Layer Name list, select the name you want to make current, and then click the Current tool (.
 - Double-click the layer name in the Layer Name list.
 - Right-click the layer name you want to change, and from the shortcut menu select Current.
- 4 To complete the command and return to your drawing, close the window.

To set the current layer to that of an existing entity

- 1 Do one of the following to choose Set Layer By Entity (- On the ribbon, choose Home > Set Layer By Entity (in Layers).
- On the menu, choose Format > Layer Tools > Set Layer By Entity.
- On the Layer Tools toolbar, click the Set Layer By Entity tool.
- Type *setlayer* or *laymcur* and then press Enter.
- 2 Select the entity to set the current layer.

Use the command bar.

In the command bar, type laybyent and choose Set, then select an entity that is assigned the layer you want to be current.


Controlling layer visibility

A layer can be visible or invisible. Entities on invisible layers are not displayed and do not print. By controlling layer visibility, you can turn off unnecessary information, such as construction lines or notes. By changing layer visibility, you can put the same drawing to multiple uses.

For example, if you are drawing a floor plan, you can draw the layout of light fixtures on one layer and the location of plumbing lines on another. By selectively turning layers on and off, you can print the electrical engineering drawings and the plumbing drawings from the same drawing file. For even more convenience, you can control the visibility of layers within individual viewports, so that layers that display in one viewport are invisible in other viewports in the same drawing.

When you turn a layer off, entities drawn on that layer are no longer visible. When you turn the layer back on, the entities on that layer are redisplayed.

To turn layers on or off

- 1 Do one of the following to choose Explore Layers (- On the ribbon, choose Home > Explore Layers (in Layers) or choose Tools > Layers (in Explorer).
- On the menu, choose Format > Explore Layers or choose Tools > Explorer > Explore Layers.
- On the Explorer toolbar, click the Explore Layers tool.
- Type *explayers* and then press Enter.
- 2 Click the On/Off column for the layer you want to turn on or off.
- 3 To complete the command and return to your drawing, close the window.

Select an entity that is assigned to the layer you want to turn off.


In the command bar, type laybyent and choose Off or type layoff, then select an entity that is assigned the layer you want to turn off.

Use a shortcut to turn on all layers.

Choose Format > Layer Tools > Turn All Layers On or type layon in the command bar to turn on all layers in a drawing.

To control the visibility of external reference layers and save any changes made to them in the current drawing, turn on xref visibility.

To turn on xref layer visibility


- 1 Do one of the following to choose Explore Layers (- On the ribbon, choose Home > Explore Layers (in Layers) or choose Tools > Layers (in Explorer).
- On the menu, choose Format > Explore Layers or choose Tools > Explorer > Explore Layers.
- On the Explorer toolbar, click the Explore Layers tool.
- Type *explayers* and then press Enter
- 2 Choose View > Records from Xref Visibility.

Use the system variable.

You can also turn on xref layer visibility by typing visretain to access the system variable.

You can also freeze layers to improve the performance of operations such as zooming and panning or producing hidden lines or shaded images. When a layer is frozen, entities drawn on that layer are no longer visible.

To freeze or thaw layers

- 1 Do one of the following to choose Explore Layers (- On the ribbon, choose Home > Explore Layers (in Layers) or choose Tools > Layers (in Explorer).
- On the menu, choose Format > Explore Layers or choose Tools > Explorer > Explore Layers.
- On the Explorer toolbar, click the Explore Layers tool.
- Type *explayers* and then press Enter.
- 2 Click the Freeze/Thaw column for the layer you want to freeze or thaw.
- 3 To complete the command and return to your drawing, close the window.


Select an entity that is assigned to the layer you want to freeze.

In the command bar, type laybyent and choose Freeze or type layfrz, then select an entity that is assigned the layer you want to freeze.

Use a shortcut to thaw all layers.

Choose the Thaw All Layers command (type laythw) to thaw all layers in a drawing.

To freeze layers automatically when new layout viewports are created

- 1 Do one of the following to choose Explore Layers (- On the ribbon, choose Home > Explore Layers (in Layers) or choose Tools > Layers (in Explorer).
- On the menu, choose Format > Explore Layers or choose Tools > Explorer > Explore Layers.
- On the Explorer toolbar, click the Explore Layers tool.
- Type *explayers* and then press Enter.
- 2 Click the New VP Freeze column for the layer you want to freeze automatically in new layout viewports that you create.
- 3 To complete the command and return to your drawing, close the window.


Select entities that are assigned layers you want to freeze in existing layout viewports.

In the command bar, type layvpi to freeze layers of selected entities in all layout viewports except the current viewport.

Locking and unlocking layers

Locking a layer makes it easy to refer to information contained on the layer, but prevents you from accidentally modifying its entities. When a layer is locked (but visible and thawed), its entities remain visible, but you cannot edit them. If you lock the current layer, you can still add new entities to it. Unlocking a layer restores full editing capabilities.


To lock or unlock layers

- 1 Do one of the following to choose Explore Layers (- On the ribbon, choose Home > Explore Layers (in Layers) or choose Tools > Layers (in Explorer).
- On the menu, choose Format > Explore Layers or choose Tools > Explorer > Explore Layers.
- On the Explorer toolbar, click the Explore Layers tool.
- Type *explayers* and then press Enter.
- 2 Click the Lock/Unlock column for the layer you want to lock or unlock.
- 3 To complete the command and return to your drawing, close the window.

Select an entity that is assigned to the layer you want to lock or unlock.

In the command bar, type laybyent and choose Lock or Unlock or type laylck or layulk, then select an entity that is assigned the layer you want to lock or unlock.

To specify how faded entities on locked layers display

- 1 Do one of the following (- On the ribbon, choose Home > Locked Layer Fade Control (in Layers).
- Type *laylockfadectl* and then press Enter.
- 2 Specify the fade percentage to apply when displaying entities on locked layers.

The fade percentage can also be specified using the Options command.

Choose Options, and on the Display tab, click Fade Control. For more details, see “Setting the main window options” on page 837.

Controlling layer printing


Controlling layer printing is another way you can specify which entities print in your drawing.

By controlling layer printing, you can turn off unnecessary information during printing. For example, if you are drawing a floor plan, you can draw the layout of light fixtures on one layer and the location of plumbing lines on another. By selectively turning layers on and off when you print, you can print the electrical engineering drawings and the plumbing drawings from the same drawing file. By changing layer printing, you can put the same drawing to multiple uses.

When you turn off printing for a layer, entities drawn on that layer are still visible, but they do not print. If you turn off a layer's visibility, entities drawn on that layer do not display or print. Controlling layer printing can be especially helpful if you want layer visibility on, but do not want to print entities on that layer.

Layer visibility must be turned on in order to print entities drawn on that layer.


To turn layer printing on or off

- 1 Do one of the following to choose Explore Layers (- On the ribbon, choose Home > Explore Layers (in Layers) or choose Tools > Layers (in Explorer).
- On the menu, choose Format > Explore Layers or choose Tools > Explorer > Explore Layers.
- On the Explorer toolbar, click the Explore Layers tool.
- Type *explayers* and then press Enter.
- 2 Click the Print column for the layer you want to turn printing on or off for.

Controlling whether a layer is sectionable

If you use section planes to visualize a model, you can also determine whether entities on a layer are affected by the section planes you draw. This helps you control which entities can be cut by section planes. For more details about section planes, see “Using a section plane to view a drawing” on page 701.

To control whether a layer is sectionable


- 1 Do one of the following to choose Explore Layers (- On the ribbon, choose Home > Explore Layers (in Layers) or choose Tools > Layers (in Explorer).
- On the menu, choose Format > Explore Layers or choose Tools > Explorer > Explore Layers.
- On the Explorer toolbar, click the Explore Layers tool.
- Type *explayers* and then press Enter.
- 2 Click the Sectionable column for the layer you want to include or exclude from cutting by section planes.
- 3 To complete the command and return to your drawing, close the window.

Setting the layer color

Each layer in a drawing is assigned a color. PTCAD uses the BYLAYER color as the default color setting for entity creation so that new entities are drawn in the color of the layer on which they are inserted (set in the Drawing Settings dialog box).

Using the Explorer, you can set or change the color assigned to a layer. With the direct-editing feature, you can click on the color you want to change, and then select a new color from the dialog box that appears. Changing a layer’s color automatically changes the color of all entities on that layer with the BYLAYER color.

To change the layer color

- 1 Do one of the following to choose Explore Layers (- On the ribbon, choose Home > Explore Layers (in Layers) or choose Tools > Layers (in Explorer).
- On the menu, choose Format > Explore Layers or choose Tools > Explorer > Explore Layers.
- On the Explorer toolbar, click the Explore Layers tool.
- Type *explayers* and then press Enter.
- 2 Click the Color column for the layer you want to change.
- 3 In the Color dialog box, select a color on one of the following tabs:
 - **Index Color** — Click BYBLOCK, BYLAYER, or one of the 255 index colors. You can also type the color number in the Index box.
 - **True Color** — Click a basic color, click a color in the color palette, enter the Hue, Saturation, and Luminance (HSL) values, or enter the Red, Green, Blue (RGB) values. There are more than 16 million true colors from which you can choose.
 - **Color Books** — Select a color book from the list, then click a color. You can select Show Only Color Book Colors Used in Drawing to limit the selection to only those color book colors that are used in the current drawing. If necessary, create or edit color books by clicking Color Book Editor. For more details, see “Using color books” on page 74.
- 4 Click OK.

Entities don't have to use the layer's color.

You can assign a specific color to an entity, which overrides the layer's color setting. When you create a new entity, use the Format > Colors > Select Color command to change the current color. For an existing entity, select the entity, right-click for the shortcut menu, and choose Properties to change the entity's color using the Properties pane.

For more details about using color in the many aspects of your drawing, see “Working with colors” on page 71.

Setting the layer linetype


Each layer uses a default linetype (a repeating pattern of dashes, dots, or blank spaces). Linetype determines the appearance of entities both on the screen and when printed.

It's a good idea to assign the BYLAYER linetype to any entities that you draw on that layer. PTCAD uses the BYLAYER linetype as the default linetype setting for Entity Creation (in the Drawing Settings dialog box).

Using the Explorer, you can set or change the linetype assigned to a layer. With the direct-editing feature, you can click on the linetype you want to change, and then select a new linetype from the dialog box that appears. Changing the linetype assigned to a layer changes the linetype of all entities drawn on that layer with the BYLAYER linetype.

Only those linetypes already set in the drawing can be assigned to layers. For more information about setting additional linetypes, see “Working with linetypes” on page 306 in this chapter.

To change the linetype assigned to one or more layers

- 1 Do one of the following to choose Explore Layers (- On the ribbon, choose Home > Explore Layers (in Layers) or choose Tools > Layers (in Explorer).
- On the menu, choose Format > Explore Layers or choose Tools > Explorer > Explore Layers.
- On the Explorer toolbar, click the Explore Layers tool.
- Type *explayers* and then press Enter.
- 2 Click the Linetype column for the layer you want to change.
- 3 In the Linetype dialog box, select a new linetype for the layer, or click Browse to select your linetype file.

Entities don't have to use the layer's linetype.

You can also assign a specific linetype to an entity, which overrides the layer's linetype setting. When you create a new entity, use the Tools > Explorer > Explore Linetypes command to change the current linetype through the Explorer. For an existing entity, select the entity, right-click for the shortcut menu, and choose Properties. You can then modify the entity's linetype using the Properties pane.

Setting the layer lineweight

Each layer uses a default lineweight. Lineweights determine the thickness of entities both on the screen and when printed.


All new layers are assigned the DEFAULT lineweight, which is .25 millimeters or .01 inches. If you want a different lineweight assigned to a layer, you can easily change it using the Explorer. For example, you may want different lineweights on each layer of your drawing to show separate elements, such as walls, dimensions, structural steel, and electrical plans. Changing the lineweight assigned to a layer changes the lineweight of all entities drawn on that layer with the BYLAYER lineweight.

When you create new entities, it's a good idea to assign the BYLAYER lineweight to any entities that you draw on that layer, unless you want to override the layer lineweight. PTCAD uses the BYLAYER lineweight as the default lineweight setting when you create entities (in the Drawing Settings dialog box).

You can change the default lineweight.

To change the DEFAULT lineweight, choose Tools > Drawing Settings, click the Display tab, click the Lineweights tab, and then select a new default.

To change the lineweight assigned to one or more layers

- 1 Do one of the following to choose Explore Layers (- On the ribbon, choose Home > Explore Layers (in Layers) or choose Tools > Layers (in Explorer).
- On the menu, choose Format > Explore Layers or choose Tools > Explorer > Explore Layers.
- On the Explorer toolbar, click the Explore Layers tool.
- Type *explayers* and then press Enter.
- 2 Click the Lineweight column for the layer you want to change.
- 3 In the Lineweight list, select a new lineweight for the layer, then click OK.

Entities don't have to use the layer's lineweight.


You can assign a specific lineweight to an entity, which overrides the layer's lineweight setting. When you create a new entity, use the Tools > Drawing Settings > Entity Creation tab to change the current lineweight. For an existing entity, select the entity, right-click for the shortcut menu, and choose Properties. You can then modify the entity's lineweight using the Properties pane.

Setting the layer transparency

Each layer uses a transparency value from 0 to 90, where zero is no transparency (regular visibility) and 90 is the most transparency (almost invisible).

By default, each layer is assigned zero transparency. If you want a different transparency assigned to a layer, you can easily change it using the Explorer.


To change the transparency assigned to one or more layers

- 1 Do one of the following to choose Explore Layers (- On the ribbon, choose Home > Explore Layers (in Layers) or choose Tools > Layers (in Explorer).
- On the menu, choose Format > Explore Layers or choose Tools > Explorer > Explore Layers.
- On the Explorer toolbar, click the Explore Layers tool.
- Type *explayers* and then press Enter.
- 2 Click the Transparency column for the layer you want to change.
- 3 Enter a new value for the transparency, then press Enter.

Setting the layer material

Each layer can be assigned a material. By default, each layer is assigned the Global material. If you want a different material assigned to a layer, you can easily change it using the Explorer.

To change the material assigned to one or more layers

- 1 Do one of the following to choose Explore Layers (- On the ribbon, choose Home > Explore Layers (in Layers) or choose Tools > Layers (in Explorer).
- On the menu, choose Format > Explore Layers or choose Tools > Explorer > Explore Layers.
- On the Explorer toolbar, click the Explore Layers tool.
- Type *explayers* and then press Enter.
- 2 Click the Material column for the layer you want to change.
- 3 In the Material list, select a new material for the layer.

Setting the layer print style


If your drawing uses named print style tables, you can specify a print style for each layer. Named print style tables contain print styles that you set up to control what entities look like when they print, without actually changing the entities in the drawing.

If your drawing uses color-dependent print style tables, you cannot specify a print style for a layer. These types of print style tables automatically determine printing requirements by the color assigned to a layer or an entity. For details about converting a drawing that uses color-dependent print style table to use named print style tables, see “Changing the print style table type of a drawing” on page 657.

In drawings that use named print style tables, the default print style is Normal for all new layers. If desired, you can assign a print style using the Explorer. Changing the print style assigned to a layer changes the print style of all entities drawn on that layer with the BYLAYER print style.

When you create new entities, it’s a good idea to assign the BYLAYER print style to any entities that you draw on that layer, unless you want to override the layer print style. PTCAD uses the BYLAYER print style as the default print style setting when you create entities (in the Drawing Settings dialog box).

To change the print style assigned to one or more layers (only in a drawing that uses named print style tables)

- 1 Do one of the following to choose Explore Layers (- On the ribbon, choose Home > Explore Layers (in Layers) or choose Tools > Layers (in Explorer).
- On the menu, choose Format > Explore Layers or choose Tools > Explorer > Explore Layers.
- On the Explorer toolbar, click the Explore Layers tool.
- Type *explayers* and then press Enter.
- 2 Click the Print Style column for the layer you want to change.
- 3 If necessary, select a different print style table in the Active Print Style Table list.
- 4 In Print Styles, select a print style.
- 5 Click OK.

Entities don’t have to use layer’s print style.


For drawings that use named print style tables, you can also assign a specific print style to an entity, which overrides the layer’s print style setting. When you create a new entity, use the Tools > Drawing Settings > Entity Creation tab to change the current print style. For an existing entity, select the entity, right-click for the shortcut menu, and choose Properties. You can then modify the entity’s print style using the Properties pane.

Undoing changes made to layers

Changes made to layer settings can be reversed. For example, you can reverse changes made to a layer's color, transparency, freeze/thaw state, etc.

Some changes cannot be reversed. You cannot restore the original name of a layer if it was renamed, restore a layer if it was deleted or purged, or remove a layer if it was added.

To undo changes made to layers

- 1 Do one of the following to choose Layer Previous (- On the ribbon, choose Home > Layer Previous (in Layers).
- On the menu, choose Format > Layer Tools > Layer Previous.
- On the Layers toolbar, click the Layer Previous tool.
- Type *layerp* and then press Enter.
- 2 Repeat the command to undo a set of changes one-by-one.

Working with layer states

Layer states are collections of individual layers and their properties. You can restore layer states at any time, which makes it easy to switch between layer configurations according to your tasks. Additionally, you can specify whether to restore a layer state to the current viewport, all viewports, all layouts and viewports, or the Model tab.

Layer states can also be imported and exported. For example, to quickly create all the layers you need in a new drawing, create and export a layer state from an existing drawing that has the layers you need, and then import the layer state to the new drawing.


Layer states are saved in the drawing, including drawing templates and exported or imported drawings.

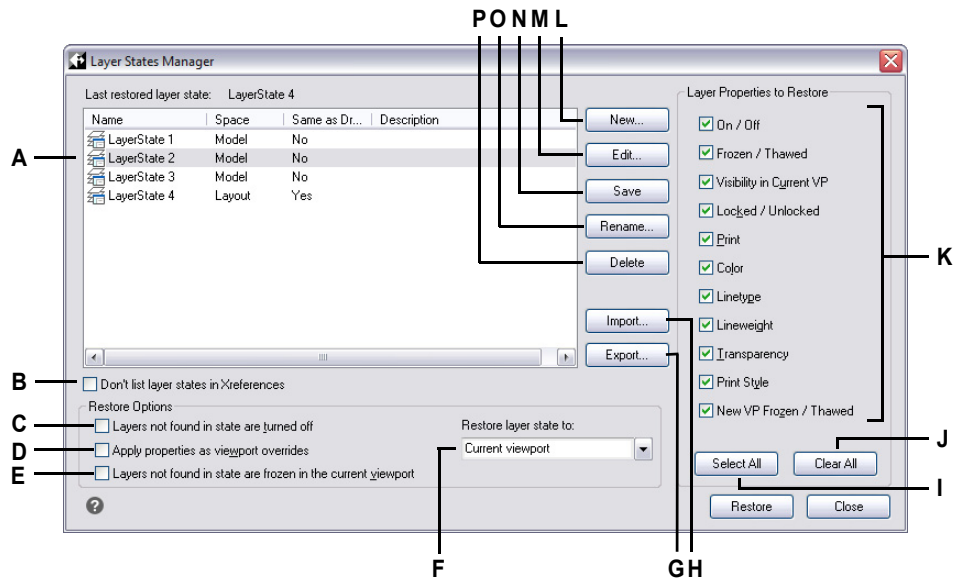
For each layer included in a layer state, you can set the following properties:

- Color
- Linetype
- On/Off
- Lock/Unlock
- Freeze/Thaw
- Lineweight
- Transparency
- Print
- New Viewports

Displaying layer states in the Layer States Manager

To display the Layer States Manager

- Do one of the following to choose Layer State Manager ():
 - On the ribbon, choose Home > Layer States Manager (in Layers).
 - On the menu, choose Format > Layer State Manager or choose Format > Explore Layers, then click the Layer States Manager tool.
 - On the Format or Layers toolbar, click the Layer States Manager tool.
 - Type *layerstate* and then press Enter.



- A** Displays layer states.
- B** Click to hide layer states located in externally referenced drawings.
- C** Click to turn off drawing layers that are not in the selected layer state when it is restored.
- D** Click to apply layer properties as overrides in the current viewport when the selected layer state is restored.
- E** Click to freeze drawing layers in the current viewport that are not in the selected layer state when it is restored.
- F** Select whether to restore the layer state to the current viewport, all viewports, all layouts and viewports, or the Model tab. If the Model tab is active, you can only restore the layer state to the current viewport.
- G** Click to export the selected layer state to an .las file.
- H** Click to import a layer state (.las file).
- I** Click to mark all checkboxes.
- J** Click to remove marks from all checkboxes.
- K** Select which properties to restore for the selected layer state.
- L** Click to create a new layer state.
- M** Click to modify the settings of the selected layer state.
- N** Click to copy current layer settings from the drawing to the selected layer state.
- O** Click to change the name or description of the selected layer state.
- P** Click to delete the selected layer state.


Creating layer states

Creating a layer state saves the current layer settings so you can restore them at a later time, modify them, and even import and export them for use in other drawings.

To create a layer state

- 1 Create all of the layers that you want to be in the layer state.

Although you can modify the layer state later, first you should set up your drawing with its layers and properties so when you create the layer state, it captures all of the settings. If you don't want to lose your current layer settings, simply save them as a layer state that you can restore later.

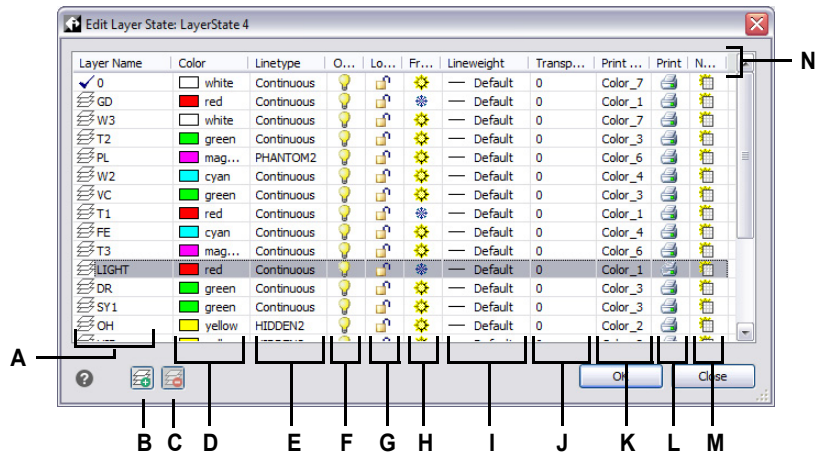
- 2 Do one of the following to choose Save Layer State (

To create a layer state using Layer State Manager

- 1 Create all of the layers that you want to be in the layer state.

Although you can modify the layer state later, first you should set up your drawing with its layers and properties so when you create the layer state, it captures all of the settings. If you don't want to lose your current layer settings, simply save them as a layer state that you can restore later.


- 2 Do one of the following to choose Layer States Manager (



- A** Displays the layers included in the layer state. Click to select a layer.
- B** Click to add a layer to the layer state.
- C** Click to delete the selected layer from the layer state.
- D** Click to change a layer's color for this layer state only.
- E** Click to change a layer's linetype for this layer state only.
- F** Click to toggle a layer's on/off status for this layer state only.
- G** Click to toggle a layer's lock/unlock status for this layer state only.
- H** Click to toggle a layer's freeze/thaw status for this layer state only.
- I** Click to change a layer's lineweight for this layer state only.
- J** Click to change a layer's transparency for this layer state only.
- K** Click to change a layer's print style for this layer state only.
- L** Click to change a layer's print status for this layer state only.
- M** Click to change a layer's visibility in new viewports for this layer state only.
- N** Click a column title to sort by category.


Applying a layer state

To apply the settings of a layer state

- 1 Do one of the following to choose Layer State Manager (- On the ribbon, choose Home > Layer States Manager (in Layers).
- On the menu, choose Format > Layer State Manager or choose Format > Explore Layers, then click the Layer States Manager tool.
- On the Format or Layers toolbar, click the Layer States Manager tool.
- Type *layerstate* and then press Enter.
- 2 Select the layer state to apply.
- 3 Layer states by design don't always contain all layers of a drawing. Make selections for the following:
 - **Layers not found in state are turned off** — After the layer state is applied, the only layers that will be turned on are those that are included in the layer state. If unmarked, the on/off status of unmatched layers is not changed.
 - **Layers not found in state are frozen in current viewport** — After the layer state is applied, the only layers that will be thawed in the current viewport are those that are included in the layer state. If unmarked, the freeze/thaw statuses of unmatched layers are not changed.
- 4 To apply layer properties as overrides in the current viewport, mark Apply Properties as Viewport Overrides.
- 5 In Restore Layer State To, select whether to restore the layer state to the current viewport, all viewports, all layouts and viewports, or the Model tab. If the Model tab is active, you can only restore the layer state to the current viewport.
- 6 In Layer Properties to Restore, mark which properties of all layers included in the layer state to apply. If a property is not marked, that property will not be applied for matching layers.
- 7 Click Restore.

Displaying layer states in the Explorer

To display layer states


Do one of the following or choose Explore Layer States ():

- On the ribbon, choose Tools > Layer States (in Explorer).
- On the menu, choose Tools > Explorer > Explore Layer States.
- On the Explorer toolbar, click the Explore Layer States tool.
- Type *explayerstates* and then press Enter.
- Choose Tools > Explorer, and then click the Layer States element.


Importing and exporting layer states from files

Layer states can be imported and exported as .las files. Exporting layer states from one drawing and importing them in a new drawing is a quick way to create all the new layers you need for the new drawing.

To import a layer state from a file

- 1 Do one of the following to choose Layer States Manager ():
 - On the ribbon, choose Home > Layer States Manager (in Layers).
 - On the menu, choose Format > Layer State Manager or choose Format > Explore Layers, then click the Layer States Manager tool.
 - On the Format or Layers toolbar, click the Layer States Manager tool.
 - Type *layerstate* and then press Enter.
- 2 Click Open.
- 3 Select the .las file to import.
- 4 Click Import.

To export a layer state to a file

- 1 Do one of the following to choose Layer State Manager ():
 - On the ribbon, choose Home > Layer States Manager (in Layers).
 - On the menu, choose Format > Layer State Manager or choose Format > Explore Layers, then click the Layer States Manager tool.
 - On the Format or Layers toolbar, click the Layer States Manager tool.
 - Type *layerstate* and then press Enter.
- 2 Select the layer state to export.
- 3 Click Export.
- 4 Enter a name for the .las file.
- 5 Click OK.

Working with linetypes

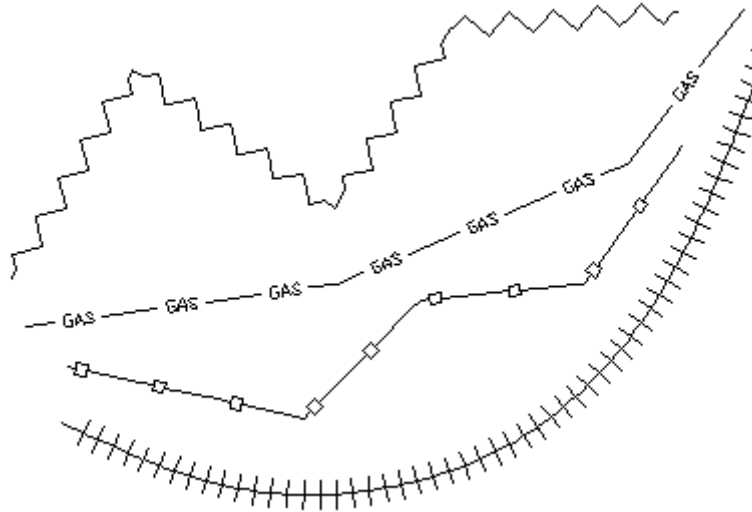
Linetypes are sequences of alternating line segments, dots, and blank spaces that affect the appearance of a line.

Understanding linetypes

PTCAD provides simple and complex linetypes:

- A *simple* linetype consists of a repeating pattern of dots, dashes, or blank spaces.
- A *complex* linetype contains embedded shape and text entities along with dots, dashes, and spaces.


You can use different linetypes to represent specific kinds of information. For example, if you are drawing a site plan, you can draw roads using a continuous linetype, a fence using a using a linetype of dashes with square posts, or a gas line using a complex linetype showing the text “GAS”.



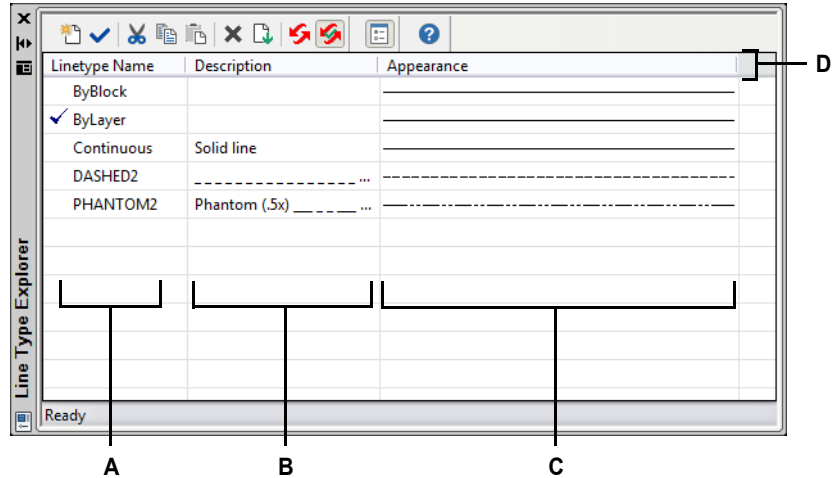
By default, every drawing has at least three linetypes: CONTINUOUS, BYLAYER, and BYBLOCK. You cannot rename or delete these linetypes. Your drawing may also contain an unlimited number of additional linetypes. You can load more linetypes into the program from a linetype library file or create and save linetypes you define.

Displaying linetype information in the Explorer

To display the Explorer Linetypes element

Do one of the following to choose Explore Linetypes ():

- On the ribbon, choose Tools > Linetypes (in Explorer).
- On the menu, choose Tools > Explorer > Explore Linetypes.
- On the Explorer toolbar, click the Explore Linetypes tool.
- Type *expltypes* and then press Enter.





- A** Lists names of linetypes loaded in the current drawing. A check mark indicates the current linetype.
- B** Describes linetypes loaded in the current drawing.
- C** Shows how linetypes will appear in the drawing.
- D** Click a column title to sort by category. Click and drag a column title to change the column order. Right-click a column title to choose which columns display or to restore all column settings to the default settings.

Setting the current linetype

You normally draw an entity using the linetype assigned to the current layer, indicated as BYLAYER. You can also assign linetypes on a per-entity basis, which overrides the layer's linetype setting. A third option is to assign the BYBLOCK linetype, whereby you draw new entities using the default linetype until you group them into a block. The entities inherit the current linetype setting when you insert the block into the drawing.

To make a linetype current

- 1 Do one of the following to choose Explore Linetypes (- On the ribbon, choose Tools > Linetypes (in Explorer).
- On the menu, choose Tools > Explorer > Explore Linetypes.
- On the Explorer toolbar, click the Explore Linetypes tool.
- Type *exptypes* and then press Enter.
- 2 In the Linetype Name list, select the linetype you want to make current.
- 3 Do one of the following:
 - Choose Edit > Current.
 - Select it in the Linetype Name list and click the Current tool ().
 - Double-click the linetype name.
- 4 To complete the command and return to your drawing, close the window.



Use the status bar.

*On the status bar, right-click on the word **BYLAYER** for the current linetype, click **Properties**, and then choose the linetype that you want to make current.*

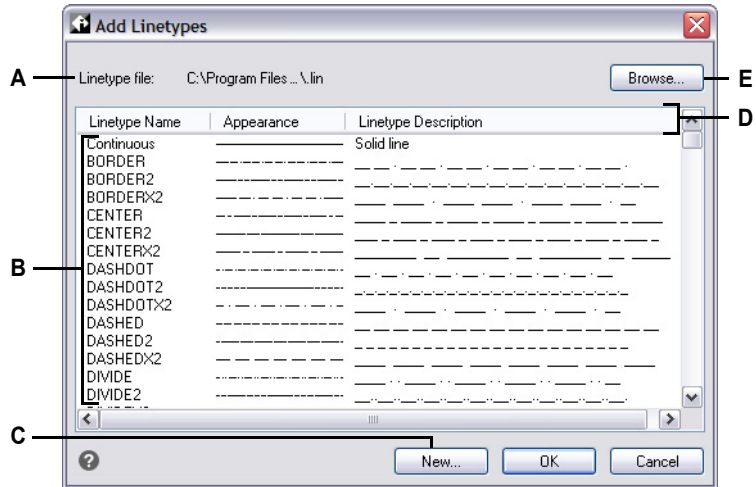
Loading additional linetypes

Before you can select a new linetype to use in a drawing, you must either create the linetype definition or load a predefined linetype from a linetype library file (*.lin). icad includes a linetype library file, *icad.lin*, which contains more than 100 predefined linetypes.

To load a new linetype from a linetype library

- 1 Do one of the following to choose Explore Linetypes (- On the ribbon, choose Tools > Linetypes (in Explorer).
- On the menu, choose Tools > Explorer > Explore Linetypes.
- On the Explorer toolbar, click the Explore Linetypes tool.
- Type *exptypes* and then press Enter.
- 2 Using one of the following methods, open the New Linetype dialog box:
 - Choose Edit > New > Linetype.
 - Click the New Item tool ().
 - With the cursor in the right side of the window, right-click to display the shortcut menu, and then choose New > Linetype.

- 3 Select the linetype to load.
- 4 If necessary, click Browse to choose a different linetype library file that contains the linetype definitions you want to load.
- 5 Click OK.




- A** Displays the name of the current linetype library file that contains the linetype definitions from which you can choose.
- B** Click to select and load a linetype.
- C** Click to create a new linetype definition for the current linetype library file.
- D** Click a column title to sort by category.
- E** Click to open a different linetype library file that contains the linetypes you want to choose.

Creating and naming linetypes

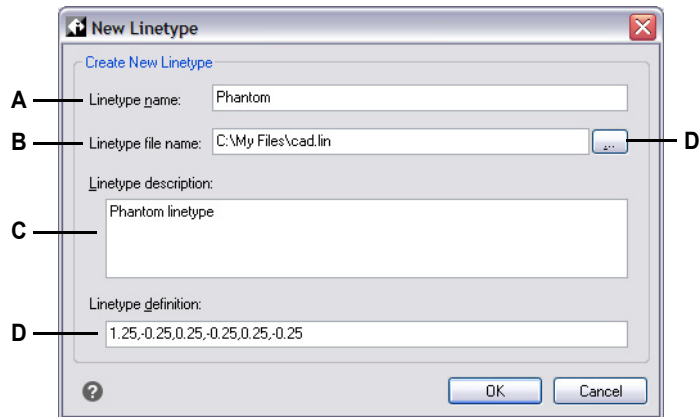
In addition to loading predefined linetypes from a linetype library file, you can create new linetypes. You can save new linetypes you create to a linetype library file for use in other drawings.

Creating a new simple linetype

To create a new simple linetype

- 1 Do one of the following to choose Explore Linetypes ():
 - On the ribbon, choose Tools > Linetypes (in Explorer).
 - On the menu, choose Tools > Explorer > Explore Linetypes.
 - On the Explorer toolbar, click the Explore Linetypes tool.
 - Type *exptypes* and then press Enter.

- 2 Using one of the following methods, open the New Linetype dialog box:
 - Choose Edit > New > Linetype.
 - Click the New Item tool (☀️).
 - With your cursor in the right side of the window, right-click to display the shortcut menu, and choose New > Linetype.
- 3 Click New.
- 4 Type the name of the line linetype. Do not use spaces between words in the new linetype name.
- 5 If necessary, in Linetype File Name, specify a different linetype library file to which you want to add the new linetype.
- 6 In Linetype Description, type the linetype description.
You can type anything in this field that will help you remember the purpose or appearance of this linetype. For example, it is helpful to type text or symbols such as ___..___.. that approximate the appearance of the linetype.
- 7 In Linetype Definition, type the linetype definition.
The definition consists of positive and negative numbers separated by commas. A positive number draws a solid line segment for the specified number of drawing units; a negative number creates a gap for the specified number of units; a zero creates a dot.
- 8 Click OK.





- | | |
|---|---|
| <p>A Type a name of the new linetype.</p> <p>B Enter the path to the linetype library file to which you want to add the new linetype.</p> <p>C Type any description in this box that helps you remember the purpose or appearance of the linetype.</p> | <p>D Type the definition of the linetype, consisting of positive and negative numbers separated by commas.</p> <p>E Click to browse and select a different linetype library file.</p> |
|---|---|

Creating a new complex linetype

To create a new complex linetype

A complex linetype can denote utilities, boundaries, contours, and so on. As with simple linetypes, complex lines are dynamically drawn as the user specifies vertices. Shapes and text entities embedded in lines are always displayed completely; they are never trimmed.

- 1 Do one of the following to choose Explore Linetypes (- On the ribbon, choose Tools > Linetypes (in Explorer).
- On the menu, choose Tools > Explorer > Explore Linetypes.
- On the Explorer toolbar, click the Explore Linetypes tool.
- Type *expltypes* and then press Enter.
- 2 Using one of the following methods, open the New Linetype dialog box:
 - Choose Edit > New > Linetype.
 - Click the New Item tool (.
 - With your cursor in the right side of the window, right-click to display the shortcut menu, and choose New > Linetype.
- 3 Click New.
- 4 Type the name of the line linetype. Do not use spaces between words in the new linetype name.
- 5 In Linetype Description, type the linetype description.

You can type anything in this field that will help you remember the purpose or appearance of this linetype. For complex linetypes, it is helpful to type a text description of the linetype.
- 6 In Linetype Definition, type the linetype definition.

As for the simple linetypes, the syntax for a complex linetype is a comma delimited list of pattern descriptors. For more details, see “Syntax for a complex linetype definition” on page 312 in this chapter.
- 7 Click OK.

Syntax for a complex linetype definition

Complex linetypes can include shape and text entities as pattern descriptors, as well as the dash and dot descriptors of simple linetypes.

The shape descriptor syntax

You can add a shape entity to a complex linetype using the following syntax:

```
[shape_name, shape_filename] or [shape_name, shape_filename, transform]
```

The definitions of the fields in the syntax are as follows.

`shape_name`

The name of the shape to add to the linetype. The shape name must exist in the specified shape file (`shape_filename`).

`shape_filename`

The name of a compiled shape definition file (extension *.shx). If no path is defined for the shape file name, the library path is searched for the file.

`transform`

The transform argument is optional and can be any series of the following (each preceded by a comma):

R=value Relative rotation

A=value Absolute rotation

S=value Scale

X=value X offset

Y=value Y offset

In this syntax, *value* represents a signed decimal number. The rotation is expressed in degrees while the other options are in linetype scaled drawing units.

`rotation`

R=value or A=value

R= determines a relative or tangential rotation with respect to the line's elaboration.

A= determines an absolute rotation of the shape with respect to the origin. All shapes have the same rotation regardless of their relative position to the line. The

value can be appended with a *d* for degrees (default), *r* for radians, or *g* for grads. If rotation is omitted, 0 relative rotation is used.

scale

S=value

Determines a factor by which the shape's internal scale is multiplied. If the shape's internal scale is 0, the scale value is used as the scale.

X offset

X=value

Determines a shift of the shape along the X axis of the linetype computed from the end of the linetype definition vertex. If X offset is omitted or is 0, the shape is elaborated with no offset. Include this field if you want a continuous line with shapes. This value is not scaled by the scale factor defined by S.

Y offset

Y=value

Determines a shift of the shape along the Y axis of the linetype computed from the end of the linetype definition vertex. If Y offset is omitted or 0, the shape is elaborated with no offset. This value is not scaled by the scale factor defined by S.

The text descriptor syntax

You can add a text entity to a complex linetype using the following syntax:

```
["string", style_name] or ["string", style_name, transform]
```

The definitions of the fields in the syntax are as follows.

string

The text to be used in the complex linetype. You cannot use the ``` or the `"` characters in the text string. To use these characters, enter a control code (`%%`) with the ASCII value for the character instead.

style_name

The name of the text style to be elaborated. The specified text style must be included. If it is omitted, use the currently defined style.

transform

The transform argument is optional and can be any series of the following (each preceded by a comma):

R=value	Relative rotation
A=value	Absolute rotation
S=value	Scale
X=value	X offset
Y=value	Y offset

In this syntax, *value* represents a signed decimal number. The rotation is expressed in degrees while the other options are in linetype scaled drawing units.

rotation

R=value or A=value

R= determines a relative or tangential rotation with respect to the line's elaboration.

A= determines an absolute rotation of the text with respect to the origin. All text has the same rotation regardless of its relative position to the line. The value can be appended with a *d* for degrees (default), *r* for radians, or *g* for grads. If *rotation* is omitted, 0 relative rotation is used.

Rotation is centered between the baseline and the nominal cap heights box.

scale

S=value

Determines a factor by which the style's height is multiplied. If the style's height is 0, the scale value is used as the scale.

Because the final height of the text is defined by both the scale value and the height assigned to the text style, you will achieve more predictable results by setting the text style height to 0. It is recommended that you create separate text styles for text in complex linetypes to avoid conflicts with other text in your drawing.

X offset

X=value

Determines a shift of the text along the X axis of the linetype computed from the end of the linetype definition vertex. If X offset is omitted or is 0, the text is elaborated by using the lower left corner of the text as the offset. Include this field if you want a continuous line with text. This value is not scaled by the scale factor that is defined by S.


Y offset

Y=value

Determines a shift of the text along the Y axis of the linetype computed from the end of the linetype definition vertex. If Y offset is omitted or is 0, the text is elaborated by using the lower left corner of the text as the offset. This value is not scaled by the scale factor that is defined by S.

Modifying linetypes

To change a linetype name

- 1 Do one of the following to choose Explore Lintypes (- On the ribbon, choose Tools > Lintypes (in Explorer).
- On the menu, choose Tools > Explorer > Explore Lintypes.
- On the Explorer toolbar, click the Explore Lintypes tool.
- Type *expltypes* and then press Enter.
- 2 Do one of the following:
 - Select the linetype, and then choose Edit > Rename.
 - Click the linetype name you want to change, and then type the new name.
 - Right-click the linetype name you want to change, and from the shortcut menu, select Rename.
- 3 To complete the command and return to your drawing, close the window.

The linetypes named CONTINUOUS, BYBLOCK, and BYLAYER are reserved.
You cannot rename them.

Working with multiline styles

A multiline style is a named, saved collection of format settings that determines the appearance of multilines.


Understanding multiline styles

When you create a multiline in a drawing, it is created using the current multiline style. The multiline style determines the number of parallel lines, the spacing between each line, and more.

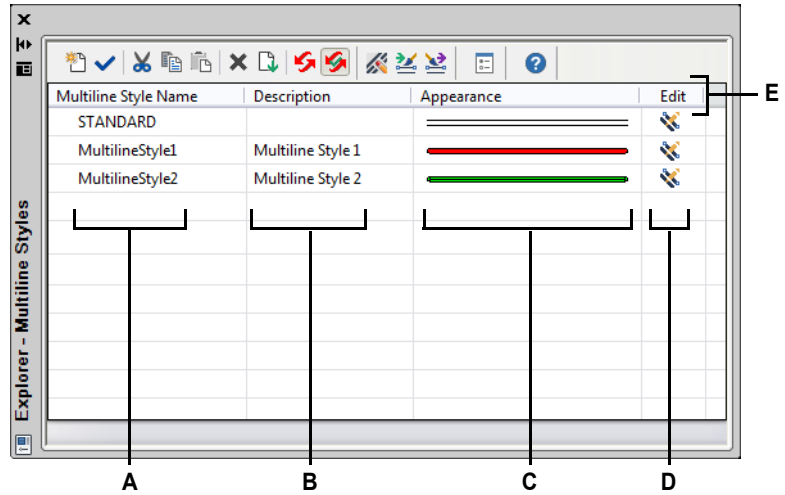
Every drawing has at least one multiline style, named Standard. You cannot delete the Standard style, but you can rename it or modify it. You can use an unlimited number of multiline styles in your drawing.

Displaying multiline style information in the Explorer

To display the Explorer Multiline Styles element

- Do one of the following to choose Explore Multiline Styles (- On the ribbon, choose Tools > Multiline Styles (in Explorer).
- On the menu, choose Tools > Explorer > Explore Multiline Styles.

- On the Explorer toolbar, click the Explore Multiline Styles tool.
- Type *expmultilinestyles* and then press Enter.




- A Lists names of multiline styles defined in the current drawing. A check mark indicates the current multiline style.
- B Describes multiline styles loaded in the current drawing.
- C Shows how multilines will appear in the drawing.
- D Click to modify the corresponding multiline style.
- E Click a column title to sort by category. Click and drag a column title to change the column order. Right-click a column title to choose which columns display or to restore all column settings to the default settings.

Creating and naming multiline styles


By using the Multiline Styles element in combination with the Modify Multiline Style dialog box, you can create new multiline styles and modify them.

To create a new multiline style

- 1 Do one of the following to choose Explore Multiline Styles ():
 - On the ribbon, choose Tools > Multiline Styles (in Explorer).
 - On the menu, choose Tools > Explorer > Explore Multiline Styles.
 - On the Explorer toolbar, click the Explore Multiline Styles tool.
 - Type *expmultilinestyles* and then press Enter.
- 2 Click the New Item tool ().
A new style is added to the multiline styles list.

- 3 Type the name for the new style by typing over the highlighted default text, and then press Enter.
- 4 Optionally click the Description and type text, and then press Enter.
- 5 Click  for the new style and make your selections.
For more details about the available settings, see “Working with multiline styles” on page 149.



To change a multiline style name in the current drawing

- 1 Do one of the following to choose Explore Multiline Styles ():
 - On the ribbon, choose Tools > Multiline Styles (in Explorer).
 - On the menu, choose Tools > Explorer > Explore Multiline Styles.
 - On the Explorer toolbar, click the Explore Multiline Styles tool.
 - Type *explinestyles* and then press Enter.
- 2 Do one of the following:
 - Click the multiline style name you want to change, type a new name, and then press Enter.
 - Right-click the multiline style name you want to change, and from the shortcut menu, select Rename, type a new name, and then press Enter.

Modifying multiline styles

A new multiline style is initially assigned default values that you can change. You can change these values for both new and existing multiline styles.

To modify a multiline style


- 1 Do one of the following to choose Explore Multiline Styles ():
 - On the ribbon, choose Tools > Multiline Styles (in Explorer).
 - On the menu, choose Tools > Explorer > Explore Multiline Styles.
 - On the Explorer toolbar, click the Explore Multiline Styles tool.
 - Type *explinestyles* and then press Enter.
- 2 Click  for the multiline style you want to modify, and then make your selections.

For more details about the available settings, see “Working with multiline styles” on page 149.

Setting the current multiline style

When you insert a multiline in a drawing, the multiline is created using the current multiline style. You can also assign a different multiline style when you create a multiline.

To make a multiline style current

- 1 Do one of the following to choose Explore Multiline Styles (.

Use a shortcut.

You can also make a multiline style current by double-clicking the multiline style name or by right-clicking the multiline style name and then choosing Current.

Working with text styles

A text style is a named, saved collection of format settings that determines the appearance of text.

Understanding text styles


When you add text to a drawing, it is created using the current text style. The text style determines the font, size, angle, orientation, if the text is annotative by default, and other text characteristics.

Every drawing has at least one text style, named Standard, which initially uses the Arial font. You cannot delete the Standard style, but you can rename it or modify it. For example, you can change the font or the oblique angle applied to the font. You also can use an unlimited number of additional text styles in your drawing.

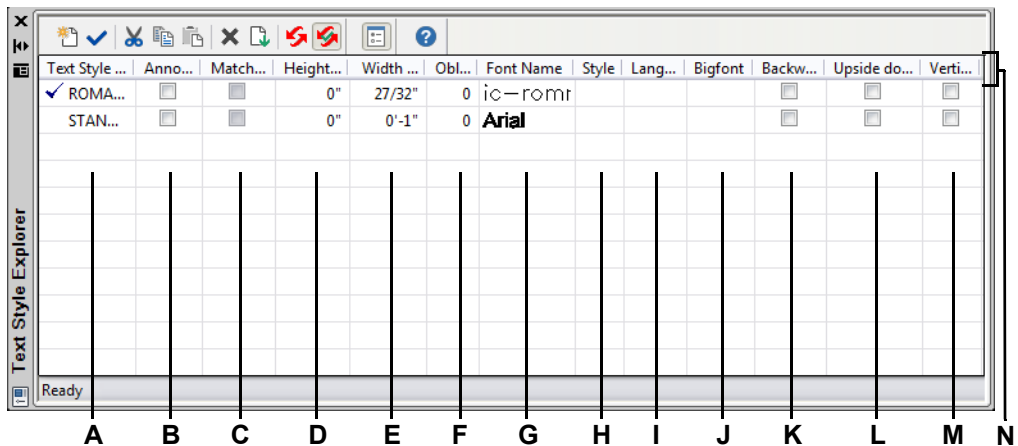
From the Explorer, you can directly edit any setting associated with a text style by using the single-click editing method to change the setting.

Displaying text style information in the Explorer

To display the Explorer Text Styles element

Do one of the following to choose Explore Text Styles ():

- On the ribbon, choose Tools > Text Styles (in Explorer).
- On the menu, choose Tools > Explorer > Explore Text Styles.
- On the Explorer toolbar, click the Explore Text Styles tool.
- Type *expstyles* and then press Enter.





- A** Lists names of text styles defined in the current drawing. A check mark indicates the current style.
- B** Indicates whether text will be annotative by default; you can click to turn the option on or off.
- C** Indicates whether annotative text in paper space will be oriented according to the layout viewport; you can click to turn the option on or off.
- D** Displays the height for the text style; you can type a new height.
- E** Displays the width factor for the text style; you can type a new width factor.
- F** Displays the oblique angle for the text style; you can enter a new oblique angle.
- G** Displays the font name on which the style is based; you can select a new font.
- H** Displays the font style, such as bold or italic; you can select a new font style.
- I** Displays the language on which the text style is based; you can select a new language.
- J** Displays whether Asian language big font files are used (for .shx file fonts only).
- K** Indicates whether text will appear backward; you can click to turn the option on or off.
- L** Indicates whether text will appear upside down; you can click to turn the option on or off.
- M** Indicates whether text will appear vertically; you can click to turn the option on or off.
- N** Click a column title to sort by category. Click and drag a column title to change the column order. Right-click a column title to choose which columns display or to restore all column settings to the default settings.

Creating and naming text styles

Fonts are character sets that consist of letters, numbers, punctuation marks, and symbols. Each font is stored in its own font file. Text styles apply additional formatting to fonts. You can create multiple text styles based on the same font, changing the various characteristics to alter the appearance of the font. To create a new text style, you assign formatting characteristics to a font.


PTCAD uses *.shx font files and provides a selection of fonts. These fonts are located in the PTCAD/Fonts directory. You can also use any font designed to work with AutoCAD as well as Windows system fonts. Many fonts are available from third-party vendors.

To create a new text style

- 1 Do one of the following to choose Explore Text Styles ():
 - On the ribbon, choose Tools > Text Styles (in Explorer).
 - On the menu, choose Tools > Explorer > Explore Text Styles.
 - On the Explorer toolbar, click the Explore Text Styles tool.
 - Type *expfonts* and then press Enter.
- 2 Do one of the following:
 - Choose Edit > New > Text Style.
 - Click the New Item tool ().

A new style is added to the text styles list with the default name, NewStyle1.
- 3 Type the name for the new style by typing over the highlighted default text, and then press Enter.
- 4 Click the columns for the items you want to change and make your selections for the new text style.
- 5 To complete the command, close the window.

To change a text style name in the current drawing


- 1 Do one of the following to choose Explore Text Styles ():
 - On the ribbon, choose Tools > Text Styles (in Explorer).
 - On the menu, choose Tools > Explorer > Explore Text Styles.
 - On the Explorer toolbar, click the Explore Text Styles tool.
 - Type *expfonts* and then press Enter.
- 2 Do one of the following:
 - Select the text style, choose Edit > Rename, type a new name, and then press Enter.
 - Click the text style name you want to change, type a new name, and then press Enter.
 - Right-click the text style name you want to change, and from the shortcut menu, select Rename, type a new name, and then press Enter.
- 3 To complete the command and return to your drawing, close the window.

Modifying text styles

A new text style is initially assigned default values for height, width factor, oblique angle, and other characteristics. You can change these values for both new and existing text styles. You can also change the font assigned to the text style. If you change the font or orientation properties of a text style assigned to text previously inserted in the drawing, all text using that style is regenerated to reflect the changes. Oblique angle and height, if specified, are given by the style definition when text is created, but are not updated for existing text when the style is changed.

A fixed text height value of 0 allows you to specify the text height at the time you insert text into the drawing. Any other value sets height of the text to that value; the program does not prompt for the text height when you insert text into the drawing. The width factor determines the horizontal scaling of text. A value less than 1 compresses the text (for example, 0.75 compresses the text 25 percent); a value greater than 1 expands the text (for example, 1.50 expands the text 50 percent). The oblique angle determines the forward or backward slant of text as an angle offset from 90 degrees. Negative values slant text to the left; positive values slant text to the right.



To modify a text style

- 1 Do one of the following to choose Explore Text Styles ():
 - On the ribbon, choose Tools > Text Styles (in Explorer).
 - On the menu, choose Tools > Explorer > Explore Text Styles.
 - On the Explorer toolbar, click the Explore Text Styles tool.
 - Type *expfonts* and then press Enter.
- 2 Click the desired column for the text style you want to change, then make the desired changes.
- 3 To complete the command, close the Explorer window.


Setting the current text style

When you insert text in a drawing, the text is created using the current text style. You can also assign a different text style when you create text.

To make a text style current

- 1 Do one of the following to choose Explore Text Styles ():
 - On the ribbon, choose Tools > Text Styles (in Explorer).
 - On the menu, choose Tools > Explorer > Explore Text Styles.
 - On the Explorer toolbar, click the Explore Text Styles tool.
 - Type *expfonts* and then press Enter.
- 2 In the Text Style Name list, click the style you want to make current.
- 3 Use one of the following methods to make the style the current style:
 - Choose Edit > Current.
 - Click the Current tool ().
 - With the cursor in the right side of the window, right-click to display the shortcut menu, and choose Current.
- 4 To complete the command and return to your drawing, close the Explorer window.

Use a shortcut.

You can also make a text style current by selecting it in the Text Style Name list and clicking the Current tool () or by double-clicking the text style name in the Text Style Name list.

Working with table styles

A table style is a named, saved collection of format settings that determines the appearance of tables.


Understanding table styles

When you add a table to a drawing, it is created using the current table style. The table style determines the table direction, text and border properties, and other settings.

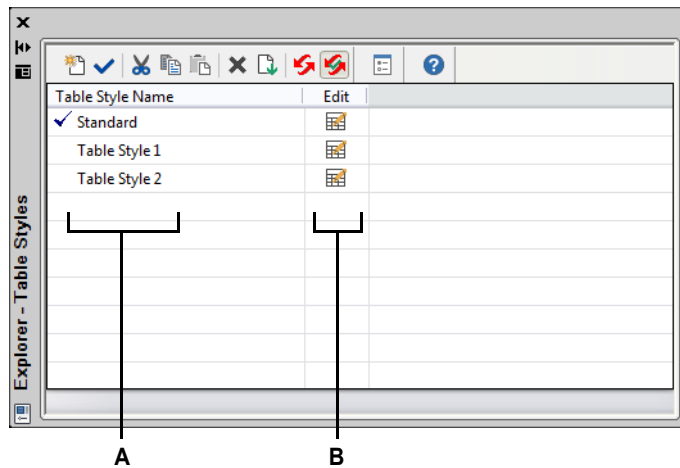
Every drawing has at least one table style, named Standard. You cannot delete the Standard style, but you can rename it or modify it, for example, to change the table direction. You can use an unlimited number of table styles in your drawing.

Displaying table style information in the Explorer

To display the Explorer Table Styles element

Do one of the following to choose Explore Table Styles ():

- On the ribbon, choose Tools > Table Styles (in Explorer).
- On the menu, choose Tools > Explorer > Explore Table Styles.
- On the Explorer toolbar, click the Explore Table Styles tool.
- Type *exptablestyles* and then press Enter.






- A** Lists names of table styles defined in the current drawing. A check mark indicates the current table style.
- B** Click to modify the corresponding table style.


Creating and naming table styles

By using the Table Styles element in combination with the Modify Table Style dialog box, you can create new table and cell styles and modify them.

To create a new table style

- 1 Do one of the following to choose Explore Table Styles (- On the ribbon, choose Tools > Table Styles (in Explorer).
- On the menu, choose Tools > Explorer > Explore Table Styles.
- On the Explorer toolbar, click the Explore Table Styles tool.
- Type *exptablestyles* and then press Enter.
- 2 Click the New Item tool ().
A new style is added to the table styles list.
- 3 Type the name for the new style by typing over the highlighted default text, and then press Enter.
- 4 Click () for the new style and make your selections.
For more details about the available settings, see “Working with table styles” on page 185.



To change a table style name in the current drawing

- 1 Do one of the following to choose Explore Table Styles (- On the ribbon, choose Tools > Table Styles (in Explorer).
- On the menu, choose Tools > Explorer > Explore Table Styles.
- On the Explorer toolbar, click the Explore Table Styles tool.
- Type *exptablestyles* and then press Enter.
- 2 Do one of the following:
 - Click the table style name you want to change, type a new name, and then press Enter.
 - Right-click the table style name you want to change, and from the shortcut menu, select Rename, type a new name, and then press Enter.

Modifying table styles

A new table style is initially assigned default values that you can change. You can change these values for both new and existing table styles.



To modify a table style

- 1 Do one of the following to choose Explore Table Styles (- On the ribbon, choose Tools > Table Styles (in Explorer).
- On the menu, choose Tools > Explorer > Explore Table Styles.
- On the Explorer toolbar, click the Explore Table Styles tool.
- Type *exptablestyles* and then press Enter.
- 2 Click () for the table style you want to modify, and then make your selections. For more details about the available settings, see “Working with table styles” on page 185.

Setting the current table style

When you insert a table in a drawing, the table is created using the current table style. You can also assign a different table style when you create tables.

To make a table style current

- 1 Do one of the following to choose Explore Table Styles (- On the ribbon, choose Tools > Table Styles (in Explorer).
- On the menu, choose Tools > Explorer > Explore Table Styles.
- On the Explorer toolbar, click the Explore Table Styles tool.
- Type *exptablestyles* and then press Enter.
- 2 In the list of table styles, click the style you want to make current.
- 3 Click the Current tool ()

Use a shortcut.

You can also make a table style current by double-clicking the table style name or by right-clicking the table style name and then choosing Current.

Working with dimension styles

From the Explorer, you can use the Dimension Styles element to cut, copy, and paste dimension styles from one drawing to another.

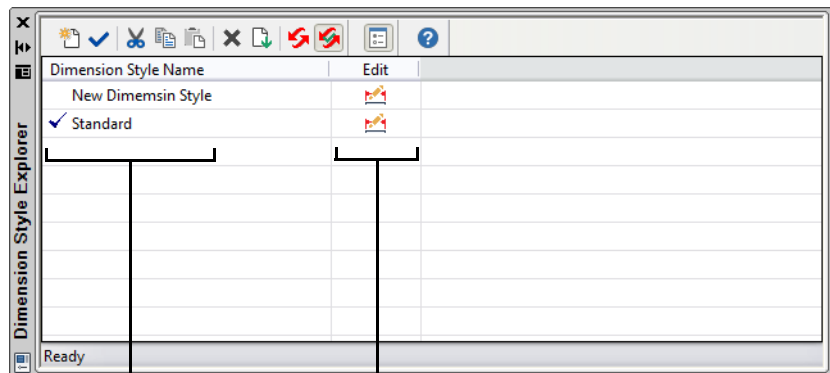
A dimension style contains the settings that control the appearance of a dimension. Although you cannot control these settings from within the Explorer, you can use the Modify Dimension Style dialog box to control settings related to the appearance of arrows, lines, text, units, and other formatting characteristics.

Displaying dimension style information in the Explorer

To display the Explorer Dimension Styles element

Do one of the following to choose Explore Dimension Styles ():

- On the ribbon, choose Tools > Dimension Styles (in Explorer).
- On the menu, choose Tools > Explorer > Explore Dimension Styles.
- On the Explorer toolbar, click the Dimension Styles tool.
- Type *setdim* and then press Enter.




- A** Lists the names of dimension styles defined in the current drawing. **B** Click to edit the corresponding dimension style.


Creating and naming dimension styles

By using the Dimension Styles element in combination with the Modify Dimension Style dialog box, you can create new dimension styles, modify them, and copy them into a different drawing.

To create a dimension style

- 1 Do one of the following to choose Explore Dimension Styles (- On the ribbon, choose Tools > Dimension Styles (in Explorer).
- On the menu, choose Tools > Explorer > Explore Dimension Styles.
- On the Explorer toolbar, click the Explore Dimension Styles tool.
- Type *expdimstyles* and then press Enter.
- 2 In the Dimension Styles Manager dialog box, click New.
- 3 Type the name of the new dimension style.
- 4 Click the Edit icon for the new dimension style.
- 5 In the Modify Dimension Style dialog box, select the desired options.
- 6 Click OK.


To change a dimension style name in the current drawing

- 1 Do one of the following to choose Explore Dimension Styles (- On the ribbon, choose Tools > Dimension Styles (in Explorer).
- On the menu, choose Tools > Explorer > Explore Dimension Styles.
- On the Explorer toolbar, click the Explore Dimension Styles tool.
- Type *expdimstyles* and then press Enter.
- 2 Do one of the following:
 - Select the dimension style, choose Edit > Rename, type a new name, and then press Enter.
 - Click the dimension style name you want to change, type a new name, and then press Enter.
 - Right-click the dimension style name you want to change, and from the shortcut menu, select Rename, type a new name, and then press Enter.
- 3 To complete the command and return to your drawing, close the window.

Copying dimension styles

You can copy and paste dimension styles between drawings. In addition to the steps below, you can also select a dimension style and use the Windows shortcut keys (Ctrl+A to select all, Ctrl+X to cut, Ctrl+C to copy, Ctrl+V to paste) to copy it in the Explorer. You can use these shortcut keys with all other Explorer elements as well.

To copy a dimension style from one drawing to another

- 1 Do one of the following to choose Explore Dimension Styles ():
 - On the ribbon, choose Tools > Dimension Styles (in Explorer).
 - On the menu, choose Tools > Explorer > Explore Dimension Styles.
 - On the Explorer toolbar, click the Explore Dimension Styles tool.
 - Type *expdimstyles* and then press Enter.
- 2 Right-click the dimension style name you want to copy.
- 3 From the shortcut menu, select Copy.
- 4 On the left, select the drawing to which you want to copy the dimension style.
- 5 Click the Dimension Styles element for that drawing.
- 6 On the right, right-click, and then from the shortcut menu, select Paste.

Each drawing contains a dimension style named Standard.

You cannot delete this dimension style, but you can rename it from within the Explorer or modify its properties in the Modify Dimension Style dialog box.

Working with multileader styles

A multileader style is a named, saved collection of format settings that determines the appearance of multileaders.


Understanding multileader styles

When you create a multileader, it is created using the current multileader style. The multileader style determines the arrowhead type, text style, colors, and more.

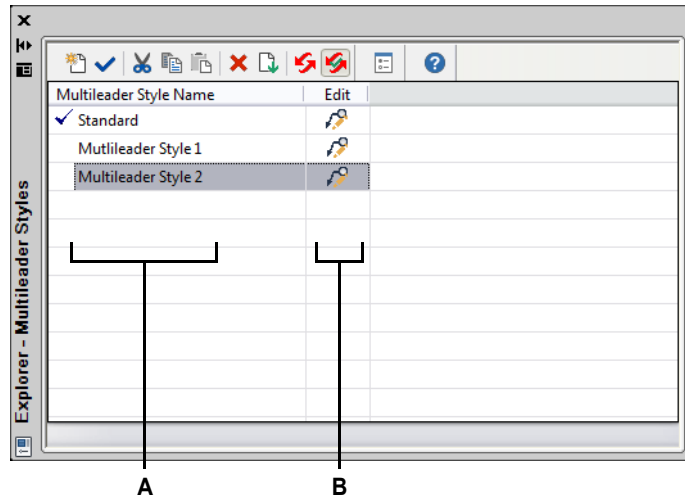
Every drawing has at least one multileader style, named Standard. You cannot delete the Standard style, but you can rename it or modify it. You can use an unlimited number of multileader styles in your drawing.

Displaying multileader style information in the Explorer

To display the Explorer Multileader Styles element

Do one of the following to choose Explore Multileader Styles ():

- On the ribbon, choose Tools > Multileader Styles (in Explorer).
- On the menu, choose Tools > Explorer > Explore Multileader Styles.
- On the Explorer toolbar, click the Explore Multileader Styles tool.
- Type *explleaderstyles* and then press Enter.






- A** Lists the names of multileader styles defined in the current drawing. **B** Click to modify the corresponding multileader style.


Creating and naming multileader styles

By using the Multileader Styles element in combination with the Modify Multileader Style dialog box, you can create new multiline styles and modify them.

To create a new multileader style

- 1 Do one of the following to choose Explore Multileader Styles (- On the ribbon, choose Tools > Multileader Styles (in Explorer).
- On the menu, choose Tools > Explorer > Explore Multileader Styles.
- On the Explorer toolbar, click the Explore Multileader Styles tool.
- Type *expmleaderstyles* and then press Enter.
- 2 Click the New Item tool ().
A new style is added to the multileader styles list.
- 3 Type the name for the new style by typing over the highlighted default text, and then press Enter.
- 4 Click () for the new style and make your selections.
For more details about the available settings, see “Working with multileader styles” on page 198.



To change a multileader style name in the current drawing

- 1 Do one of the following to choose Explore Multileader Styles (- On the ribbon, choose Tools > Multileader Styles (in Explorer).
- On the menu, choose Tools > Explorer > Explore Multileader Styles.
- On the Explorer toolbar, click the Explore Multileader Styles tool.
- Type *expmleaderstyles* and then press Enter.
- 2 Do one of the following:
 - Click the multileader style name you want to change, type a new name, and then press Enter.
 - Right-click the multileader style name you want to change, and from the shortcut menu, select Rename, type a new name, and then press Enter.

Modifying multileader styles

A new multileader style is initially assigned default values that you can change. You can change these values for both new and existing multileader styles.

To modify a multileader style



- 1 Do one of the following to choose Explore Multileader Styles (- On the ribbon, choose Tools > Multileader Styles (in Explorer).
- On the menu, choose Tools > Explorer > Explore Multileader Styles.
- On the Explorer toolbar, click the Explore Multileader Styles tool.
- Type *expmleaderstyles* and then press Enter.
- 2 Click () for the multileader style you want to modify, and then make your selections.

For more details about the available settings, see “Working with multileader styles” on page 198.

Setting the current multileader style

When you insert a multileader in a drawing, the multileader is created using the current multileader style. You can also assign a different multileader style when you create multileaders.

To make a multileader style current

- 1 Do one of the following to choose Explore Multileader Styles (- On the ribbon, choose Tools > Multileader Styles (in Explorer).
- On the menu, choose Tools > Explorer > Explore Multileader Styles.
- On the Explorer toolbar, click the Explore Multileader Styles tool.
- Type *expmleaderstyles* and then press Enter.
- 2 In the Multileader Style Name list, click the style you want to make current.
- 3 Click the Current tool ()

Use a shortcut.

You can also make a multileader style current by double-clicking the multileader style name or by right-clicking the multileader style name and then choosing Current.


Working with named views

A named view is a saved collection of view settings that determines a view's direction, boundaries, UCS, visual style, background, and perspective, etc. Named views can be saved and restored in the current window at any time. Use the Explorer to manage named views and quickly switch between views.

Also accessible from the Explorer is the View Manager, which is used to create new views and modify any view settings. For more details, see “Creating and switching to named views” on page 690.

Displaying views in the Explorer

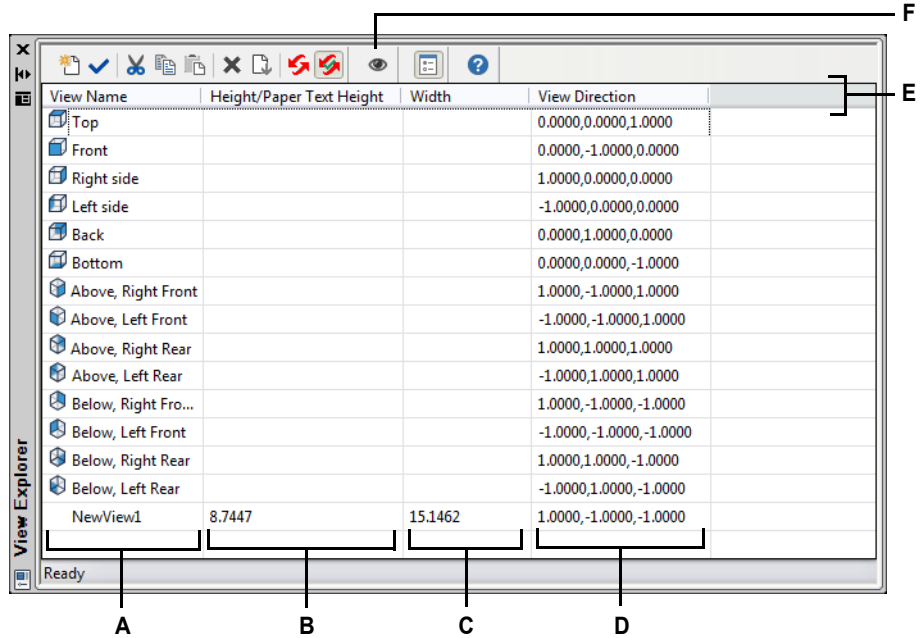
To display views in the Explorer

Do one of the following to choose Explore Views (

- On the ribbon, choose Tools > Views (in Explorer).
- Choose Tools > Explore Views.
- On the Explorer toolbar, click the Explore Views tool.
- Type *expviews* and then press Enter.
- Choose Tools > Explorer, and then click the Views element.

Named views are stored in the drawing.

You can use the Explorer to copy and paste named views from one drawing to another. Click the Show as Dialog tool to display the Explorer as a dialog box to see the currently open drawings, then copy and paste between them.





- A Lists names of views defined in the current drawing. A check mark indicates the current view.
- B Displays the height of the view in drawing units.
- C Displays the width of the view in drawing units.
- D Displays the direction of the view, expressed as a three-dimensional coordinate in the WCS.
- E Click a column title to sort by category. Click and drag a column title to change the column order. Right-click a column title to choose which columns display or to restore all column settings to the default settings.
- F Click to open the View Manager to specify additional view settings.

Saving and naming views

You can save the view displayed in the current window as a named view. After you save a named view, you can restore that view in the current window at any time.

To save a portion of the current view as a named view

- 1 Do one of the following to choose Explore Views ():
 - On the ribbon, choose Tools > Views (in Explorer).
 - On the menu, choose Tools > Explorer > Explore Views.
 - On the Explorer toolbar, click the Explore Views tool.
 - Type `expviews` and then press Enter.
- 2 Do one of the following:
 - Choose Edit > New > View.
 - On the Explorer toolbar, click the New Item tool ().
- 3 Specify the first corner of the view window.
- 4 Specify the opposite corner of the view window.
- 5 Type a new name for the view, and then press Enter.

Named views can be created in more ways using the View Manager.

From Explore Views, click the View Manager tool. For more details, see “Creating and switching to named views” on page 690.

Restoring named views

After you save one or more named views, you can restore any of those views in the current window using either the View command or PTCAD's Explorer.

To restore a named view from the Explorer

Do one of the following:

- Select the view name in the list of View settings, and then choose Edit > Current.
- Select the view name in the list of View settings, and then click the Current tool (✓).
- Double-click the view name in the View list.

To restore a named view using the View command

- 1 Do one of the following to choose View Manager (👁):
 - On the ribbon, choose View > View Manager (in Views).
 - On the menu, choose View > View Manager.
 - On the View toolbar, click the View Manager tool.
 - Type *view* and then press Enter.
- 2 Select the view you want to restore.
- 3 Click OK.

Restoring a named view can be affected by smooth view transitions.



Type voptions to specify whether smooth view transitions are used when restoring a named view (the view rotation), and to specify speed and performance settings.

Modifying the settings of named views

Once you create a named view you can modify its properties, such as the target direction and twist angle. This gives you access to many of the view settings after a view has been defined. Some properties are available directly in the Explorer, while others are available in the Views Manager.

Only named views that you create can be changed; preset views cannot be modified.

To change the view options

- 1 Do one of the following to choose Explore Views ():
 - On the ribbon, choose Tools > Views (in Explorer).
 - On the menu, choose Tools > Explore Views.
 - On the Explorer toolbar, click the Explore Views tool.
 - Type *expviews* and then press Enter.
- 2 To rename a view, do one of the following:
 - Select the view, choose Edit > Rename, type a new name, and then press Enter.
 - Click the view name you want to change, type a new name, and then press Enter.
 - Right-click the view name you want to change, and from the shortcut menu, select Rename, type a new name, and then press Enter.
- 3 Click the Height, Width, or View Direction column for the view you want to change, and then type your changes.
- 4 To make additional changes, click the View Manager tool (), change the desired settings, and then close the dialog box.

Working with visual styles

Visual styles are used to control how CAD geometry displays in real-time: 2D wireframe, hidden-line, shaded, shaded with materials, etc. Visual styles vary based on the capabilities of the OpenGL and OpenGL ES graphics devices.

Understanding visual styles


Visual styles can change the display of a drawing as you work in real-time. The current visual style determines whether a background displays, halo gap percentage, face settings, edge settings, and more.

Every drawing has a predefined list of visual styles, which cannot be deleted or renamed. However, you can create additional visual styles and change the properties of any visual style.

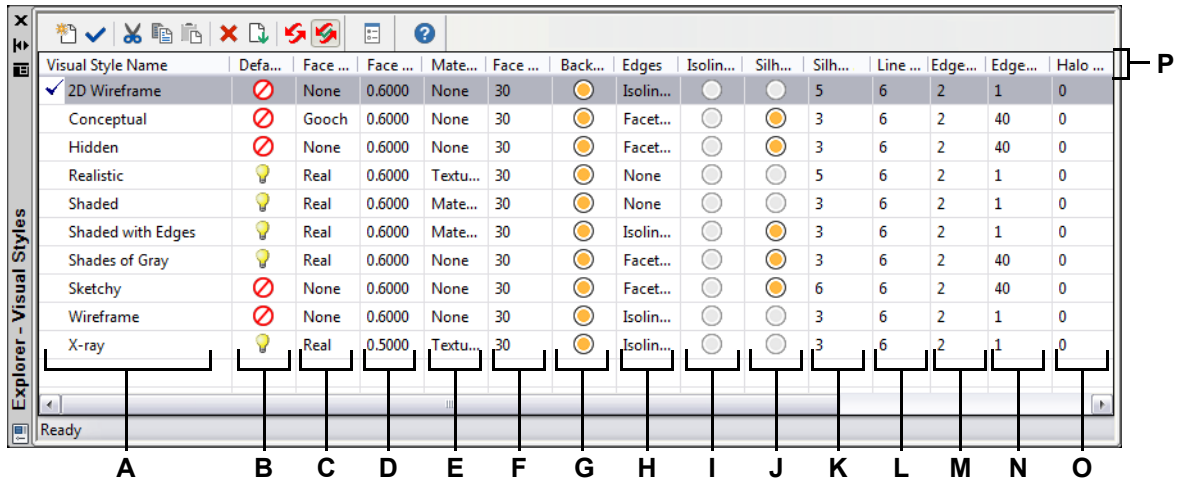
For more details about visual styles, see “Using dynamic visual styles for hiding, shading, and rendering” on page 767.

Displaying visual style information in the Explorer

To display the Explorer Visual Styles element

Do one of the following to choose Explore Visual Styles (

- On the ribbon, choose Tools > Visual Styles (in Explorer).
- On the menu, choose Tools > Explorer > Explore Visual Styles.
- On the Explorer toolbar, click the Explore Visual Styles tool.
- Type *expvstyles* and then press Enter.





- A** Lists names of visual styles defined in the current drawing. A check mark indicates the current visual style.
- B** Controls whether default lighting is on or off (for example, if a drawing has its own lights).
- C** Controls the style used to display faces: None, Real (realistic), or Gooch (non-realistic).
- D** Controls the opacity level applied to faces from 0 (fully transparent) to 1 (fully opaque).
- E** Controls whether materials and textures display.
- F** Controls the level of specular highlight used to display faces that do not have materials.
- G** Controls whether a background displays.
- H** Controls whether to display edges, and whether to display them as isolines or facet edges.
- I** Controls whether isolines display on top of shaded entities.
- J** Controls whether to display silhouette edges, which are the edges that connect a back facing entity with a front facing entity.
- K** Controls the number of pixels used to display silhouette edge width.
- L** Controls the number of pixels that lines and edges display beyond their endpoints for a hand-drawn effect.
- M** Controls the number of lines that are offset for a pencil effect.
- N** Controls the angle used for displaying crease edges.
- O** Controls the gap size displayed where one entity is hidden by another entity.
- P** Click a column title to sort by category. Click and drag a column title to change the column order. Right-click a column title to choose which columns display or to restore all column settings to the default settings.


Creating and naming visual styles

Using the Explorer, you can create new visual styles and modify their properties.

To create a new visual style

- 1 Do one of the following to choose Explore Visual Styles (- On the ribbon, choose Tools > Visual Styles (in Explorer).
- On the menu, choose Tools > Explorer > Explore Visual Styles.
- On the Explorer toolbar, click the Explore Visual Styles tool.
- Type *expvstyles* and then press Enter.
- 2 Click the New Item tool ().
A new style is added to the visual styles list.
- 3 Type the name for the new style by typing over the highlighted default text, and then press Enter.
- 4 Optionally click the Description and type text, and then press Enter.


To change a visual style name in the current drawing

- 1 Do one of the following to choose Explore Visual Styles (- On the ribbon, choose Tools > Visual Styles (in Explorer).
- On the menu, choose Tools > Explorer > Explore Visual Styles.
- On the Explorer toolbar, click the Explore Visual Styles tool.
- Type *expvstyles* and then press Enter.
- 2 Click the visual style name you want to change, type a new name, and then press Enter. Predefined visual styles cannot be renamed.

Modifying visual styles

A new visual style is initially assigned default values that you can change.

To modify a visual style

- 1 Do one of the following to choose Explore Visual Styles (- On the ribbon, choose Tools > Visual Styles (in Explorer).
- On the menu, choose Tools > Explorer > Explore Visual Styles.
- On the Explorer toolbar, click the Explore Visual Styles tool.
- Type *expvstyles* and then press Enter.
- 2 Change the properties for the visual style you want to modify.


You also use the Properties pane to modify a visual style.

When no entities are selected, the Properties pane displays properties for the drawing file. In Visual style, select the visual style you want to modify, then change its properties.

Setting the current visual style

The current visual style is the style used to display the drawing.

To make a visual style current

- 1 Do one of the following to choose Explore Visual Styles ()

Use a shortcut.

In the upper left corner of the drawing window, click the visual style control and choose a visual style to make it current. Or using the Explorer, you can also make a visual style current by double-clicking the visual style name or by right-clicking the visual style name and then choosing Current.

Working with coordinate systems

A coordinate system is a system of points that represents the drawing space in relation to an origin (0,0,0) and a set of axes that intersect at the origin.

Understanding coordinate systems

When you create entities in a drawing, they are located in relation to the drawing's underlying Cartesian coordinate system. Every drawing has a fixed coordinate system called the World Coordinate System (WCS). You cannot delete or modify the WCS.

Your drawing may contain additional coordinate systems, however, each with its own 0,0,0 origin and orientation. You can create as many user coordinate systems as you want, and then save and recall them as you need them. You can edit the origin of a coordinate system from within PTCAD's Explorer by single-clicking the origin coordinates and then typing new coordinates.

For example, you can create a separate user coordinate system (UCS) for each side of a building. Then, by switching to the UCS for the east side of the building, you can draw the windows on that side by specifying only their x- and y-coordinates.

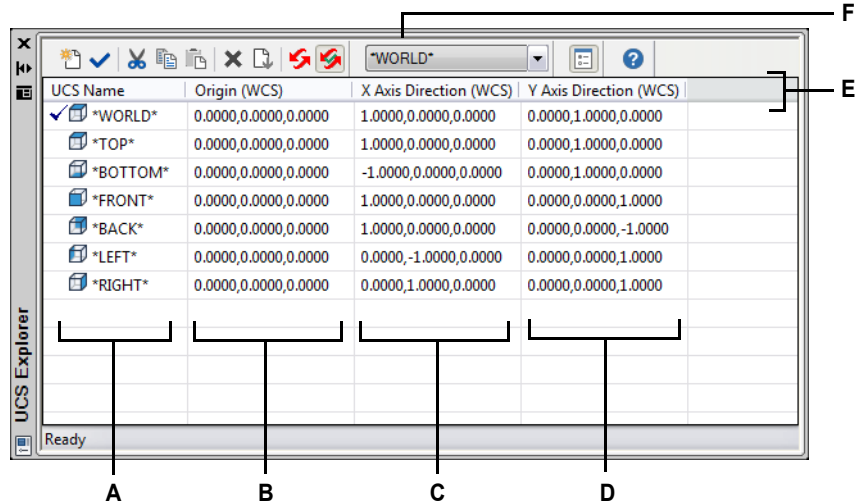
You can create and then switch between various user coordinate systems by selecting Coordinate Systems in the Explorer.

Displaying coordinate system information in the Explorer

To display the Coordinate Systems element

Do one of the following to choose Explore Coordinate Systems ():

- On the ribbon, choose Tools > Coordinate Systems (in Explorer).
- On the menu, choose Tools > Explorer > Explore Coordinate Systems.
- Choose Tools > User Coordinate Systems > Explore Coordinate Systems.
- On the Explorer toolbar, click the Explore Coordinate Systems tool.
- Type *expucs* and then press Enter.





- A** Lists names of coordinate systems defined in the current drawing. A check mark indicates the current coordinate system.
- B** Displays the origin of the coordinate system in relation to the WCS.
- C** Displays the x-axis direction of the coordinate system in relation to the WCS.
- D** Displays the y-axis direction of the coordinate system in relation to the WCS.
- E** Click a column title to sort by category. Click and drag a column title to change the column order. Right-click a column title to choose which columns display or to restore all column settings to the default settings.
- F** Select the base user coordinate system.

Defining and naming user coordinate systems

A drawing can contain as many coordinate systems as you want and can be named appropriate names so you can remember how they are used in your drawing for recalling them later.

To define new user coordinate systems in the Explorer


- Do one of the following to choose Explore Coordinate Systems ():
 - On the ribbon, choose Tools > Coordinate Systems (in Explorer).
 - On the menu, choose Tools > Explorer > Explore Coordinate Systems.
 - On the Explorer toolbar, click the Explore Coordinate Systems tool.
 - Type *expucs* and then press Enter.
- Do one of the following:
 - Choose Edit > New > UCS.
 - Click the New Item tool ().
 - With your cursor in the right side of the window, right-click to display the shortcut menu, and choose New > UCS.

- 3 Select a method from the prompt box or command bar by which to define the UCS in the drawing window.

For example, select *3 Point* and then specify three points in the drawing window to define the x, y, and z axes for your coordinate system.

- 4 Type the name for the new user coordinate system by typing over the highlighted default text, and then press Enter.
- 5 To complete the command, close the window.

To change a user coordinate system name in the current drawing


- 1 Do one of the following to choose Explore Coordinate Systems (- On the ribbon, choose Tools > Coordinate Systems (in Explorer).
- On the menu, choose Tools > Explorer > Explore Coordinate Systems.
- On the Explorer toolbar, click the Explore Coordinate Systems tool.
- Type *expucs* and then press Enter.
- 2 Do one of the following:
 - Select the user coordinate system, and then choose Edit > Rename, type a new name, and then press Enter.
 - Click the user coordinate system name you want to change, type a new name, and then press Enter.
 - Right-click the user coordinate system name you want to change, and from the shortcut menu, select Rename, type a new name, and then press Enter.
- 3 To complete the command and return to your drawing, close the window.

Setting the current user coordinate system

When you draw new entities, they are created in relation to the current coordinate system. You can set the current UCS from the Explorer.

To set the current UCS from the Explorer

Do one of the following:

- Double-click the UCS name in the UCS Name list.
- Select the UCS in the UCS Name list, and then choose Edit > Current.
- Select the UCS in the UCS Name list, and then click the Current tool (.

Working with blocks

Blocks represent a special type of entity that, once saved, can be inserted and manipulated in the drawing as a single entity.

Understanding blocks

A block can consist of visible entities such as lines, arcs, and circles as well as visible or invisible data called attributes. You can use attributes to track things such as part numbers and prices and to export attribute information to an external database. You can also track the number of parts by counting the number of times a block has been inserted into the drawing. Blocks are stored as part of the drawing file.

External references have similar uses to blocks. Using external references, you can attach entire drawings to your current drawing. Unlike a block, however, an external reference does not become part of the current drawing.


You can save blocks to an external .dwg file from the Explorer. You can also use the Explorer to manage and insert copies of blocks. The Explorer window lists the names of all blocks contained in the current drawing, along with other information about each block or external reference.

You can also rename a block, modify its insertion point, and change the path of an externally referenced drawing by single-clicking on the property and making your edits within the Explorer.

In addition to the Explorer, you can also work with blocks by using the Block Editor, by using various block commands, and by adding blocks to tool palettes. For more details about the Block Editor and various block commands, see “Working with blocks” on page 546. For more details about tool palettes, see “Customizing tool palettes” on page 864.

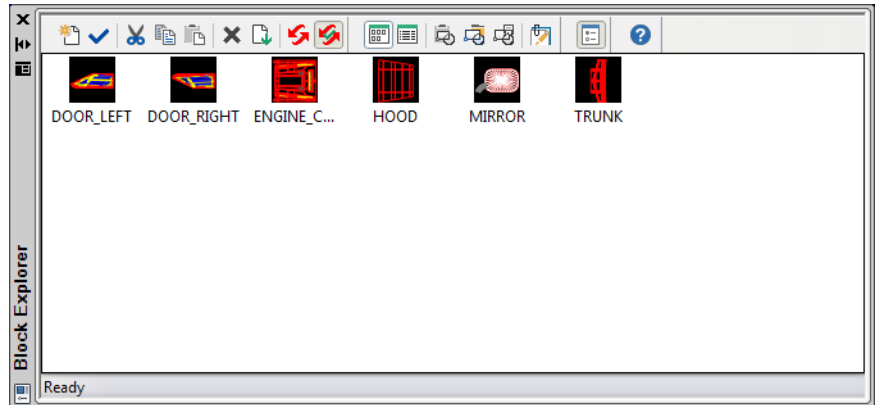
Displaying block information in the Explorer

To display blocks in the Explorer

Do one of the following to choose Explore Blocks (

- On the ribbon, choose Tools > Blocks (in Explorer).
- On the menu, choose Tools > Explorer > Explore Blocks.
- On the Explorer toolbar, click the Explore Blocks tool.
- Type *expblocks* and then press Enter.







The Blocks element in the Explorer defaults with icons on. The Icons view shows you a small image of each block.




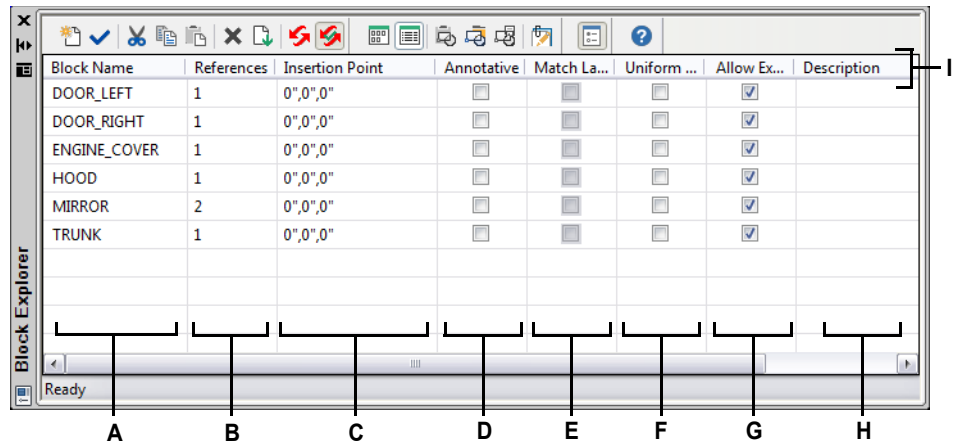
The Images view shows an image of each block in the selected drawing. Click an image to select it.

When blocks are displayed, additional tools on the Block toolbar provide the functions described in the following table:

Additional tools on the Block toolbar

Tool	Function
 Icons	Displays an image of each block.
 Details	Displays information about each block.
 Insert	Inserts a block.
 Insert External File Block	Inserts a drawing available from disk as a block.
 Save Block	Saves the selected block as an independent <i>.dwg</i> file. If more than one block is selected, saves each block as an independent <i>.dwg</i> file in a folder you specify.
 Edit Block Reference	Redefines the definition of a block.

To see more information about each block, click the Details tool (). In the Details view, you can edit the path and the insertion point by clicking the setting and typing your changes.








- A** Lists names of blocks defined in the current drawing. Click and type to modify it.
- B** Displays the number of occurrences of the block in the current drawing.
- C** Displays the insertion point of the block in the current drawing. Click and type to modify it.
- D** Indicates whether the block is annotative by default; click to turn the option on or off.
- E** Indicates whether the block, when located in paper space, is oriented automatically according to the layout viewport; click to turn the option on or off. (Available only for annotative blocks.)
- F** Indicates whether the block must scale proportionately; click to turn the option on or off.
- G** Indicates whether the block can be exploded; click to turn the option on or off.
- H** Displays the description of the block. Click and type to modify it.
- I** Click a column title to sort by category. Click and drag a column title to change the column order. Right-click a column title to choose which columns display or to restore all column settings to the default settings.

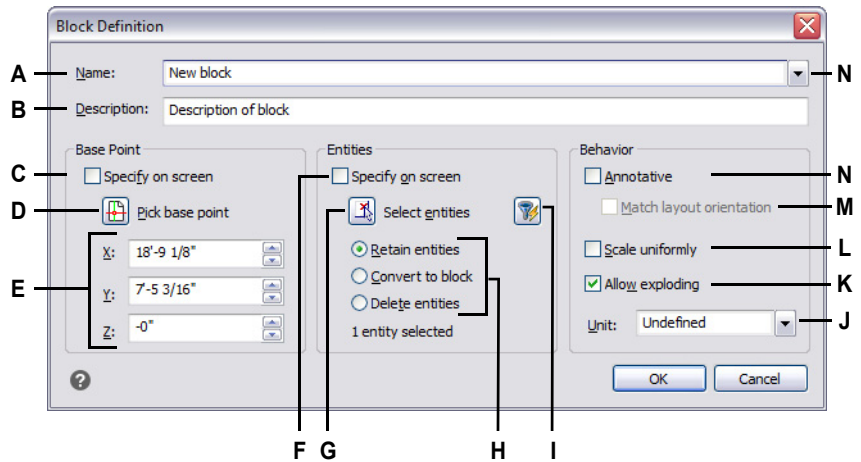
Creating and naming blocks

You can combine any number of entities into a single block. After you create a block, you can insert copies of it into a drawing. Each block insertion is treated as a single entity; for example, you can rotate or scale each block when you insert it. The program adds the name of the new block you insert to the Block Name list in the Explorer.

To create a block


- 1 Do one of the following to choose Explore Blocks (- On the ribbon, choose Tools > Blocks (in Explorer).
- On the menu, choose Tools > Explorer > Explore Blocks.
- On the Explorer toolbar, click the Explore Blocks tool.
- Type *expblocks* and then press Enter.
- 2 Do one of the following:
 - Choose Edit > New > Block.
 - On the Explorer toolbar, click the New Item tool (.
- 3 In the Block definition dialog box, enter a name and description for the new block.
- 4 Specify the insertion point for the block by doing one of the following:
 - **Specify on Screen** Mark this check box to select the base point in the drawing after you click OK.
 - **Pick Base Point** Click () to temporarily close the dialog box immediately, select the base point in the drawing, then return to the dialog box. This option is available only if Specify on Screen is not marked.
 - **X, Y, and Z** Enter the x-, y-, and z-coordinates of the base point. This option is available only if Specify on Screen is not marked.
- 5 Select the entities to be combined into the block by doing one of the following:
 - **Specify on Screen** Mark this check box to select the entities in the drawing after you click OK.
 - **Select entities** Click () to temporarily close the dialog box immediately, select the entities in the drawing, then return to the dialog box. Or you can click () to select entities by type or value. This option is available only if Specify on Screen is not marked.
- 6 Select what to do with the entities after the block is created:
 - **Retain entities** Entities selected for the block remain in the drawing.
 - **Convert to block** Entities selected for the block are converted to the block, which remains in the drawing.
 - **Delete entities** Entities selected for the block are removed from the drawing.

- 7 Select any of the following options for the block:
- **Annotative** Determines whether the block is annotative by default. The display and printing of annotative blocks is affected by annotation scaling. If annotative by default, you can determine whether the block, when located in paper space, is oriented automatically according to the layout viewport.
 - **Scale uniformly** Mark this check box to retain the aspect ratio if the block is scaled. Annotative blocks must be scaled proportionately.
 - **Allow exploding** Mark this check box to allow the block to be exploded into separate entities.
 - **Unit** Defines the unit of the block, for example inches or millimeters.
- 8 Click OK.
- The program adds a new block to the blocks list, with the name you entered for it.



- A** Enter the name of the block.
- B** Enter a description of the block.
- C** Select to pick the base point in the drawing after you click OK.
- D** Select to pick the base point now in the drawing.
- E** Displays the base point coordinates or you can enter them manually.
- F** Select to select entities to include in the block after you click OK.
- G** Click to select entities now in the drawing.
- H** Select whether to keep, convert, or delete entities after the block is created.
- I** Click to select entities by type or value.
- J** Choose block units.
- K** Select to allow the block to be exploded into entities.
- L** Select to maintain aspect ratio if the block is scaled.
- M** Select to make the block, when located in paper space, oriented according to the layout viewport (available only if Annotative is marked).
- N** Mark to make the block annotative by default.



To change a block name in the current drawing

- 1 Do one of the following to choose Explore Blocks (- On the ribbon, choose Tools > Blocks (in Explorer).
- On the menu, choose Tools > Explorer > Explore Blocks.
- On the Explorer toolbar, click the Explore Blocks tool.
- Type *expblocks* and then press Enter.
- 2 Do one of the following:
 - Select the block, choose Edit > Rename, type a new name, and then press Enter.
 - Click the block name you want to change, type a new name, and then press Enter.
 - Right-click the block name you want to change, and from the shortcut menu, select Rename, type a new name, and then press Enter.
- 3 To complete the command and return to your drawing, close the window.

Inserting a block

You can insert into a drawing any block listed in the Block Name list in the Explorer. This includes blocks contained within any open drawing (you copy and paste the block first) and also dynamic blocks that were created using a different CAD application.

To insert a block

- 1 Do one of the following to choose Explore Blocks (- On the ribbon, choose Tools > Blocks (in Explorer).
- On the menu, choose Tools > Explorer > Explore Blocks.
- On the Explorer toolbar, click the Explore Blocks tool.
- Type *expblocks* and then press Enter.
- 2 If you want to insert a block from another open drawing, select the drawing in the left side of the Explorer window, copy the block, and paste it in the destination drawing. (If the block is contained within the same drawing, you can skip this step.)
- 3 Select the block to be inserted.
- 4 On the Explorer toolbar, click the Insert tool (.
- 5 In the drawing, specify the insertion point.

If inserting a dynamic block, you can press Ctrl while left-clicking the mouse to cycle through block insertion points defined by its dynamic parameters.
- 6 Specify the x, y, and z scale factor and the rotation angle, or in the prompt box, select Done.
- 7 To complete the command and return to your drawing, close the window.

The scale can easily be changed after insertion.

To change the scale of a block after insertion, choose Express Tools > Blocks > Block Scale. You can also change the position, scale, and rotation using the Properties pane.


Use a shortcut.

You can insert a block by choosing Tools > Explorer, and then double-clicking the name of the block you want to insert in the Block Name list. And, you can also insert a block from the Insert menu, see Chapter 13, “Working with other files in your drawings.”

Inserting a drawing as a block

You can insert another drawing as a block into the current drawing. After you do this, the block name is added to the Block Name list in the Explorer. Changes made later to the inserted drawing will not be reflected in this drawing.

To insert a drawing as a block

- 1 Do one of the following to choose Explore Blocks (.
- 3 In the Insert Block dialog box, select the drawing you want to insert, and then click Open.
- 4 In the drawing, specify the insertion point.
- 5 Specify the x, y, and z scale factor and the rotation angle, or in the prompt box, select Done.
- 6 To complete the command and return to your drawing, close the window.


The scale can easily be changed after insertion.

To change the scale of a block after insertion, choose Express Tools > Blocks > Block Scale. You can also change the position, scale, and rotation using the Properties pane.

Saving a block as a separate drawing

You can save a block as a separate drawing, and then you can open and modify that drawing as you would any other drawing.

To save a block as a separate drawing file


- 1 Do one of the following to choose Explore Blocks (.
- 4 In the Save Block dialog box, select the folder in which you want to save the block.
- 5 In the File Name field, type a name for the new drawing file (or accept the default, in which case the new drawing name is the same as the name of the block), and then click Save.

Working with groups

In PTCAD's Explorer, you can manage groups — collections of entities saved together as one unit — that exist in an open drawing, create new groups, manage the entities contained in a group, and change the settings of groups.


Displaying information about groups in the Explorer

To display groups in the Explorer

- 1 Do one of the following to choose Explore Groups (

Creating a new group using the Explorer

To create a new group using the Explorer

- 1 Do one of the following:
 - Choose Edit > New > Group.
 - On the Explorer toolbar, click the New Item tool (.

A new group is added as Anonymous.
- 2 Type the name for the new group by typing over the highlighted default text, and then press Enter.
- 3 Click the [+] tool to select the entities to be included in the group; click the [-] tool to select the entities to remove from the group.

You can also click the Number of Entities column and choose Add Entities or Remove Entities.
- 4 Press Enter when done selecting entities.
- 5 To complete the command, close the window.

Modifying groups

To modify a group using the Explorer

- 1 In the Explorer, select Groups.
- 2 To rename a group, do one of the following:
 - Select the group, choose Edit > Rename, type a new name, and then press Enter.
 - Click the group name you want to change, type a new name, and then press Enter.
 - Right-click the group name you want to change, and from the shortcut menu, select Rename, type a new name, and then press Enter.
- 3 To add entities to a group, select the group and click the [+] tool, select the entities in the drawing to be included in the group, then press Enter when done selecting entities.
- 4 To remove entities from a group, select the group and click the [-] tool, select the entities in the drawing to remove from the group, then press Enter when done selecting entities.
- 5 To make a group selectable or unselectable in the drawing, click Selectable for it.
- 6 To reorder the entities in a group, select the group and click the Reorder tool, make your selections, then click OK.
- 7 To select a group in the drawing and have that group highlighted in the list of groups, click the Highlight tool, select the desired group, then press Enter. The appropriate group will be highlighted in the list.
- 8 To complete the command, close the window.

For more details about working with groups, see “Grouping entities” on page 450.




Working with references to external files

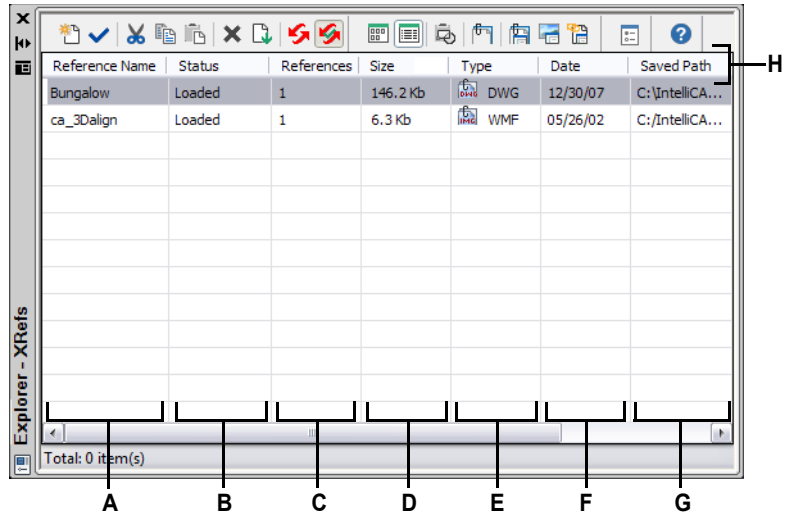
In PTCAD's Explorer, you can work with any file that is referenced from an open drawing. In addition to commonly used xrefs, or externally referenced drawing files, you can manage other attached external files such as data links, raster images, .dwf files, .dgn files, .pdf files, and point clouds directly from the Explorer. If your program version supports working with BIM files, you can also manage .ifc, .rvt, and .rfa files.

When you reference an external file from a drawing, the contents of referenced file appear in the current drawing, but the contents themselves are not added to the drawing.

Displaying information about referenced files in the Explorer

To display information about referenced files



- 1 Do one of the following to choose Explore External References (- On the ribbon, choose Tools > External References (in Explorer).
- On the menu, choose Tools > Explorer > Explore External References.
- On the Explorer toolbar, click the Explore External References tool.
- Type *exprefs* and then press Enter.
- 2 Do one of the following:
 - Click the Icons tool () to see a small image of each referenced file,
 - Click the Details tool () to see more detailed information about each block.



- A** Lists names of files referenced from the current drawing.
- B** Displays the load status of the file in the current drawing.
- C** Displays the number of times the file is referenced from the current drawing.
- D** Displays the size of the file.
- E** Displays the type of file.
- F** Displays the date of the file.
- G** Displays the location of the file.
- H** Click a column title to sort by category. Click and drag a column title to change the column order. Right-click a column title to choose which columns display or to restore all column settings to the default settings.

Attaching referenced files

To attach an externally referenced file


- 1 Do one of the following to choose Explore External References (- On the ribbon, choose Tools > External References (in Explorer).
- On the menu, choose Tools > Explorer > External References.
- On the Explorer toolbar, click the Explore External Reference tool.
- Type *exprefs* and then press Enter.
- 2 Do one of the following:
 - Choose Edit > New > External Reference.
 - On the Explorer toolbar, click the New Item tool (.
- 3 In Files of type, select the type of file to attach, then click Open.
- 4 Specify the insertion point for the file and complete the additional prompts.
- 5 To complete the command and return to your drawing, close the window.

After an external file is attached, the Manage Xrefs icon displays in the status bar, notifying that the drawing contains a linked external reference file.


Modifying the settings for referenced files

Any referenced file can be modified from the Explorer, including renaming, linking to a new location, clipping, and more.




To change the name of a referenced file in the current drawing

- 1 Do one of the following to choose Explore External References (- On the ribbon, choose Tools > External References (in Explorer).
- On the menu, choose Tools > Explorer > Explore External References.
- On the Explorer toolbar, click the Explore External Reference tool.
- Type *exprefs* and then press Enter.
- 2 Do one of the following:
 - Select the file, choose Edit > Rename, type a new name, and then press Enter.
 - Click the file name you want to change, type a new name, and then press Enter.
 - Right-click the file name you want to change, and from the shortcut menu, select Rename, type a new name, and then press Enter.
- 3 To complete the command and return to your drawing, close the window.

To change the name of a referenced file in the current drawing

- 1 Do one of the following to choose Explore External References (- On the ribbon, choose Tools > External References (in Explorer).
- On the menu, choose Tools > Explorer > Explore External References.
- On the Explorer toolbar, click the Explore External Reference tool.
- Type *exprefs* and then press Enter.
- 2 Do one of the following:
 - Select the file, choose Edit > Rename, type a new name, and then press Enter.
 - Click the file name you want to change, type a new name, and then press Enter.
 - Right-click the file name you want to change, and from the shortcut menu, select Rename, type a new name, and then press Enter.
- 3 To complete the command and return to your drawing, close the window.

To manage an externally referenced file

- 1 Do one of the following to choose Explore External References (- On the ribbon, choose Tools > External References (in Explorer).
- On the menu, choose Tools > Explorer > Explore External References.
- On the Explorer toolbar, click the Explore External Reference tool.
- Type *exprefs* and then press Enter.
- 2 Do one of the following:
 - Select a file in the list, choose Edit, then the desired option.
 - Right-click the desired file, and from the shortcut menu, select the desired option.
 - Click either the Image Manager tool () or the Xref Manager tool ()
- 3 To complete the command and return to your drawing, close the window.

For more details about working with external references, see “Working with external references” on page 572.

Working with layouts


Each drawing that you create contains the area where you do most of your work on the Model tab and can also contain numerous layouts on Layout tabs that simulate the paper on which you will print a copy of the drawing.

Using the Explorer, you can manage the layouts in a drawing, assign a page setup, and you can also easily copy layouts to be reused in other drawings.

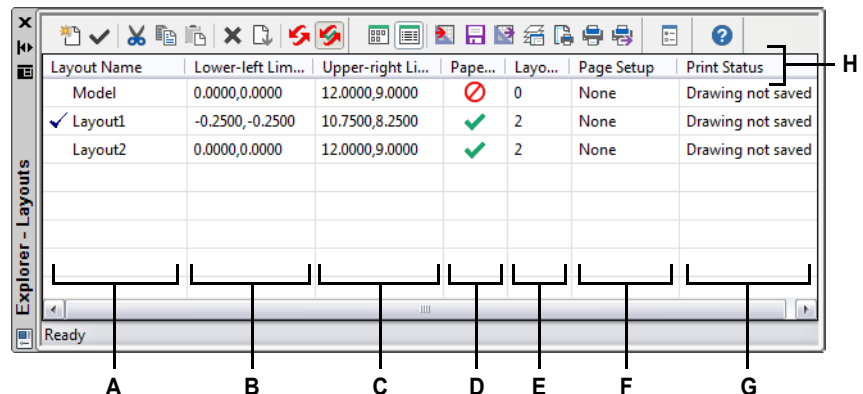
Making a layout active in the Explorer is the equivalent of clicking its corresponding tab in the drawing window.

Displaying layouts in the Explorer

To display layouts in the Explorer

Do one of the following to choose Explore Layouts ():

- On the ribbon, choose Tools > Layouts (in Explorer).
- On the menu, choose Tools > Explore Layouts.
- On the Explorer toolbar, click the Explore Layouts tool.
- Type *explayouts* and then press Enter.
- Choose Tools > Explorer, and then click the Views element.





- A** Lists names of layouts defined in the current drawing. A check mark indicates the current layout.
- B** Displays the coordinates of the lower-left corner of the layout.
- C** Displays the coordinates of the upper-right corner of the layout.
- D** Controls whether linetype scaling in layout viewports is always the same regardless of the zoom factor. (Layouts only; not the model.)
- E** Displays the number of viewports in the layout.
- F** Choose the page setup for the layout. Choose Page Setup Manager to modify page setups.
- G** Displays whether a layout is ready to print.
- H** Click a column title to sort by category. Click and drag a column title to change the column order. Right-click a column title to choose which columns display or to restore all column settings to the default settings.

Creating and naming layouts

Each drawing can contain one layout for the model and up to 255 other layouts.


To create a new layout

- 1 Do one of the following to choose Explore Layouts (- On the ribbon, choose Tools > Layouts (in Explorer).
- On the menu, choose Tools > Explorer > Explore Layouts.
- On the Explorer toolbar, click the Explore Layouts tool.
- Type *explayouts* and then press Enter.
- 2 Do one of the following:
 - Choose Edit > New > Layout.
 - Click the New Item tool (.

A new layout is added to the layouts list with a default name.

- 3 Type the name for the new layout by typing over the highlighted default text, and then press Enter.
- 4 To complete the command, close the window.

To change a layout name using the Explorer

- 1 Do one of the following to choose Explore Layouts (- On the ribbon, choose Tools > Layouts (in Explorer).
- On the menu, choose Tools > Explorer > Explore Layouts.
- On the Explorer toolbar, click the Explore Layouts tool.
- Type *explayouts* and then press Enter.
- 2 Do one of the following:
 - Select the layout, choose Edit > Rename, type a new name, and then press Enter.
 - Click the layout name you want to change, type a new name, and then press Enter.
 - Right-click the layout name you want to change, and from the shortcut menu, select Rename, type a new name, and then press Enter.
- 3 To complete the command and return to your drawing, close the window.


Specifying page setup options for a layout

Each layout can have its own page setup assigned to it. This enables you to accommodate unique print settings for each layout. If some layouts use the same print settings, those layouts can be assigned the same page setup.


Assigning a page setup to a model or layout doesn't mean it will always print with the specified settings. All of the print settings specified for a page setup can be overridden at print time.

For more details about page setups, see “Working with page setups” on page 630.

To assign a page setup to layouts

- 1 Do one of the following to choose Explore Layouts (- On the ribbon, choose Tools > Layouts (in Explorer).
- On the menu, choose Tools > Explorer > Explore Layouts.
- On the Explorer toolbar, click the Explore Layouts tool.
- Type *explayouts* and then press Enter.
- 2 Select one or more layouts.
- 3 In the Page Setup column for one of the layouts, select the desired page setup from the list.
- 4 To complete the command, close the Explorer window.

To modify the settings of an assigned page setup

- 1 Do one of the following to choose Explore Layouts (- On the ribbon, choose Tools > Layouts (in Explorer).
- On the menu, choose Tools > Explorer > Explore Layouts.
- On the Explorer toolbar, click the Explore Layouts tool.
- Type *explayouts* and then press Enter.
- 2 In the Page Setup column for the desired layout, choose Page Setup Manager.
- 3 Select the page setup that requires changes, then click Modify.
- 4 Select the new options, then click OK.
- 5 If necessary, select the page setup that you want to assign to the layout, then click Set Current.
- 6 Click Close.
- 7 To complete the command, close the Explorer window.

Working with materials


Materials are images or textures that enhance the visual appearance of entities when they are rendered. From PTCAD's Explorer, you can use the Materials element to load materials from a material library to your drawing so your drawing contains only those materials you need, which helps reduce the drawing file size. You can also cut, copy, and paste materials from one drawing to another.

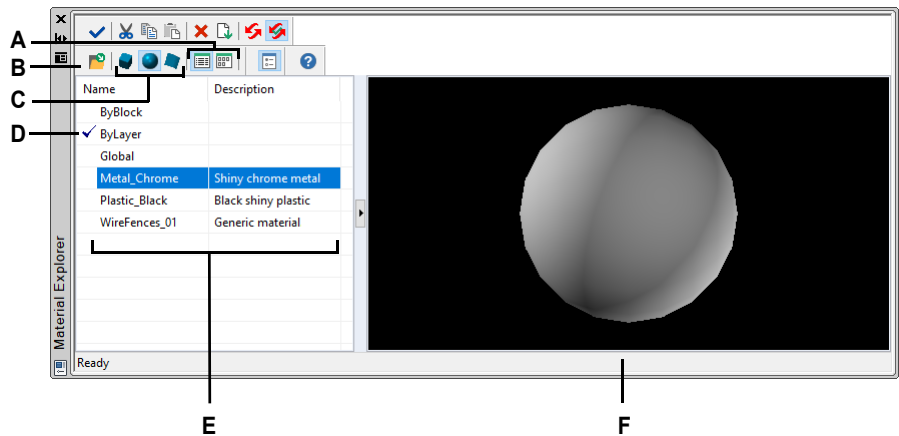
Once you import materials to the drawing in the Explorer, you can then assign them to entities and layers, include them in packages created with the eTransmit command, and render them with visual styles directly in PTCAD.

Display materials in the drawing using the Realistic visual style. For more details about working with visual styles, see "Using dynamic visual styles for hiding, shading, and rendering" on page 767.

Displaying information about materials in the Explorer

To display materials in the Explorer

- Do one of the following to choose Explore Materials ():
 - On the ribbon, choose Tools > Materials (in Explorer).
 - On the menu, choose Tools > Explorer > Explore Materials.
 - On the Format toolbar, click the Explore Materials tool.
 - Type *expmaterials* and then press Enter.





- | | |
|---|--|
| <p>A Choose to view the materials in a list or image view.</p> <p>B Click to add materials to the drawing from the library so materials can be assigned to entities and layers.</p> <p>C Choose to preview the selected material on the right as a cube, sphere, or plane.</p> | <p>D Double-click a material to set it as the current material.</p> <p>E Lists the names of materials available in the current drawing.</p> <p>F Displays the selected material on a shape.</p> |
|---|--|

Importing materials to the drawing

Materials are stored in a library with an .icmat file extension. To reduce the file size of drawings, you import from the library only those materials you need in the drawing.

Imported materials are stored in the drawing, and then you can work with them throughout PTCAD, including assigning them to entities and layers, include them in packages created with the eTransmit command, purge them, and display them on entities using any visual style other than 2D Wireframe.

To import materials to a drawing

- 1 Do one of the following to choose Explore Materials ():
 - On the ribbon, choose Tools > Materials (in Explorer).
 - On the menu, choose Tools > Explorer > Explore Materials.
 - On the Format toolbar, click the Explore Materials tool.
 - Type *expmaterials* and then press Enter.
- 2 On the Explorer toolbar, click the Import Materials tool (.

Use a shortcut.

At the command bar, type materialimport to open the Import Materials dialog box.

- 3 Right-click either list, and from the shortcut menu, choose how to view materials while you select them.
- 4 On the left, choose the category of materials you want to view.
- 5 On the right, select the materials you want to add to the drawing. Press Shift + click to select multiple materials.
- 6 Click Import.
The selected materials are added to the drawing and appear in the Material Explorer.
- 7 Click Close when done importing.

Materials of older file formats can be updated to the current format.



If you open a drawing that contains materials that were created in older formats, type CONVERTOLDMATERIALS at the command prompt to update them to the current format.

Assigning materials to entities and layers

When you create new entities, they are drawn with the current material. To draw new entities with a different material, you must first make that material the current material.

You can also separately assign materials to specific entities and layers.


To make a material current

- 1 Do one of the following to choose Explore Materials ():
 - On the ribbon, choose Tools > Materials (in Explorer).
 - On the menu, choose Tools > Explorer > Explore Materials.
 - On the Format toolbar, click the Explore Materials tool.
 - Type *expmaterials* and then press Enter.
- 2 Select the material you want to make current.
- 3 Do one of the following:
 - Click the Current tool (.
 - Double-click the material name.
 - Right-click the material name, and from the shortcut menu, select Current.

To assign a material to an entity

- 1 Do one of the following:
 - Drag-and-drop a material from the Materials Explorer to an entity.
 - Select an entity and in the Properties pane, choose the desired material.
 - Select an entity, type *change*, and follow the prompts to modify the material property.



To assign a material to a layer

- 1 Do one of the following to choose Explore Layers ():
 - On the ribbon, choose Home > Explore Layers (in Layers) or choose Tools > Layers (in Explorer).
 - On the menu, choose Format > Explore Layers or choose Tools > Explorer > Explore Layers.
 - On the Explorer toolbar, click the Explore Layers tool.
 - Type *explayers* and then press Enter.
- 2 Click the Material column for the desired layer.
- 3 Select a material from the list.

Copying materials

You can copy and paste materials between drawings. In addition to the steps below, you can also select a material and use the Windows shortcut keys (Ctrl+A to select all, Ctrl+X to cut, Ctrl+C to copy, Ctrl+V to paste) to copy it in the Explorer. You can use these shortcut keys with all other Explorer elements as well.

To copy a material from one drawing to another

- 1 Do one of the following to choose Explore Materials (- On the ribbon, choose Tools > Materials (in Explorer).
- On the menu, choose Tools > Explorer > Explore Materials.
- On the Format toolbar, click the Explore Materials tool.
- Type *expmaterials* and then press Enter.
- 2 Right-click the material you want to copy.
- 3 From the shortcut menu, select Copy.
- 4 If necessary, click Show as Dialog () to display Material Explorer as a dialog box instead of a pane. (Panels display information about the current drawing only.)
- 5 On the left, select the drawing to which you want to copy the material.
- 6 Click the Materials element for that drawing.
- 7 On the right, right-click, and then from the shortcut menu, select Paste.

Each drawing contains a material named Global.

You cannot delete or rename this material.

Getting drawing information

PTCAD stores accurate, detailed information about all the entities in a drawing. You can get details about an existing drawing and its entities using the tools for measuring distances and calculating areas. You can create a table or external file that reports details about entities in one or more drawings. You can also track the amount of time you spend editing a drawing. This section explains how to:

- Measure distances along an entity.
- Measure distances and angles.
- Divide an entity into a number of equal segments.
- Calculate areas.
- Display information about entities in a drawing.
- Track the amount of time spent editing a drawing.
- Report and update information about entities in one or more drawings.

Many of the functions described in this section require that you set PTCAD to the Advanced experience level.

Topics in this chapter

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Specifying measurements and divisions

You can divide a line, arc, circle, polyline, ellipse, or spline into a number of equal segments or mark off intervals of a specific length along an entity. (Note that divide is not the same as break.) For example, you may want to place station-point markers every 50 feet along the centerline of a roadway or divide the plan view of a window into three equal-width sections of glass, placing a mullion at each division point.

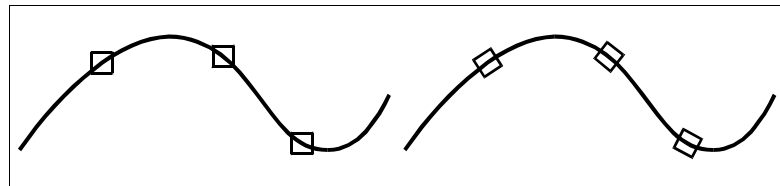
Understanding measurements and divisions

To specify measurements and divisions, use these commands:

- For the length of the segments, use the Measure command.
- For the number of equal-length segments, use the Divide command.

You can measure or divide arcs, circles, lines, polylines, ellipses, and splines. With either command, you can identify the segments by placing either a block or point entity at the end of each interval. If you use points, you can snap to the ends of intervals using the point entity snap. The appearance of the point entities is determined by the current point display type, which you control in the Drawing Settings dialog box.

To use a block as the marker, the block must already be defined in the current drawing. You can further indicate whether to rotate the block to align perpendicularly to the entity you are measuring or dividing.



Block not aligned with entity.

Block aligned with entity.

PTCAD begins measuring or dividing based on the point at which you select the entity and the type of entity with which you are working. For most entities, measuring starts from the endpoint closest to the point you used to select the entity. If you select the entity to be measured or divided using a method other than pointing (for example, using a window or fence selection), the program prompts you to specify the end from which you want to begin measuring.


Measuring intervals on entities

You can mark specified length increments along the length or perimeter of a selected entity using either a point entity or a block.

You can measure intervals on lines, arcs, circles, ellipses, polylines, and splines.

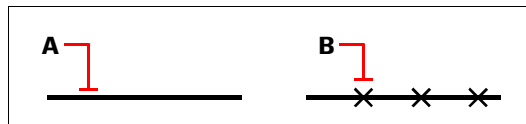
To measure intervals along an entity and mark them using point entities

Advanced experience level

- 1 Do one of the following to choose Measure ():
 - On the ribbon, choose Edit > Measure (in Modify).
 - On the menu, choose Modify > Measure.
 - On the Modify toolbar, click the Measure tool.
 - Type *measure* and then press Enter.
- 2 Select the entity.
- 3 Specify the segment length, and then press Enter.

Points on the entity might not be visible immediately.


You can change the style and size of all points in the drawing. On the Drawing Settings - Entity Creation tab, choose Points. Or type `ptype` in the command bar.



When you select the entity by pointing, intervals are measured from the end closest to the point at which you select the entity (A). Blocks or point entities (B) are placed along the entity at the specified interval.

To measure intervals along an entity and mark them using blocks

Advanced experience level

- 1 Do one of the following to choose Measure (- On the ribbon, choose Edit > Measure (in Modify).
- On the menu, choose Modify > Measure.
- On the Modify toolbar, click the Measure tool.
- Type *measure* and then press Enter.
- 2 Select the entity.
- 3 Choose Insert Blocks.
- 4 Type the name of the block you want to insert as the marker.
- 5 Choose either Yes-Align Blocks to rotate each insertion of the block so that its vertical alignment is always perpendicular to the entity or No-Do Not Align to insert each copy of the block with a zero rotation angle.
- 6 Specify the segment length, and then press Enter.


Dividing entities into segments

You can place markers along a selected entity, dividing that entity into a specified number of equal-length segments. You can use either a point entity or a block to mark the segments.

You can divide lines, arcs, circles, ellipses, polylines, and splines.

To divide an entity into equal segments and mark them using point entities

Advanced experience level

- 1 Do one of the following to choose Divide ():
 - On the ribbon, choose Edit > Measure (in Modify).
 - On the menu, choose Modify > Divide.
 - On the Modify toolbar, click the Divide tool.
 - Type *divide* and then press Enter.
- 2 Select the entity.
- 3 Specify the number of segments, and then press Enter.

Points on the entity might not be visible immediately.


You can change the style and size of all points in the drawing. On the Drawing Settings - Entity Creation tab, choose Points. Or type `ptype` in the command bar.



When you select the entity by pointing, divisions are marked beginning from the end closest to the point at which you select the entity (A). Blocks or point entities (B) are placed along the entity to mark it in equal intervals.

To divide an entity into equal segments and mark them using blocks

Advanced experience level

- 1 Do one of the following to choose Measure ():
 - On the ribbon, choose Edit > Measure (in Modify).
 - On the menu, choose Modify > Divide.
 - On the Modify toolbar, click the Divide tool.
 - Type *divide* and then press Enter.
- 2 Select the entity.
- 3 Choose Insert Blocks.
- 4 Type the name of the block you want to insert as the marker.
- 5 Choose either Yes-Align Blocks to rotate each insertion of the block so that its vertical alignment is always perpendicular to the entity or No-Do Not Align to insert each copy of the block with a zero rotation angle.
- 6 Specify the number of segments, and then press Enter.


Calculating areas

You can calculate the area and perimeter of entities or areas defined by points you specify. You can also determine the area of several combined entities and add or subtract the area of one or more entities from a total combined area.

Calculating areas defined by points

You can find the area and perimeter of any closed region by specifying a series of points. The program calculates the area and perimeter of the space that is enclosed by an imaginary polygon consisting of straight-line segments connecting each point.

To calculate the area defined by points you specify

- 1 Do one of the following to choose Area ():
 - On the ribbon, choose Tools > Area (in Inquiry).
 - On the menu, choose Tools > Inquiry > Area.
 - On the Inquiry toolbar, click the Area tool.
 - Type *area* and then press Enter.
- 2 Specify the first point.
- 3 Specify the second point.

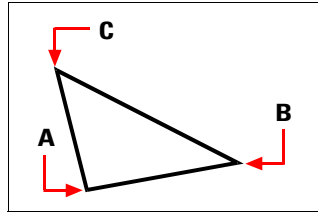
- 4 Continue specifying points in sequence to define the perimeter of the area you want to measure.

As you select each successive point, the resulting polygon is displayed on the screen.

- 5 To complete the calculation, press Enter.

The calculations of the area you defined are displayed. For example, the following type of information is displayed:

Area = 11.0583, Perimeter = 15.3092




Select the points (A, B, C) that form a polygon. The area and perimeter are then calculated.

Calculating areas of entities

You can find the area of entities, such as circles, arcs, ellipses, polylines, polygons, splines, 3D solids, facet model meshes, and planes.

In addition, the program calculates the perimeter of the entity, depending on the type of entity you select.

To calculate the area of an entity

- 1 Do one of the following to choose Area ():
 - On the ribbon, choose Tools > Area (in Inquiry).
 - On the menu, choose Tools > Inquiry > Area.
 - On the Inquiry toolbar, click the Area tool.
 - Type *area* and then press Enter.
- 2 Choose Entity.
- 3 Select the entity.


The following type of information is displayed:

Area = 62.3837, Perimeter = 27.9989

Calculating combined areas

You can find the total area of several entities or defined areas. You can also subtract the areas of entities or polygons from a running total.


To calculate the total area of entities

- 1 Do one of the following to choose Total Area (- On the ribbon, choose Tools > Total Area (in Inquiry).
- On the menu, choose Tools > Inquiry > Total Area.
- On the Inquiry toolbar, click the Total Area tool.
- Type *totalarea* and then press Enter.
- 2 Select the entities you want to calculate the area for.
The following type of information is displayed:


```
Total area = 62.3837 sq. units
```

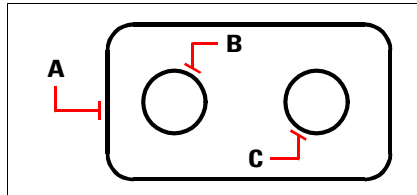
- 3 Continue selecting (or deselecting) entities to display the running total area.
- 4 When finished, press Enter.

To add areas to calculate a combined area

- 1 Do one of the following to choose Area (- On the ribbon, choose Tools > Area (in Inquiry).
- On the menu, choose Tools > Inquiry > Area.
- On the Inquiry toolbar, click the Area tool.
- Type *area* and then press Enter.
- 2 Choose Add.
- 3 Using one of the following methods, identify the first area:
 - Specify points defining a polygon, and then press Enter. Continue to specify points to define more areas to include in calculations.
 - Choose Entity, select the entities you want to add, and then press Enter to complete the calculations.
- 4 To complete the command, press Enter.

To subtract areas when calculating a combined area

- 1 Do one of the following to choose Area ():
 - On the ribbon, choose Tools > Area (in Inquiry).
 - On the menu, choose Tools > Inquiry > Area.
 - On the Inquiry toolbar, click the Area tool.
 - Type *area* and then press Enter.
- 2 Choose Add.
- 3 Using one of the following methods, identify the total area to subtract from:
 - Specify points defining a polygon, and then press Enter. Continue to specify points to define more areas to include in calculations
 - Choose Entity, select the entities you want to add to the total running area, and then press Enter to complete the calculations.
- 4 Choose Subtract.
- 5 Using one of the following methods, identify the area to be subtracted:
 - Specify points defining a polygon, and then press Enter. Continue to specify points to define more areas to subtract from the total running area.
 - Choose Entity, select the entities you want to subtract, and then press Enter to complete the calculations.
- 6 To complete the command, press Enter.



To calculate the area of the gasket using the Area command, first add the area of the entire gasket (A), and then subtract the areas of the two circles (B and C).

Calculating distances and angles

You can calculate the distance between any two points you select to determine the following information:


- The distance between the points, measured in drawing units.
- Their angle in the xy plane.
- Their angle measured from the xy plane.
- The change (delta) in the x, y, and z distances between the two points.

You can also determine the running total length of entities in a selection set.

Calculating the distance between two points or entities

When calculating the distance between points using the Distance command, it is often helpful to use entity snaps to specify precise points.

To calculate the distance between two points and their angle


- 1 Do one of the following to choose Distance (

Use the Distance command to calculate the distance (A) between two points (B and C), the angle in the xy plane (D), the angle from the xy plane, and the delta x (E), delta y (F), and delta z distances between the two points.

Calculating the total length of entities

You can calculate the length of any selected entities using the Total Length command.

To calculate the total length of entities

- 1 Do one of the following to choose Total Length (- On the ribbon, choose Tools > Total Length (in Inquiry).
- On the menu, choose Tools > Inquiry > Total Length.
- On the Inquiry toolbar, click the Total Length tool.
- Type *totallength* and then press Enter.
- 2 Select the entities you want to calculate the length for. The total length displays in the command bar.
- 3 Continue selecting entities to display the running total length.
- 4 When finished, press Enter.

Viewing calculated distance details

As you select entities, the program displays the calculations. If the command bar is displayed, the information appears there. If the command bar is not displayed, the program opens the Prompt History window and displays the calculations. The following type of information is an example of what is displayed:

```
Distance = 13.2850, Angle in XY Plane = 31°, Angle from XY Plane = 0°
Delta X = 11.3878, Delta Y = 6.8418, Delta Z = 0.0000
```

Displaying information about your drawing

You can display a variety of information about a drawing and the entities it contains, including:

- Information in the drawing database about selected entities.
- The current drawing status.
- The time spent working on the drawing.

This information is displayed in the Prompt History window and in the command bar.


Displaying information about entities

You can display information about the selected entities. The information varies, depending on the type of entities you select. All of the listings display the following information:

- Entity type.
- Layer.
- Color.
- Linetype.
- The location of the entity (its xyz-coordinates relative to the current user coordinate system [UCS]).
- The current space (model space on the Model tab or paper space on a Layout tab).
- The size of the entity (the information varies, depending on the entity type).

To display information about an entity

Advanced experience level

- 1 Do one of the following to choose List Entity Info (

Use a shortcut.

To return to the drawing window, press F2.

The following type of information is displayed:

```

----- Circle -----
Handle: 2C
Current space: Model
Layer: 0
Color: BYLAYER
Linetype: CONTINUOUS
Handle: 4C
Current space: Model
Center point: X= -5.8583 Y= 7.2752 Z= 0.0000
Radius: 4.4562
Circumference: 27.9989
Area: 62.383y


```

Displaying property information about external references and blocks

You can display property information about a selected external reference or block. All of the listings display the following information:

- Entity type
- Name
- Color
- Layer
- Linetype and linetype scale
- Print style
- Lineweight
- Material
- Transparency
- Thickness

To display property information about an external reference or block

- 1 Do one of the following to choose List Xref/Block Properties (- On the ribbon, choose Express Tools > List Xref/Block Properties (in Blocks).
- On the menu, choose Express Tools > Blocks > List Xref/Block Properties.
- Type *xlist* and then press Enter.
- 2 Select an external reference or block.


Displaying the drawing status

You can display information about the current status of a drawing, including:

- Drawing name.
- Limits.
- Insertion base point.
- Snap and grid settings.
- Current layer, color, and linetype.
- Current settings for various modes (fill, grid, orthogonal, snap, blips, and so on).

To display the drawing status

Advanced experience level

Do one of the following to choose Drawing Status (

- On the ribbon, choose Application button > Drawing Utilities > Drawing Status or choose Tools > Drawing Status (in Inquiry).
- On the menu, choose Tools > Inquiry > Drawing Status.
- On the Inquiry toolbar, click the Drawing Status tool.
- Type *status* and then press Enter.

The following type of information is displayed:

```
Current drawing name: Site Plan
      Drawing limits are: X=0.0000 Y=0.0000 Z=0.0000
      X=12.0000 Y=9.0000 Z=0.0000
Paper space limits are: X=0.0000 Y=0.0000 Z=0.0000
X=12.0000 Y=9.0000 Z=0.0000
Screen width (pixels): 971
Screen height (pixels): 569
Insertion base is: X=0.0000 Y=0.0000 Z=0.0000
Snap resolution is: X=0.5000 Y=0.5000 Z=0.0000
Grid spacing is: X:0.5000 Y:0.5000 Z=0.0000
Current layer: 0
Current color: BYLAYER
Current linetype: BYLAYER
Current elevation: 0.0000
Current thickness: 0.0000
Fill: on
Grid: off
Ortho: off
Snap: off
Blips: off
Drag: on
Command echo: on
Positive angle direction: Counterclockwise
Angular units: Decimal degrees
Dimension units: Decimal
Pick box height: 3
Entities in drawing: 288
```

Tracking time spent working on a drawing

You can display information about the amount of time you have spent working on a drawing, including:

- The date and time the drawing was created.
- The date and time the drawing was most recently saved.
- The total amount of time spent working on the drawing.
- The time spent working on the drawing during the current editing session.

You can turn the elapsed-time timer on and off or reset it to zero.

To display the timer information

Advanced experience level

- 1 Do one of the following to choose Time Variables (Ⓛ):
 - On the ribbon, choose Tools > Time Variables (in Inquiry).
 - On the menu, choose Tools > Inquiry > Time Variables.
 - On the Inquiry toolbar, click the Time Variables tool.
 - Type *time* and then press Enter.
- 2 Select any of the prompt box options:
 - Choose Timer On to turn the elapsed timer on.
 - Choose Timer Off to turn the elapsed timer off.
 - Choose Display Timer to redisplay the timer information.
 - Choose Reset Timer to reset the elapsed timer to zero.
- 3 In the prompt box, choose Cancel to exit the command.

Each time you display the timer information, the following type of information is displayed:

```
The current time is Fri Dec 19 09:58:43 1997
Drawing was created on: Wed 17 Dec 1997 at 16:17:59.8090
Drawing was last updated on: Thur 18 Dec 1997 at 09:58:43.3040
Total editing time: 1 2: 35:4.2345
Elapsed timer (on): 0 1: 21:5.6324
```

Reporting information about entities

Information about entities can be collected from one or more drawings into reports such as entity lists, schedules, and parts lists. The extracted information can include property settings, entity area calculations, coordinate positions, and more.


First you create a report template that contains the settings for the type of report you need, for example, you can include only block entities with annotative properties and control the order in which the columns display. Then the report template can be used to create a report for any drawing file.

Reports are created as tables in the current drawing or as external .xls or .csv files. If you create a table in the drawing with report data, the table contents can later be updated to include the latest information about referenced entities.

Creating a report

When you create a report, entity information is extracted from one or more drawings and can be included in the drawing as a new table or in an external .xls or .csv file.

To create a report

- 1 Do one of the following to choose Extract Data (- On the ribbon, choose Insert > Extract Data (in Data) or Annotate > Extract Data (in Tables).
- On the menu, choose Insert > Extract Data.
- Type *dataextraction* and then press Enter.
- 2 Choose one of the following:
 - **Create a New Report Template** Creates a report based on a new report template that you save as an .idxe file. To optionally create the new report template based on an existing one, mark Use Previous Template, then click [...] to select the report template (.idxe) file.
 - **Edit an Existing Report Template** Creates a report based on an existing report template that you can modify. Click [...] to select the report template (.idxe) file.
 - **Create Report from Existing Report Template** Creates a report for the current drawing based on an existing report template without modifications. Click [...] to select the report template (.idxe) file.
- 3 Click Next and make your selections. Click [?] for more details about each option.

Updating report data

Tables created with the Extract Data command contain data that is linked to the entities used to create them. As entities and drawings change, you can automatically update the contents of tables that contain report data.

To update report data in a table

- 1 Select any cell of a table that contains report data.
- 2 Right-click the cell.
- 3 Choose Data Extraction > Update Data Extraction.

Table contents can also be unlinked.

To remove the link between a table that contains report data and referenced entities, select any cell of the table, right-click the cell, then choose Data Extraction > Detach Data Extraction.

Modifying entities

PTCAD provides many editing tools for modifying a drawing. You can easily move, rotate, or stretch drawing entities, or change their scale. When you want to remove an entity, you can delete it with a few clicks of the mouse. You can also make multiple copies of any entity and copy entities from one drawing to another.

You can modify most entities using general-purpose editing commands. Some complex entities require special commands. This section explains how to:

- Select entities using entity-selection methods and grips.
- Change the properties of entities.
- Hiding, isolating, and deleting entities.
- Rearrange entities by moving, rotating, or changing the display order.
- Resize entities by stretching, scaling, extending, trimming, or editing their lengths.
- Split and combine entities by breaking, joining, exploding, and grouping them.
- Edit polylines.
- Create chamfers and fillets.

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Selecting entities

You can create a selection set that consists of one or more entities for modification. Use any of the following methods to create a selection set:

- Choose a command or tool first, and then select entities.
- Select entities first, and then choose a command or tool (most entities).
- Select entities by clicking while not in a command, and then use grips to modify them.

Understanding when to select entities

You can select entities before or after you choose a command.

Selecting entities first

When you select entities and then issue a command, the program immediately acts on the entities you've selected. For example, you can select an entity and its properties automatically display in the Properties pane or you can right-click the selection set to display a prompt box with various options to choose from.

After you select one or more entities, you can choose an entity-modification command, such as the Copy or Move command or use grips to modify the entities. You can also click the right mouse button to display a shortcut menu containing the entity-modification commands appropriate for the selected entities, and then choose the command from the menu.

You can control whether selection is allowed before using commands.

Choose Tools > Drawing Settings, click the Coordinate Input tab then the Entity Selection tab, and mark Pre-pick to allow selection before using commands.

If grips are turned on, grips also display for entities that are selected before issuing a command. For more information, see "Using grips" on page 401 in this chapter.

Choosing a command first

When you choose an entity-modification tool or command first, the program prompts you to select entities and displays a prompt box from which you can choose a selection method. You can select individual entities or use other techniques such as selection windows to select multiple entities.

When you select entities, you add them to the selection set. After you select at least one entity, you can remove entities from the selection set. To finish adding entities to the selection set, press Enter. Most entity-modification commands then act on the entire selection set.

Understanding entity-selection methods

When you choose a command that requires you to select entities (when you're deleting or modifying entities, for example), you can use any of the following selection methods by choosing them in the prompt box or entering them in the command bar:


Entity-selection methods

Selection method	Command bar	Description
Select all entities	ALL	Selects all entities in the current drawing.
Add to set	+ or A	Adds one or more entities to the selection set.
Subtract from set	- or R	Removes one or more entities from the selection set.
Previous selection	P	Selects entities included in the previous selection set.
Last entity in drawing	L	Selects the entity most recently added to the drawing.
Window-Inside	W	Selects entities contained entirely within a rectangular selection window.
Crossing window	C	Selects entities contained within or crossing the boundary of a rectangular selection window.
Outside window	O	Selects entities falling completely outside a rectangular selection window.
Window polygon	WP	Selects entities contained entirely within a polygon selection window.
Crossing polygon	CP	Selects entities contained within or crossing the boundary of a polygon selection window.
Outside polygon	OP	Selects entities falling completely outside a polygon selection window.
Window circle	WC	Selects entities contained entirely within a circular selection window.
Crossing circle	CC	Selects entities contained within or crossing the boundary of a circular selection window.
Outside circle	OC	Selects entities falling completely outside a circular selection window.
Point	PO	Selects any closed entities that surround the selected point.
Fence	F	Selects entities crossing a line or line segments.
Quick Select	QSELECT	Selects entities by type according to a specified value or range of values.
Select by Properties	PRO	Selects entities that match a particular set of properties—for example, all entities on a particular layer or drawn in a certain color.

You can also use a few selection methods automatically, without displaying the prompt box:

- Click one or more entities to select them.
- Shift+ click to add or remove an entity from the selection set.
- Click an entity to select it, then Ctrl + click to cycle through the entities below the cursor, selecting one at a time.
- Click two opposite corners of a rectangular selection window. The direction in which you define the points of the rectangle (left-to-right or right-to-left) determines which type of window you create.

Selecting multiple entities can be affected by the PICKADD setting.

If you select one or more additional entities while entities are already selected, the additional entities can either replace or add to the existing selection set. Click () on the Properties pane or use the PICKADD system variable to toggle what happens when selecting multiple sets of entities.

Selecting entities by clicking them

You can click entities to select them.

To select entities by clicking them

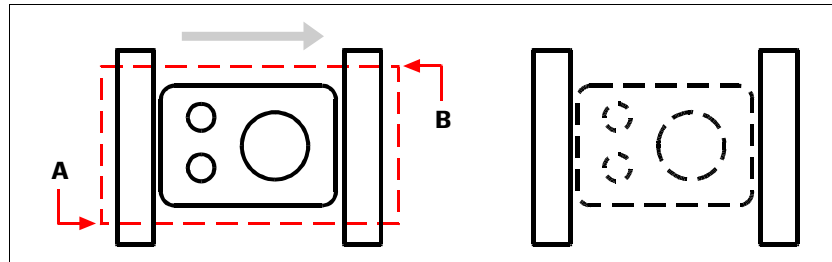
- 1 Click an entity.

Selecting entities by drawing a selection window

You can draw a selection window to include the entities contained within the window.

To create a selection window from left to right

- 1 Click to select a point in the drawing.
- 2 Click to the right of the first point to select a second point in the drawing.

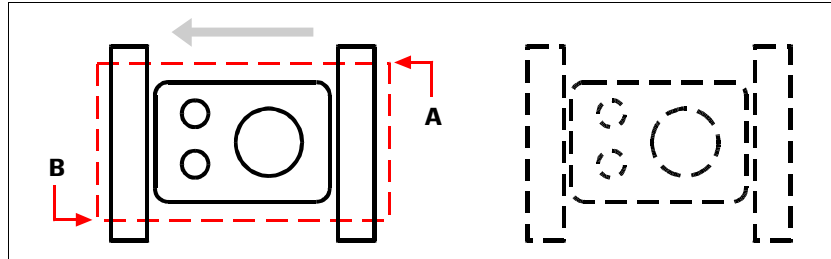


Window-Inside by selecting the first (A) and second (B) points.

Resulting selection.

To create a selection window from right to left

- 1 Click to select a point in the drawing.
- 2 Click to the left of the first point to select a second point in the drawing.



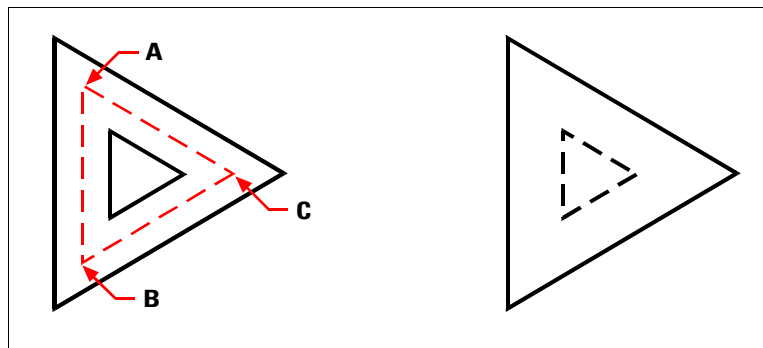
Crossing Window by selecting the first (A) and second (B) points.

Resulting selection.

In addition to a rectangular window, you can define a selection window using other shapes such as a polygon or circle.

To select entities using a polygon selection window

- 1 Activate an entity-modification command.
- 2 In the prompt box, choose Window Polygon.
- 3 Specify the vertices of the polygon.
- 4 To complete the selection polygon, press Enter.



Window Polygon by specifying the vertices of the polygon (A, B, and C).

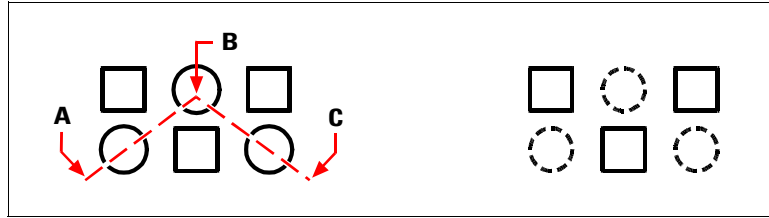
Resulting selection.

Selecting entities using a fence

A selection fence is a multisegmented line that selects entities it crosses.

To select entities using a fence

- 1 Activate an entity-modification command.
- 2 In the prompt box, choose Fence.
- 3 Specify the endpoints of the Fence segments.
- 4 To complete the Fence, press Enter.



Fence by specifying the endpoints of the fence segments (A, B, and C).

Resulting selection.

Filtering entity selection

Filtering a selection is an efficient way to select a set of entities that have something in common. For example, you can select all entities with the same property such as color, all entities that are proxies, or all entities with the same value, block name, or type. You can even add or remove more filters to a set of entities to further customize the selection.

Selecting entities by property

PTCAD makes it easy to select entities according to their common properties, allowing you to modify large sets of entities quickly and efficiently. When selecting entities, simply use a filter to add or remove entities according to these common properties: color, layer, linetype scale, linetype, lineweight, thickness, and width.

Use the List command to get more information for filtering.

If you are unsure what properties are available for filtering, type list to select an area of the drawing and list the selected entities and their properties.

To select entities by property using a properties filter

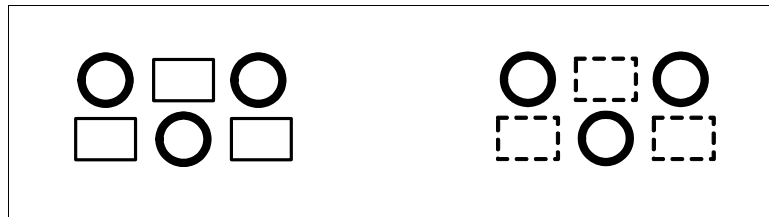
- 1 Activate an entity-modification command, or type *select*.
- 2 Choose Filter.
- 3 Choose a filter option:
 - Color — Enter the color of entities you want to select.
 - Layer — Enter the layer name of entities you want to select.
 - LinetypeScale — Enter the linetype scale of entities you want to select.
 - Linetype — Enter the linetype of entities you want to select.
 - Lineweight — Enter the lineweight of entities you want to select.
 - Thickness — Enter the thickness of entities you want to select. Note that some entities have thickness, however, lines, circles, arcs, and polylines all have thickness.
 - Width — Enter the width of entities you want to select. Note that only polylines have width.

Names of properties are case sensitive.

For example, a drawing with layers SAMPLE Layer 1, sample Layer 2, and SAMPLE Layer 3 will return no selection if you specify "SAMPLE" for the layer name. Specifying "SAMPLE" returns two layers, "*Layer*" returns all layers, and "sample*" returns one layer.*

- 4 If desired, add or remove more entities using a filter:
 - Choose any other filter option to add more entities to the selection set.
 - Choose Remove to remove entities from the selection set according to the filter you choose next.

A prompt displays the total number of entities in the selection set.
- 5 To complete the selection, press Enter.



Selection by specifying the smaller of two lineweight properties.

Resulting selection.

Selecting proxy entities using a filter

Proxy entities are entities or custom objects that PTCAD does not support. When a drawing containing proxy entities is loaded into PTCAD, a message displays indicating that some entities will not display, however, the entities reappear when you open the drawing later in a CAD application that supports those entities.

Proxy entities can be selected using typical selection methods, but they can also be selected using a filter, for example, you might want to select all proxies and place them on a hidden layer or delete them if you know they won't be needed in the future.

To select proxy entities using a filter

- 1 Activate an entity-modification command, or type *select*.
- 2 Choose Filter.
- 3 Choose Proxy.
- 4 If desired, add or remove more entities using a filter:
 - Choose any other filter option to add more entities to the selection set.
 - Choose Remove to remove entities from the selection set according to the filter you choose next.

A prompt displays the total number of entities in the selection set.
- 5 To complete the selection, press Enter.

Selecting blocks of the same name

Some drawings contain many of the same blocks, which are easy to select as a set using a filter.

Use the List command to get block names.

If you are unsure what blocks are available for filtering, type list to select an area of the drawing and list the selected entities and their block names.

To select entities by block name

- 1 Activate an entity-modification command, or type *select*.
- 2 Choose Filter.
- 3 Choose Block.
- 4 Enter the block name of the entities you want to select.
- 5 If desired, add or remove more entities using a filter:
 - Choose any other filter option to add more entities to the selection set.
 - Choose Remove to remove entities from the selection set according to the filter you choose next.

A prompt displays the total number of entities in the selection set.
- 6 To complete the selection, press Enter.

Selecting entities by type

You can filter entities in a selection set according to their type, for example, a circle, line, text, attribute, or block type.

Use the List command to get type names.

If you are unsure what entity types are available for filtering, type list to select an area of the drawing and list the selected entities and their types.

To select entities by type using the filter option

- 1 Activate an entity-modification command, or type *select*.
- 2 Choose Filter.
- 3 Choose Type.
- 4 Enter the type name (a string value) of the entities you want to select.
- 5 If desired, add or remove more entities using a filter:
 - Choose any other filter option to add more entities to the selection set.
 - Choose Remove to remove entities from the selection set according to the filter you choose next.A prompt displays the total number of entities in the selection set.
- 6 To complete the selection, press Enter.


Use the Selection pane and the Quick Select command to select entities by type.

You can also type selfilter or qselect to select entities by type.

Selecting entities by value

You can filter entities in a selection set according to common properties and their values. For example, you can filter a selection set to include all the entities that are the color red and use the Dashed2 linetype, and then change the value of the linetype.

To select entities by value using the Selection Filter pane

- 1 Do one of the following to choose Selection Filter (- On the ribbon, choose Edit > Selection Filter (in Selection) or View > Selection Filter (in Panes).
- On the menu, choose Edit > Selection > Selection Filter or View > Display > Selection Filter.
- Type *selfilter* and then press Enter.
- Press Ctrl+2.

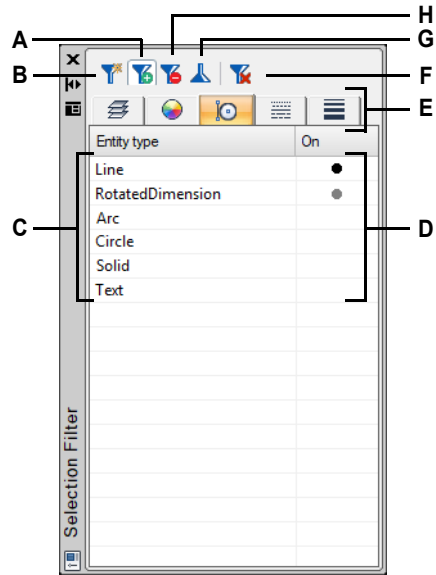
- 2 Click the Layer, Color, Entity Type, or Linetype tab.

The Selection Filter pane lists all layers, colors, entity types, and linetypes that are available in the drawing.

- 3 In the On column, select a corresponding cell to either include or exclude all entities with the corresponding value. The clicked cell then displays blank (selection is off) or with a black circle (selection is on), depending on its status before clicking it.



This can affect the On status of other values on other tabs. The On column for all tabs indicates the following:

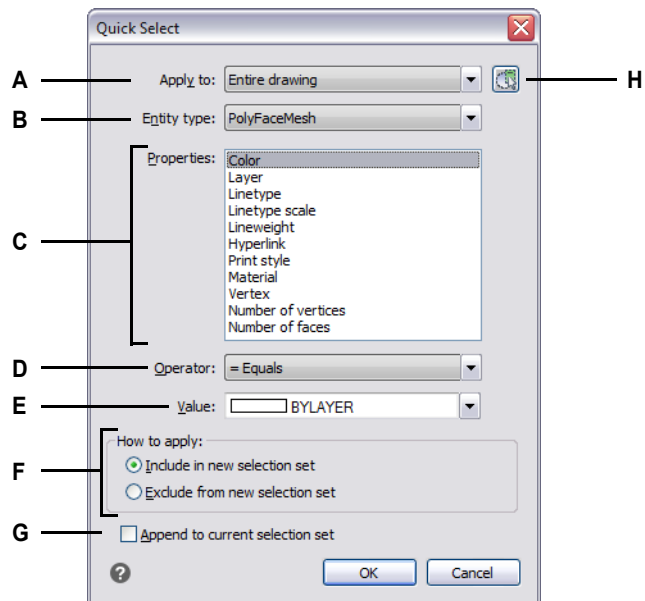
- **Blank** — No entities with the corresponding value are selected in the drawing.
- **Black circle** — All entities with the corresponding value are selected in the drawing.
- **Gray circle** — Only some entities with the corresponding value are selected in the drawing (due to specified values on other tabs or your entity selections in the drawing).



- A** Click to include a new entity in the current selection set.
- B** Click to start a new selection. After you select an entity, the previous selection is cleared.
- C** Displays all layers, colors, entity types, or linetypes that are available in the drawing depending on the selected tab.
- D** Click to change the selection to include or exclude all entities with the corresponding value.
- E** Click a tab to view and select entities in the drawing according to their assigned layer, color, entity type, linetype, or lineweight.
- F** Click to deselect all entities.
- G** With entities selected, click then window-select the currently selected entities and the additional entities you want to invert selection for.
- H** Click to exclude an entity from the selection set.

To select entities by value using Quick Select

- 1 Do one of the following to choose Quick Select ():
 - On the ribbon, choose Home > Quick Select (in Utilities) or Edit > Quick Select (in Selection).
 - On the menu, choose Edit > Selection > Quick Select or Tools > Quick Select.
 - On the Properties pane or Save Block to Disk dialog box, click the Quick Select tool.
 - Type *qselect*.
- 2 In Apply To, specify which entities to consider for selection. For example, select Entire Drawing to consider all entities in the drawing. To specify a portion of the drawing to consider for selection, click () and make your selection directly in the drawing.
- 3 In Entity Type, specify the type of entity you want to select.
- 4 Specify the property to filter for selection, its operator, and value. The options vary by entity type.
- 5 Select one of the following:
 - **Include in New Selection Set** Creates a new selection that includes only those entities that meet the selected options.
 - **Exclude from New Selection Set** Creates a new selection set that includes all of the entities except those that meet the selected options.
- 6 To add the newly selected entities to a current selection set (available if entities were selected before using the Quick Select command), mark Append to Current Selection Set.
- 7 Click OK.



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|--|---|
| <p>A Choose which entities to consider for selection.</p> <p>B Choose the type of entity to select.</p> <p>C Choose the desired property.</p> <p>D Choose how to compare the value and property.</p> <p>E Choose the desired value.</p> | <p>F Choose whether to include or exclude selected entities in the new selection set.</p> <p>G Select to add the newly selected entities to a previously specified selection set (if available).</p> <p>H Click to specify a portion of the drawing to consider for selection. The option Current Selection will appear in Apply To.</p> |
|--|---|

To select entities by value using the filter option

- 1 Activate an entity-modification command, or type *select*.
 - 2 Choose Filter.
 - 3 Choose Value.
 - 4 Enter the value (a string) of the entities you want to select.
 - 5 If desired, add or remove more entities using a filter:
 - Choose any other filter option to add more entities to the selection set.
 - Choose Remove to remove entities from the selection set according to the filter you choose next.
- A prompt displays the total number of entities in the selection set.
- 6 To complete the selection, press Enter.

Use the List command to get values.


If you are unsure what values are available for filtering, type list to select an area of the drawing and list the selected entities and their values.

Selecting entities that match the properties of other entities

You can select all entities in a drawing that have the same color, layer, linetype, linetype scale, lineweight, print style, entity style, name, entity type, or any combination of these properties. First you select one or more entities that have the properties you want to match, then you specify which properties to search for and match. The matched entities are then added to the selection set.

The selection set can be reused in other commands either through the pick-first selection or by selecting the previous selection.

To select entities that match one or more properties of other entities

- 1 Do one of the following to choose Select Similar (- On the ribbon, choose Edit > Select Similar (in Selection) or Express Tools > Select Similar (in Tools).
- On the menu, choose Edit > Selection > Select Similar or Express Tools > Tools > Select Similar.
- Type *selectsimilar* and then press Enter.
- 2 Select one or more entities that have the properties you want to match.
- 3 Choose Settings.
- 4 In the Select Similar Settings dialog box, mark the properties of the selected entities that you want to match.

The SELECTSIMILARMODE system variable also controls the dialog box settings.


See the System Variables online help for details about using this system variable.

- 5 Click OK.
Entities with matched properties are added to the selection set.


Creating filters used to select entities

Creating a filter that you can reuse is helpful if you regularly filter entities using the same parameters. Reusable filters are as easy to create even for complicated selection sets.

To create a filter used to select entities

- 1 Do one of the following to choose Filter ():
 - On the ribbon, choose Edit > Filter (in Selection).
 - On the menu, choose Edit > Selection > Filter.
 - Type *filter* and then press Enter.
- 2 In Select Filter, select an item for the filter list. For example, choose Line to include line entities in the selection set.
- 3 Specify the item's parameters if necessary. For example, if you chose Linetype, click Select to choose the linetype; if you chose Line End, enter the x-, y-, and z-coordinates.
- 4 Click Add to List.
- 5 Repeat for additional filter items, and include operators as necessary:
 - **AND** — Add BEGIN AND before one or more items to add them; add END AND after the items.
 - **OR** — Add BEGIN OR before one or more items to compare them and include only one; add END OR after the items.
 - **XOR** — Add BEGIN XOR before multiple items to specify exclusions; add END XOR after the items.
 - **NOT** — Add BEGIN NOT before one item to specify an exclusion; add END NOT after the item.
- 6 In Save As, enter a filter name.
- 7 Click Save As.

To select entities using a filter you created

- 1 Do one of the following to choose Filter ():
 - On the ribbon, choose Edit > Filter (in Selection).
 - On the menu, choose Edit > Selection > Filter.
 - Type *filter* and then press Enter.
- 2 In Current, select the desired filter.
- 3 Click Apply.



- A Lists the parameters that define the filter.
- B Select an item to include in the filter.
- C Enter the x-, y-, and z-coordinates if applicable to the filter item.
- D Click to add the specified item to the filter.
- E Click to add the currently selected entity in the drawing to the filter.
- F Click to replace the currently selected line in the filter list with the currently specified item.
- G Click to specify a value for the selected filter item.
- H Click to delete the named filter that is selected in Current.
- I Click to save the filter.
- J Enter a name for the filter.
- K Select a previously saved filter to load it.
- L Click to edit the settings of the currently selected line in the filter.
- M Click to remove all line items from the filter.
- N Click to remove the currently selected line item from the filter.

Deselecting entities

If an entity is no longer needed in a selection set, you can deselect it to remove it from the selection set.

To remove an entity from the selection set

- 1 Press Shift, and then select the entity again.

Use a shortcut.

Pressing Shift while selecting entities using a crossing window removes all entities from the specified selection set.

To remove all entities from the selection set

- 1 Press Escape.

Using grips

To use grips for editing, you select one or more entities to display the grips, and then click a grip to make it active and use it. You do not need to enter a command to use grips.

Grips have different shapes which indicate different behaviors. For example, moving a line's arrow grip changes the line length but not the angle, and moving a line's square grip changes the endpoint — both the line length and angle.

There are two types of grips:


- Entity grips — Grips that display for each selected entity and allow you to quickly move and scale entities one-by-one.
- Selection set grips — Grips that display for an entire selection set and allow you to quickly move, scale, and rotate all selected entities. This type of grip can be helpful for schematic drawings where precise scale or sizing is not required.

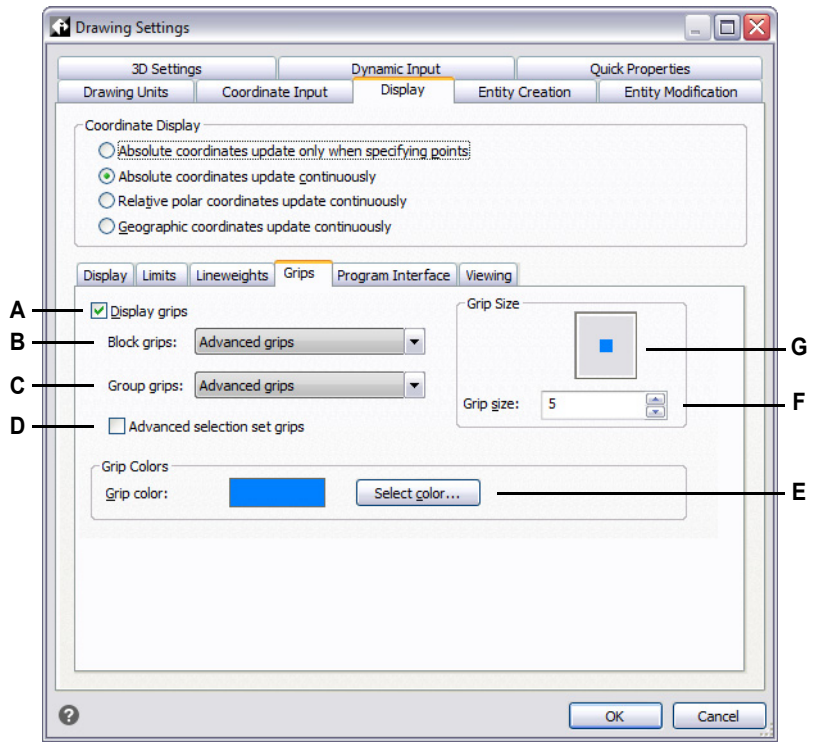
The grip you select depends on the type of grips that are active, the type of entity you're modifying, and the editing operation you're performing. For example, to move a line entity with entity grips active, drag it by its midpoint grip. To stretch the line, drag one of the endpoint grips. With selection grips active, however, you move a line entity by repositioning the base point grip.

Changing grip settings

You can turn the use of grips on and off, control the size and color of grips, and determine how grips display for blocks, groups, and selection sets.

To change grip settings

- 1 Do one of the following to choose Drawing Settings ():
 - On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
 - On the menu, choose Tools > Drawing Settings.
 - On the Tools toolbar, click the Drawing Settings tool.
 - Type *ddgrips* and then press Enter.
- 2 In the Drawing Settings dialog box, click the Display tab.
- 3 Click the Grips tab.
- 4 Mark Display Grips to activate grips.
 - In Block Grips, choose from the following:
 - **Insertion point grip** — Choose to enable a single grip that displays at the insertion point of the block.
 - **Grips within blocks** — Choose to enable grips for each entity within the block.
 - **Advanced grips** — Choose to enable a set of advanced grips to move, rotate and scale the whole block. Use side grips for non-uniform scaling; use corner grips for uniform scaling.
- 5 In Group Grips, choose from the following:
 - **Grips within groups** Choose to enable grips for each entity within the group.
 - **Center point grip** Choose to display a single grip at the center of the group.
 - **Center point grip and bounding box** Choose to display a single grip at the center of the group and a bounding box.
 - **Advanced grips** Choose to enable a set of advanced grips to move, rotate and scale the whole group.
- 6 Mark Advanced Selection Set Grips if you want to display a single set of advanced grips for all selected entities. These grips can be helpful for schematic drawings where precise scale or sizing is not required.
- 7 In Grip Size, specify the size of the grip square.
- 8 In Grip Color, select the color for grips.
- 9 Click OK.



- | | |
|---|--|
| <p>A Select to turn on the display of grips.</p> <p>B Select how you want grips to display for selected blocks.</p> <p>C Select how you want grips to display for selected groups.</p> | <p>D Select to turn on the display of advanced grips for selection sets.</p> <p>E Click to choose the color of grips.</p> <p>F Enter the size of the grip box.</p> <p>G Displays a preview of grips.</p> |
|---|--|

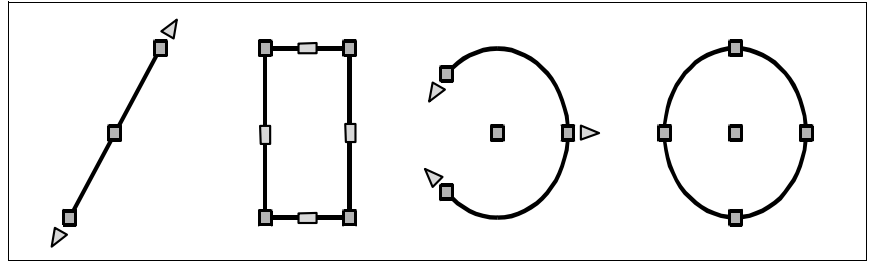
Using entity grips for editing

You can select entities while not in a command and then modify the entities using grips, which are small squares and arrows that display on each entity. Grips appear at strategic points on the entity according to the selected entity.

The locations of entity grips depend on the type of entity selected. For example, grips appear at the endpoints and midpoint of a line, at the quadrant points and center point of a circle, and at the endpoints, midpoint, and center of an arc.

Arrow grips are different from square grips. For example, the arrow grips on a line allow you to change the line length but at the same angle. Square grips at the end of lines allow you to change the endpoints, that is, the length and the angle.

Depending on your grip settings, blocks and groups can have unique grips that allow you to edit each entity within the block or group, or the entire block or group, including uniform and non-uniform scaling.



Examples of entity grip locations.

To edit while using entity grips

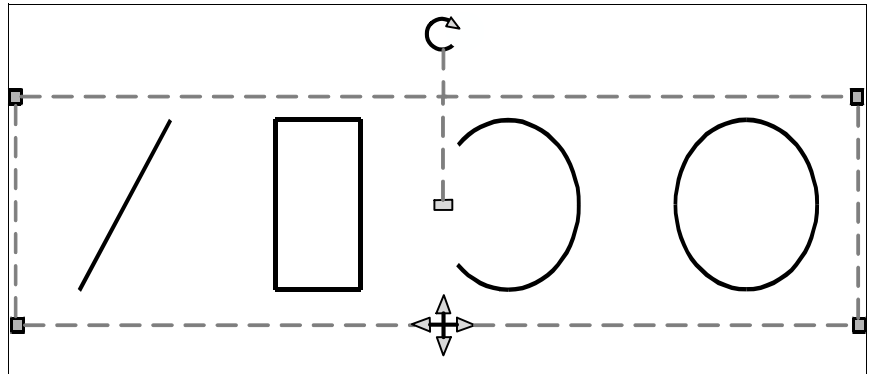
- 1 Make sure entity grips are turned on. For details, see “Changing grip settings” on page 402.
- 2 Click an entity to select it and display grips.
- 3 Click a grip to make it active.
- 4 Do any of the following:
 - Drag the grip to move it.
 - Choose a command, such as Base Point or Move.

Using selection grips for editing

With selection grips you can quickly move, scale and rotate all selected entities. Selection grips are particularly useful for schematic drawings where precise scale or sizing is not required. In addition, this standard interface for manipulating images and objects in Windows applications, can be helpful to users who may not be familiar with precision CAD editing interfaces.

Selection grips are similar to entity grips, but they display as a single set of grips for all entities in the selection set.

The locations of selection grips are uniform for the selection set. Grips at the corners of the selection set are for scaling, the bottom center grip is for moving, the top grip is for rotating, and the center grip is for positioning the base point. When a single block is selected, additional grips appear for non-uniform scaling for all three axis.



Example of selection grip locations. The rotation grip is located at the top and the move grip is located at the bottom center.


To edit using selection grips

- 1 Make sure selection grips are turned on. For details, see “Changing grip settings” on page 402.
- 2 Click an entity to select it and display selection grips. If you select more than one entity, a single set of selection grips displays for the entire selection set.
- 3 Click a grip to make it active.
- 4 Drag the grip to move it.

Customizing the display of selected entities

You can make selections easier to see by specifying whether to display selected entities with a dashed line or highlighted. By default, selected entities are highlighted.

To customize the display of selected entities

- 1 Do one of the following to choose Drawing Settings ():
 - On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
 - On the menu, choose Tools > Drawing Settings.
 - On the Tools toolbar, click the Drawing Settings tool.
 - Type *dsettings* and then press Enter.
- 2 Click the Display tab.
- 3 Click the secondary Display tab.
- 4 Select or clear the Highlight Item When Selected check box.
- 5 Click OK.

Modifying the properties of entities

You can change the layer, thickness, linetype, color, and linetype scale of one or more entities. Depending on the type of entity or entities you select, you can also change other properties, such as the start point and endpoint of lines, the center point and radius of circles, and the vertices of polylines.


The typical way to change entity properties is using the Properties and Quick Properties panes. You can also change general properties using the Entity Properties toolbar. Note that the Entity Properties toolbar settings that display when no entities are selected determine the properties of new entities when you draw them.

Modifying entity properties by using the Properties pane

Use the Properties pane to select properties for any entity. The options that display in the Properties pane depend on the type of entity that is selected and whether more than one entity is selected.

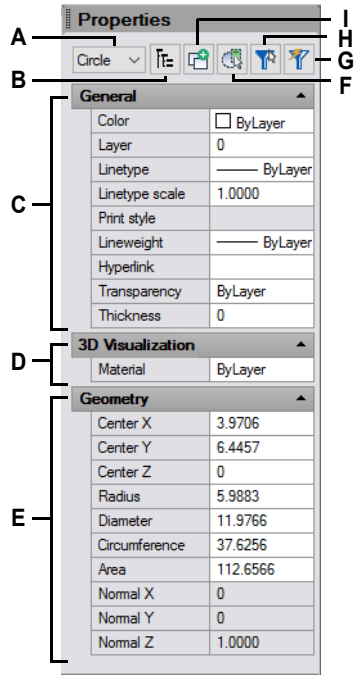
If no entities are selected, the Properties pane displays various default settings for the drawing that you can also modify.

To use the Properties pane to modify properties of entities

- 1 Do one of the following to choose Properties ():
 - On the ribbon, choose View > Properties (in Panes).
 - On the menu, choose View > Display > Properties or Modify > Properties.
 - On the Modify toolbar, click the Properties tool.
 - Type *entprop* and then press Enter.
 - Press Ctrl +1.

The Properties pane displays.

- 2 Select the desired entities.
- 3 Make changes to the properties. The available properties vary according to the entities you selected.



- A** Displays the types of entities in the selection set.
- B** Click to display entity types as a flat list or a hierarchical list.
- C** Specify properties that are common for most entities.
- D** Specify properties for 3D visualization.
- E** Specify properties specific to the selected geometry.
- F** Click to clear any previous selection and start selecting entities in the drawing.
- G** Click to select entities by type according to specified values.
- H** Click to display or hide the Selection Filter pane.
- I** Click to determine whether the current selection set is replaced by or added to subsequent entities that you select.

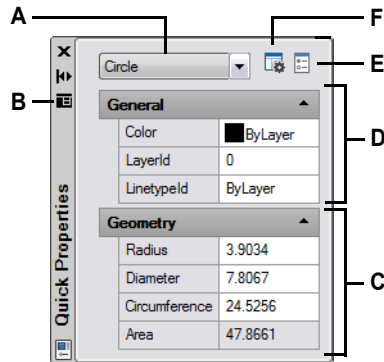
Modifying entity properties by using the Quick Properties pane

Compared with the Properties pane which typically displays in a large size and remains open until you close it manually, the Quick Properties pane displays in a smaller size and opens and closes dynamically. This allows you to access entity-specific properties dynamically when you need them without crowding available screen space. The pane displays automatically when you double-click an entity that does not have a double-click action assigned to it (for example, multiline text), and it closes automatically when you choose a subsequent command or action, or press Esc. The pane can also be set up to display automatically when you select an entity using a single-click, crossing window, etc. when a command is not active.

The Quick Properties pane generally doesn't display all properties of an entity, but you can easily open the Properties pane directly from the Quick Properties pane to work with all properties, if needed. The list of properties available in the Quick Properties pane for each entity type is customizable using the Customize UI command. For more details, see "Customizing the Quick Properties pane" on page 914.


To use the Quick Properties pane to modify properties of entities

- 1 Do one of the following to display the Quick Properties pane:
 - Double-click an entity.
 - Select entities using any selection method. The Quick Properties pane displays if it is turned on for selection; press Ctrl + Shift + P, choose Quick Properties On/Off in the status bar when no entities are selected, or type *dsettings* and use the Quick Properties tab.
 - Type *quickproperties*, select entities, then press Enter.
- 2 Make changes to the properties. The available properties vary according to the entities you selected and the properties specified to display in the Quick Properties pane for each entity type.



- A** Displays the types of entities in the selection set. Select from the list to change properties for an entity type.
- B** Click to display a menu of options, including pane location and settings that determine how quick properties work.
- C** Specify properties specific to the selected geometry.
- D** Specify properties that are most common.
- E** Click to open the Properties pane and view a complete set of properties for selected entities. Available only if the Properties pane is not displayed.
- F** Click to open the Customize User Interface dialog box to manage the properties that display in this pane for each entity type.

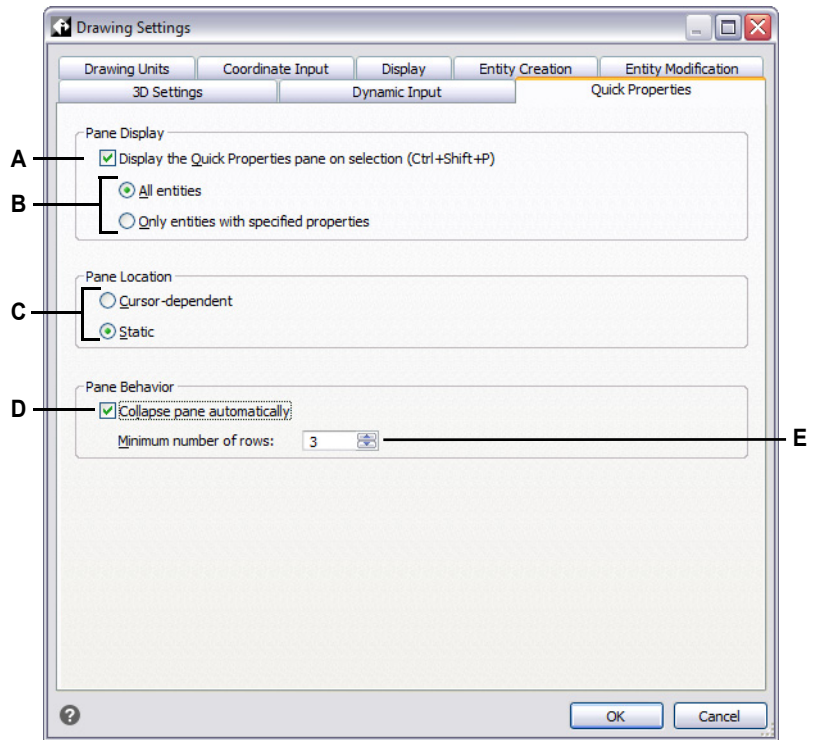
To change the behavior of the Quick Properties pane

- 1 Do one of the following to choose Drawing Settings ():
 - On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
 - On the menu, choose Tools > Drawing Settings.
 - On the Tools toolbar, click the Drawing Settings tool.
 - Type *dsettings* and then press Enter.
 - On the status bar, right-click Quick Properties On/Off, then choose Settings.
- 2 Click the Quick Properties tab.
- 3 To turn on the display of the Quick Properties pane when an entity is selected in the drawing (in addition to double-clicking):
 - Mark Display the Quick Properties Pane on Selection.
 - Choose All Entities to display the pane when any entity is selected or choose Only Entities with Specified Properties to display the pane only for entities that are specified in the Customize User Interface dialog box. For more details, see “Customizing the Quick Properties pane” on page 914.

Use a shortcut.

*To change whether the Quick Properties pane displays on selection, press **Ctrl + Shift + P** or click **Quick Properties On/Off** in the status bar when no entities are selected.*

- 4 To specify the default position of the Quick Properties pane when it displays, choose one of the following:
 - **Cursor-dependent** — Displays relative to the cursor location, below and to the right of the cursor.
 - **Static** — Displays in a fixed location without reference to the cursor. To specify a new location, click and drag the pane. The next time the pane opens, it displays at the same location.
- 5 To display a collapsed version of the Quick Properties pane until the cursor hovers over it, when it then automatically expands to display all defined quick properties for the selected entity, do the following:
 - **Mark Collapse Pane Automatically.**
 - In **Minimum Number of Rows**, enter the number of properties to display when the pane displays as collapsed.
- 6 Click **OK**.



- A** Select to turn on the display of the Quick Properties pane when selecting entities.
- B** Choose whether the pane displays when any entity is selected or only for assigned entities. Available only when displaying the pane on selection is turned on.
- C** Select to display the pane relative to the cursor location or in a fixed location.
- D** Select to display a collapsed version of the pane until the cursor hovers over it.
- E** Enter the number of properties to display when the pane displays as collapsed. Available only when displaying the pane as collapsed is turned on.

Modifying the properties of multiple entities

You can modify all the properties of all selected entities simultaneously. For example, using the *entprop* command, select all entities on a particular layer, and then move the entities to another layer by simply selecting a name from the Layer text box. Additionally, using the *select* command and the Select by Properties option, select all blue entities and change their color to green.


In the Properties pane, changes that you make in the Layer, Color, Thickness, Lineweight, Linetype, Linetype Scale, and Print Style (if using named print style tables) fields affect all selected entities. To change the properties of a single entity in the selection set, choose the entity from the list at the top of the Properties pane.

You can select the entities to be changed using any entity-selection method.

Changing multiple properties to ByLayer

The Set to ByLayer command sets properties of selected entities to ByLayer. Properties that can be changed using the Set to ByLayer command include: color, linetype, lineweight, material, print style, and transparency.

To modify properties of entities


- 1 Do one of the following to choose Set to ByLayer (- On the ribbon, choose Home > Set to ByLayer (in Layers).
- On the menu, choose Format > Layer Tools > Set to ByLayer.
- On the Layer Tools toolbar, click the Set to ByLayer tool.
- Type *setbylayer* and then press Enter.
- 2 Press Enter.
- 3 In the SetByLayer Settings dialog box, make your selections for the properties you want to change:
 - Color — Resulting entities will have a Color property set to ByLayer.
 - Linetype — Resulting entities will have a Linetype property set to ByLayer.
 - Lineweight — Resulting entities will have a Lineweight property set to ByLayer.
 - Material — Resulting entities will have a Material property set to ByLayer.
 - Print Style — Resulting entities will have a Print Style property set to ByLayer.
 - Transparency — Resulting entities will have a Transparency property set to ByLayer.
- 4 Click OK.
- 5 At the prompt, select the desired entities.
- 6 Choose Yes to change any found ByBlock properties to ByLayer. Otherwise, choose No.
- 7 Choose Yes to change the properties of blocks that are selected. Blocks must be on unlocked layers. Otherwise, choose No.

Hiding and isolating entities


You can temporarily hide selected entities or isolate selected entities so that all remaining entities are temporarily hidden. When you're ready to view all entities again, use the Unisolate Entities command.

You can also hide and isolate entities by assigning them a layer that you hide or isolate. For more details about layers, see "Organizing information on layers" on page 275.


To hide a selection set

- 1 Do one of the following to choose Hide Entities (- On the status bar, click Isolate/Unisolate Entities, and then choose Hide Entities.
- Type *hideobjects*, and then press Enter.
- 2 Select the entities you want to make invisible, and then press Enter.
All selected entities are hidden.

To isolate a selection set

- 1 Do one of the following to choose Isolate Entities (- On the status bar, click Isolate/Unisolate Entities, and then choose Isolate Entities.
- Type *isolateobjects*, and then press Enter.
- 2 Select the entities you want to remain visible, and then press Enter.
All other entities are hidden.

To unhide and unisolate entities

- 1 Do one of the following to choose Unisolate Entities (- On the status bar, click Isolate/Unisolate Entities, and then choose Unisolate Entities.
- Type *unisolateobjects*, and then press Enter.

Deleting entities

You can remove entities from a drawing. You can delete entities using any of the entity-selection methods.

To delete a selection set

- 1 Do one of the following to choose Delete (✖):
 - On the ribbon, choose Home > Delete (in Modify) or choose Edit > Delete (in Modify).
 - On the menu, choose Edit > Delete.
 - On the Standard (or Modify) toolbar, click the Delete tool.
 - Type *delete* and then press Enter.
- 2 Select the entities, and then press Enter.

Typing the Undo command restores the most recently deleted selection set.

If you have made additional modifications since deleting the entities, use Undo rather than Undo to restore those entities without reversing those modifications.

The Overkill command deletes extra entities all at once.

If a drawing has overlapping or duplicate entities that you want to delete or combine, for example polylines with too many vertices, use the Overkill command to clean up the drawing.

Copying entities

You can copy one or more entities, making one copy or multiple copies within the current drawing. You can also copy entities between drawings.

Use any of the following methods to copy entities within the current drawing:


- Create a copy at a location referenced from the original.
- Create a copy aligned parallel to the original.
- Create a copy as a mirror image of the original.
- Create several copies in a rectangular or circular pattern.

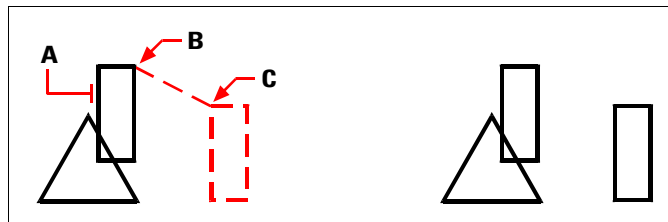
Copying entities within a drawing

You can duplicate entities within the current drawing. The default method is to create a selection set and then specify a starting point, or base point, and an endpoint, or displacement point, for the copy. You can also make multiple copies or copy the selection set to a location you specify, using a direction vector. You can also create an array of copied entities; for more details, see “Array entities” on page 423 in this chapter.

In addition to copying regular entities in a drawing, you can also copy entities that are contained within blocks, external references, and underlays.

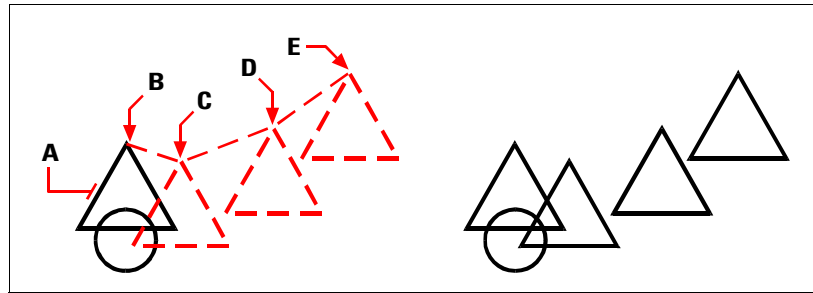
To copy a selection set

- 1 Do one of the following to choose Copy ():
 - On the ribbon, choose Edit > Copy (in Modify).
 - On the menu, choose Modify > Copy.
 - On the Modify toolbar, click the Copy tool.
 - Type *copy* and then press Enter.
- 2 Select the entities, and then press Enter.
- 3 Specify the base point.
- 4 Specify the insertion point.
- 5 Continue specifying insertion points to place additional copies.
- 6 To complete the command, press Enter.



To make a single copy of an entity, select the entity to copy (A), base point (B), and then displacement point (C).

Result.



To make multiple copies of an entity, select the entity to copy (A), specify the base point (B), and then specify the displacement points (C, D, and E).

Result.

Use a shortcut.

Press and hold Ctrl, then click and drag the left mouse button to copy a selected entity.

Use a system variable.

The COPYMODE system variable controls whether you are prompted for multiple copies.

Use the Move/Copy/Rotate command.

You can move, copy, rotate, and scale selected entities using a single command. Choose the Move/Copy/Rotate express tool or type moco.


To copy nested entities within blocks, external references, or underlays

- 1 Do one of the following to choose Copy Nested Entities (📄):
 - On the ribbon, choose Express Tools > Copy Nested Entities (in Blocks).
 - On the menu, choose Express Tools > Blocks > Copy Nested Entities.
 - Type *ncopy* and then press Enter.
- 2 Select the entities nested within blocks, external references, or underlays, and then press Enter.
- 3 Specify the base point.
- 4 Specify the insertion point.
- 5 Continue specifying insertion points to place additional copies.
- 6 To complete the command, press Enter.


Copying between drawings

You can use the Clipboard to cut or copy entities from one drawing to another. Cutting removes the selected entities from a drawing and stores them on the Clipboard. Copying duplicates the selected entities from a drawing and places them on the Clipboard. Pasting places the Clipboard contents in a drawing, and depending on the command you choose, at the same or different coordinates as the entities in the original drawing.

To cut entities to the Clipboard

- 1 Select the entities you want to cut.
- 2 Do one of the following to choose Cut (- On the ribbon, choose Edit > Cut (in Modify).
- On the menu, choose Edit > Cut.
- On the Standard toolbar, click the Cut tool.
- Type *cutclip* and then press Enter.

To copy entities to the Clipboard


- 1 Select the entities you want to copy.
- 2 Do one of the following to choose Copy to Clipboard (- On the ribbon, choose Home > Copy to Clipboard (in Clipboard).
- On the menu, choose Edit > Copy to Clipboard.
- On the Standard toolbar, click the Copy to Clipboard tool.
- Type *copyclip* and then press Enter.

Anything that you can copy to the Clipboard can be pasted into a drawing. The format in which the program adds the Clipboard contents to the drawing depends on the type of information in the Clipboard. For example, if you copy PTCAD drawing entities to the Clipboard, the program pastes them into the drawing as PTCAD entities. If you copy items to the Clipboard from other programs, they are pasted into the current drawing as embedded ActiveX® objects.


Sometimes the format you want to paste is not available on the Clipboard.

This is mostly likely due to the settings on the Clipboard tab in Tools > Options. For details, see “Changing the options on the Selection Cycling tab” on page 863.

To paste entities from the Clipboard

- 1 Do one of the following to choose Paste (- On the ribbon, choose Home > Paste (in Clipboard) or choose Edit > Paste (in Modify).
- On the menu, choose Edit > Paste.
- On the Standard toolbar, click the Paste tool.
- Type *pasteclip* and then press Enter.
- 2 Specify the insertion point.

To paste entities from the Clipboard to their original coordinates

- 1 Do one of the following to choose Paste to Original Coordinates (- On the ribbon, choose Edit > Paste to Original Coordinates (in Modify).
- On the menu, choose Edit > Paste to Original Coordinates.
- Type *pasteorig* and then press Enter.

Clipboard contents can also be inserted as a block.


Type pasteblock to convert Clipboard contents into a block upon insertion.

Copying between spaces

You can copy entities from model space to paper space or from paper space to model space. You must be viewing a Layout tab that has at least one layout viewport in order to copy entities between spaces. You can also move entities between spaces using the same Change Space command.

For more details about model space and paper space, see “Understanding paper space and model space” on page 616.

To copy entities between spaces

- 1 Click a Layout tab.
- 2 Select the entities you want to copy.
- 3 Do one of the following to choose Change Space (- On the ribbon, choose Edit > Change Space (in Modify).
- On the menu, choose Modify > Change Space.
- On the Modify toolbar, click the Change Space tool.
- Type *chspace* and then press Enter.
- 4 Choose Copy.


If using the command bar, the Copy option can get confused with crossing selections.

To use the Crossing selection method, type the full keyword “crossing”. Typing “c” calls the Copy option.

Cloning entities

Cloning an entity is similar to copying it, except you draw the new entity with its own geometric values. The entity type and properties (such as layer, color, linetype, line-weight, transparency, thickness) are copied and are the same as the entity that you initially select.

To clone an entity

- 1 Do one of the following to choose Clone (


When the command is finished, properties such as layer, color, etc., are reset to their default values.

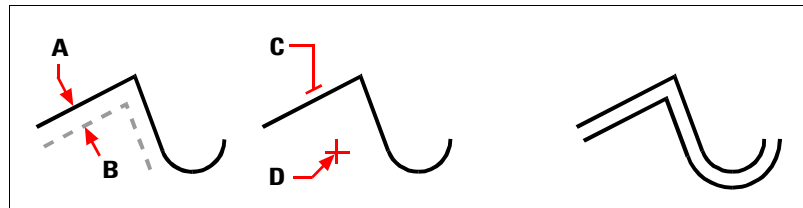
Making parallel copies

You can use the offset feature to copy selected entities and align them parallel to the original entities at a specified distance. You can make parallel, offset entities using arcs, circles, ellipses, elliptical arcs, lines, two-dimensional polylines, rays, and infinite lines.

Making parallel, offset copies of curved entities creates larger or smaller curves, depending on which side of the original entity you place the copy. For example, placing a parallel copy of a circle outside the circle creates a larger concentric circle; positioning the copy inside the circle creates a smaller concentric circle.

To make a parallel copy by specifying the distance


- 1 Do one of the following to choose Offset ():
 - On the ribbon, choose Edit > Offset (in Modify).
 - On the menu, choose Modify > Basic Editing > Offset.
 - On the Modify toolbar, click the Offset tool.
 - Type *offset* and then press Enter.
- 2 Specify the distance by selecting two points or by entering a distance.
- 3 Select the entity to copy.
- 4 Specify on which side of the entity to place the parallel copy.
- 5 Select another entity to copy, or press Enter to complete the command.

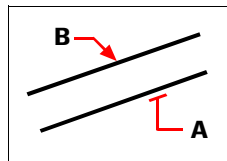


To make a parallel copy, specify the distance between copies by entering a distance or selecting two points (A and B), select the entity to copy (C), and specify on which side to place the copy (D).

Result.

To make a parallel copy passing through a point

- 1 Do one of the following to choose Offset ():
 - On the ribbon, choose Edit > Offset (in Modify).
 - On the menu, choose Modify > Basic Editing > Offset.
 - On the Modify toolbar, click the Offset tool.
 - Type *offset* and then press Enter.
- 2 In the prompt box, choose Through Point.
- 3 Select the entity to copy.
- 4 Specify the point for the entity to pass through.
- 5 Repeat steps 3 and 4, or press Enter to complete the command.



To make a parallel copy passing through a point, select the entity to copy (A) and then specify the through point (B).


Use additional offset options.

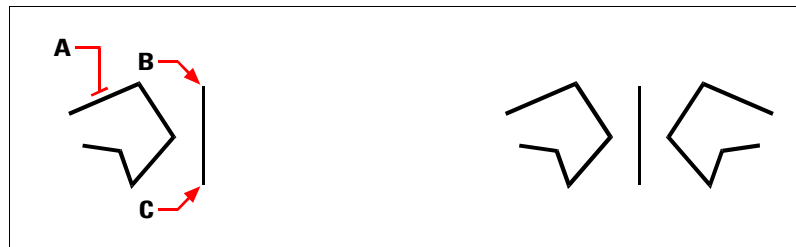
When first starting the Offset command, you can choose Erase to determine whether the source entity is deleted after it is offset and Layer to determine on which layer the offset entities are created.

Mirroring entities

You can create a mirror image of an entity. You mirror the entity about a mirror line, which you define by specifying two points in a drawing. You can delete or retain the original entities.

To mirror entities

- 1 Do one of the following to choose Mirror ():
 - On the ribbon, choose Edit > Offset (in Modify).
 - On the menu, choose Modify > Mirror.
 - On the Modify toolbar, click the Mirror tool.
 - Type *mirror* and then press Enter.
- 2 Select the entity, and then press Enter.
- 3 Specify the first point of the mirror line.
- 4 Specify the second point of the mirror line.
- 5 In the prompt box, choose one of the following:
 - Yes, Delete Entities – deletes the original entities.
 - No, Keep Entities – retains the original entities.



To mirror an entity, select it (A), and then specify the first point (B) and second point (C) of the mirror line.

Result.

Arraying entities


You can copy an entity in a pattern that creates an array. There are several ways to create arrays:

- **Linear array** — Use the Copy command to control the number of copies in the array and the line along which the copies are placed.
- **Rectangular array** — Use the Rectangular Array command to control the number of copies in the array by specifying the number of rows and columns. You also specify the distance between each row and column.
- **Polar (circular) array** — Use the Polar Array command to control the number of copies that compose the array and whether to rotate the copies.

- **Path array** — Use the Path Array command to control the number of copies that compose the array and whether to rotate the copies.

Rectangular, polar, and path arrays can be associative, which means the individual entities contained in the array relate to one another and behave more like a block or group. Associative arrays can be easily edited using the Edit Array command.

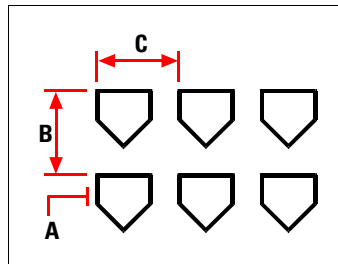
To create a linear array

- 1 Do one of the following to choose Copy (

To create a rectangular array


- 1 Do one of the following to choose Rectangular Array (

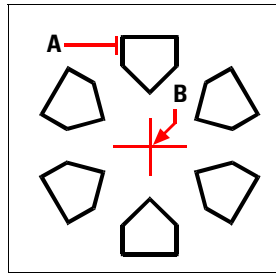
- 6 To specify the number of rows and columns (a rectangular array must have at least two rows *or* two columns), do any of the following:
 - Choose Count, enter the number of columns, then enter the number of rows.
 - Choose Columns, enter the number of columns, then specify the amount of horizontal space you want between each column in the array.
 - Choose Rows, enter the number of rows, then enter the amount of vertical space you want between each row in the array, then enter the increasing or decreasing elevation for each subsequent row.
- 7 To specify spacing between columns or rows directly in the drawing, do the following:
 - Choose Spacing.
 - Choose Unit cell.
 - Select two corners of a rectangle that represents the vertical and horizontal distance between rows and columns.
- 8 To specify the number of levels, choose Levels, enter the number of levels, then enter the amount of space you want between each level in the array. Positive and negative values create levels in opposite directions along the z-axis.



To create a rectangular array, select the entity to copy (A), type the number of rows and columns, and then specify the distance between each row (B) and column (C).


To create a polar array

- 1 Do one of the following to choose Polar Array (- On the ribbon, choose Home > Polar Array (in Modify) or Edit > Polar Array (in Modify).
- On the menu, choose Modify > Polar Array.
- On the Modify toolbar, click the Polar Array tool.
- Type *arraypolar* and then press Enter.
- 2 Select the entities to include in the array, and then press Enter.
- 3 Select the center point around which you want the entities arrayed.
- 4 Accept the default settings by pressing Enter again or continue with the next steps.
- 5 If you want to create an array with associative entities, choose Associative, then choose Yes.
- 6 To specify a new base point for the array, choose Base Point, then specify a new base point for the array in the drawing.
- 7 Specify two of the following values:
 - Items — Enter the number of items to create in the array, including one for the original selection set.
 - Fill angle — Enter the angle the array will fill: 0 to 360 degrees. The default setting for the angle is 360 degrees. Positive values create the array in a counter-clockwise direction; negative values create the array in a clockwise direction.
 - Angle between — Enter the angle between the arrayed items. The default setting is 90 degrees.
- 8 To specify the number of rows, do the following:
 - Choose Rows.
 - Enter the number of rows.
 - Enter the amount of vertical space you want between each row in the array.
 - Enter the incrementing elevation between rows. The default value 0 creates a two-dimensional array. Positive values create each subsequent row with an increasing elevation; negative values create each subsequent row with a decreasing elevation.
- 9 To set automatic orientation of items in the array, choose Rotate, then choose Yes. Automatic rotation orients subsequent items to match the angle of the path curve.
- 10 To specify the number of levels, choose Levels, enter the number of levels, then enter the amount of space you want between each level in the array. Positive and negative values create levels in opposite directions along the z-axis.

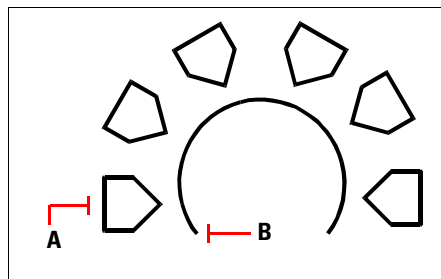


To create a polar array, select the entity to copy (A), specify the center point of the array (B), and then specify other options.

To create a path array



- 1 Do one of the following to choose Path Array ():
 - On the ribbon, choose Home > Path Array (in Modify) or Edit > Path Array (in Modify).
 - On the menu, choose Modify > Path Array.
 - On the Modify toolbar, click the Path Array tool.
 - Type *arraypath* and then press Enter.
- 2 Select the entities, and then press Enter.
- 3 Select a line, polyline, three-dimensional polyline, spline, helix, arc, circle, or ellipse to use for the path of the array.
- 4 Accept the default settings by pressing Enter again or continue with the next steps.
- 5 To specify a new base point for the array, choose Base Point, then specify a new base point for the array in the drawing.
- 6 To specify the distribution of array items along the path, choose Method, then do one of the following:
 - Choose Divide to place a specified number of copies that are evenly distributed along the entire length of the path.
 - Choose Measure to place copies along the path at a specified distance between them.
- 7 To specify the alignment of arrayed items relative to the start direction of the path, choose Tangent Direction, then do the following:
 - Specify the first point on the entity. Or choose Normal to align the z- direction of the first item with the starting direction of the path.
 - Specify the second point on the entity.

- 8 To specify the number of items in the array, choose **Items**. The prompts that follow depend on the setting selected in **Method**:
 - **Specify number of items**: Specify the number of copies of selected entities you want in the array. Displays if **Divide** is the **Method** setting.
 - **Specify the distance between items**: Enter the distance to use between arrayed entities. Displays if **Measure** is the **Method** setting.
 - **Enter number of items**: The maximum number of items required to fill the entire length of the path is automatically calculated. You can specify a smaller number of items if needed.
- 9 To specify the number of rows, do the following:
 - **Choose Rows**.
 - **Enter the number of rows**.
 - **Enter the amount of vertical space you want between each row in the array**.
 - **Enter the incrementing elevation between rows**. The default value 0 creates a two-dimensional array. Positive values create each subsequent row with an increasing elevation; negative values create each subsequent row with a decreasing elevation.
- 10 To specify the alignment of arrayed items relative to each other, choose **Align Items**, then choose **Yes** to make all items follow the alignment of the path, or choose **No** to specify a parallel alignment of arrayed items.
- 11 To maintain the z-direction of arrayed items, choose **Z Direction**, then choose **Yes** to make all items follow the z-direction of the path, or choose **No** to specify a parallel alignment of arrayed items in the z- direction.
- 12 To specify the number of levels, choose **Levels**, enter the number of levels, then enter the amount of space you want between each level in the array. Positive and negative values create levels in opposite directions along the z-axis.



To create a path array, select the entity to copy (A), select the path (B), and then specify other options.

To edit an associative array

- 1 Do one of the following to choose Edit Array (- On the ribbon, choose Home > Edit Array (in Modify) or Edit > Edit Array (in Modify).
- On the menu, choose Modify > Edit Array.
- On the Modify toolbar, click the Edit Array tool.
- Type *arrayedit* and then press Enter.
- 2 Select the array you want to edit.
- 3 To edit the array by modifying one of its arrayed items, do the following:
 - Choose Edit Source.
 - Select one of the items in the array.
 - Modify the item as you would modify any entity in the drawing. You can also create new entities to include them in the source array item.
- 4 To edit the array by replacing its items with other entities, do the following:
 - Choose Replace Item.
 - Select new source entities to use as a replacement.
 - Specify a base point for placing each replacement entity in the array.
 - Select the array item that you want to replace with the new source entities. Continue selecting array items for replacement. Or choose Source entities to replace all array items with the new source entities.
- 5 To reverse modifications, choose Reset Array, which reverses edits one by one.
- 6 Continue editing the array. The options vary depending on the type of array you are editing, and most editing options are the same as the array creation options described previously.
- 7 When finished, click Close Array () on the ribbon (if the ribbon is supported in your program version) or choose Exit.


Rearranging entities

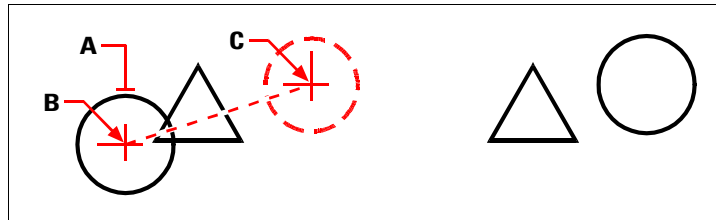
You can move one or more entities, and you can also rotate entities about a specified point. If you have entities that overlap, you can also change the display order.

Moving entities

You can move entities around within the current drawing or from one drawing to another. The default method is to create a selection set and then specify a starting point, or base point, and an endpoint, or displacement point, to define the relocation of the entities. You can also relocate the entities using a direction vector.

To move a selection set

- 1 Do one of the following to choose Move ():
 - On the ribbon, choose Edit > Move (in Modify).
 - On the menu, choose Modify > Move.
 - On the Modify toolbar, click the Move tool.
 - Type *move* and then press Enter.
- 2 Select the entities, and then press Enter.
- 3 Specify the base point.
- 4 Specify the displacement point.



To move an entity, select it (A), and then specify the base point (B) and the displacement point (C).

Result.

Use shortcuts.

Press Ctrl + arrow keys to move selected entities.

Click and drag the left mouse button to move selected entities. If you often move entities accidentally using the mouse, you can turn off this feature by unmarking Use Click-Drag to Move Selected Entities on the Display tab of the Drawing Settings dialog box.

Use additional commands to move entities.

You can move, copy, rotate, and scale selected entities using a single command. Choose the Move/Copy/Rotate command or type `mocoro`.

You can move entities interactively using a three-dimensional visual aid, which also works for two-dimensional entities. Choose the 3D Positioner command or type `3dpositioner`.

You can also move entities using grips. To move an entity using grips, select the entity to display its grips, and then click a grip and drag it. The grip you select depends on the type of entity you're modifying. For example, to move a line entity, select the midpoint grip. To move a curved entity, such as an arc, circle, or ellipse, select the center point grip. Not all entities can be moved using grips.

To move an entity using grips


- 1 Select the entity.
- 2 Click a grip to select it.
- 3 Drag the entity to where you want to relocate it.
- 4 Click to release.

Moving entities between spaces

You can move entities from model space to paper space or from paper space to model space. You must be viewing a Layout tab that has at least one layout viewport in order to move entities between spaces. You can also copy entities between spaces using the same Change Space command.

For more details about model space and paper space, see "Understanding paper space and model space" on page 616.


To move entities between spaces

- 1 Click a Layout tab.
- 2 Select the entities you want to move.
- 3 Do one of the following to choose :
 - On the ribbon, choose Edit > Change Space (in Modify).
 - On the menu, choose Modify > Change Space.
 - On the Modify toolbar, click the Change Space tool.
 - Type `chspace` and then press Enter.
- 4 Choose Move.

Rotating entities

You can rotate entities about a specified point at a specified rotation angle or by an angle referenced to a base angle. The default method rotates the entities using a relative rotation angle from their current orientation.

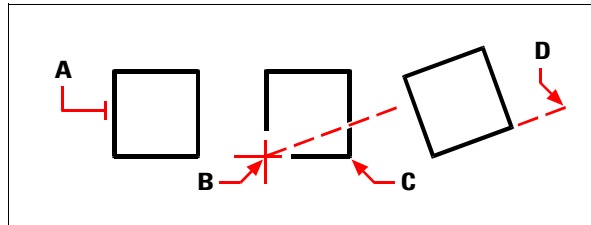
To rotate a selection set

- 1 Do one of the following to choose Rotate (

To rotate an entity, select the entity to rotate (A), and then specify the rotation point (B) and the rotation angle (C).

To rotate a selection set in reference to a base angle

- 1 Do one of the following to choose Rotate (



To rotate an entity in reference to a base angle, select the entity (A), specify the rotation point (B), select the base angle and pick point (B) again (or type the @ symbol), specify the second point (C), and then specify the point representing the new angle (D).

There are other ways to rotate entities too.

You can move, copy, and rotate selected entities using a single command. Choose the Move/Copy/Rotate command or type `mocoro`.

You can move entities interactively using a three-dimensional visual aid, which also works for two-dimensional entities. Choose the 3D Positioner command or type `3dpositioner`.

You can rotate single-line text, multiline text, attribute definitions, and block attribute text using an express tool, which has a convenient option for choosing the rotation that is easiest to read. Choose the Rotate Text command or type `orient`.


Reordering entities

When multiple entities overlap, you can change the order in which they are displayed and printed. You can move entities to the front, back, or on top or below of another entity.

You can also change the draw order of hatches so that they are drawn and printed behind all other entities in a drawing.

And for text, dimensions, and leaders, it is easy to move them to the front of other entities. If they are part of a block or external reference though, they are not moved to the front.

To reorder entities:


- 1 Do one of the following to choose Draw Order ():
 - On the ribbon, choose Home > Modify expander > Front, Back, Above, or Under; Edit > Front, Back, Above, or Under (in DrawOrder).
 - On the menu, choose Modify > Draw Order > Front, Back, Above, or Under.
 - On the Draw Order toolbar, click Front, Back, Above, or Under.
 - Type `draworder` and then press Enter.
- 2 Select the entity you want to reorder, and then press Enter.

- 3 In the prompt box, specify the new drawing order, and then press Enter.
- 4 If you are reordering above or under, select the entity you want the first entity to be above or below, and then press Enter.

The SORTENTS system variable automatically turns on.


The SORTENTS system variable automatically turns on, which may affect system performance.

To reorder hatches to the back:

- 1 Do one of the following to choose Send Hatches Back (- On the ribbon, choose Home > Modify expander > Send Hatches Back (in Draw Order tools) or choose Edit > Send Hatches Back (in Draw Order).
- On the menu, choose Modify > Draw Order > Send Hatches Back.
- On the Draw Order - Annotation toolbar, click the Send Hatches Back tool.
- Type *hatchtoback* and then press Enter.

The number of hatches affected is reported in the command bar.

To reorder text, dimensions, or leaders to the front:

- 1 Do one of the following to choose Text to Front (- On the ribbon, choose Home > Modify expander > Bring Text Front, Bring Leaders Front, Bring Dimensions Front, Bring All Annotations Front (in Draw Order tools); or Edit > Bring Text Front, Bring Leaders Front, Bring Dimensions Front, Bring All Annotations Front (in Draw Order).
- On the menu, choose Modify > Draw Order > Bring Text Front, Bring Leaders Front, Bring Dimensions Front, Bring All Annotations Front.
- On the Draw Order - Annotation toolbar, click the Bring Text Front, Bring Leaders Front, Bring Dimensions Front, or Bring All Annotations Front tool.
- Type *texttofront* and then press Enter.

Reversing the direction of entities

You can reverse the direction of lines, polylines, splines and helixes. The entity position and size does not change; only the vertices of the selected segments are swapped. This can be helpful if segments are assigned linetypes that include text or if polylines have varying start and end widths. If text is specified as upright, the text orientation is not changed.

To reverse the direction of entities:

- 1 Do one of the following to choose Reverse (


Resizing entities

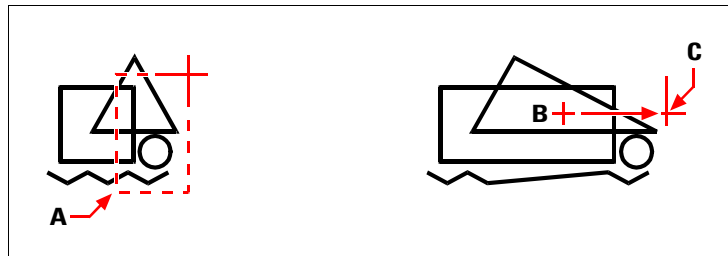
You can change the size of an entity or set of entities by stretching, scaling, extending, trimming, or editing their lengths.

Stretching entities

You can change the size of entities by stretching them. When you stretch entities, you must select the entities using either a crossing window or a crossing polygon. You then specify a displacement distance or select a base point and a displacement point. Entities that cross the window or polygon boundary are stretched; those completely within the crossing window or crossing polygon are simply moved.

To stretch an entity

- 1 Do one of the following to choose Stretch ():
 - On the ribbon, choose Home > Stretch (in Edit); Edit > Stretch (in Modify).
 - On the menu, choose Modify > Stretch.
 - On the Modify toolbar, click the Stretch tool.
 - Type *stretch* and then press Enter.
- 2 In the prompt box, choose Crossing Window or Crossing Polygon.
- 3 Select the entities, and then press Enter.
- 4 Specify the base point.
- 5 Specify the second point of displacement.

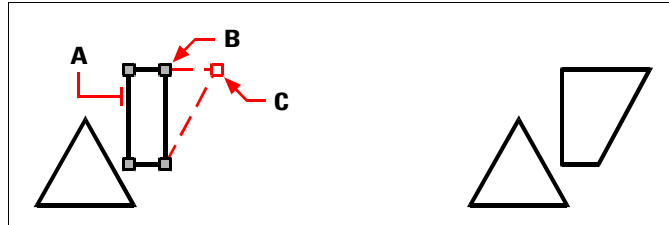


To stretch entities, select them using a crossing window (A) or crossing polygon, and then specify the base point (B) and displacement point (C).

To stretch an entity using grips, you select it to display its grips and then select a grip to make it the active grip. This becomes the base point. Then you move the active grip to a new location. The grip you select depends on the type of entity you're modifying. For example, to stretch one corner of a rectangle, select the corner point grip. To stretch a line, select an endpoint grip. Not all entities can be stretched using grips.

To stretch an entity using grips

- 1 Select the entity.
- 2 Click a grip to activate it.
- 3 Drag the grip.
- 4 Click to release.




To stretch an entity using grips, select the entity (A), select a grip (B), and drag the grip to its new location (C).

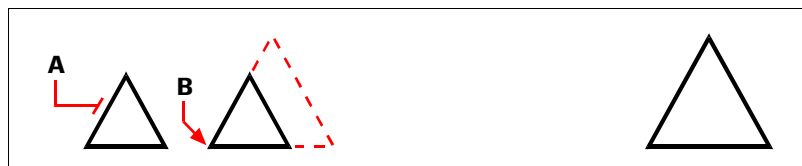
Result.

Scaling entities

You can change the size of a selected entity by scaling it in relation to a base point. You can change the size of an entity by specifying a base point and a length, which is used as a scale factor based on the current drawing units, or by specifying a scale factor. You can also use a scale factor referenced to a base scale factor, for example, by specifying the current length and a new length for the entity.

To scale a selection set by a scale factor

- 1 Do one of the following to choose Scale (- On the ribbon, choose Edit > Scale (in Modify).
- On the menu, choose Modify > Scale.
- On the Modify toolbar, click the Scale tool.
- Type *scale* and then press Enter.
- 2 Select the entities, and then press Enter.
- 3 Specify the base point.
- 4 Specify the scale factor.



To scale an entity by a scale factor, select the entity (A), and then specify the base point (B) and the scale factor.

Result.

You can also scale some entities using grips. To scale an entity, you select the entity, and then click a grip. You then change the size of the entity by moving the grip. The grip you select depends on the type of entity you're modifying. For example, to scale a circle, select a quadrant point grip.

To scale an entity using grips

- 1 Select the entity.
- 2 Click a grip to select it.
- 3 Drag the grip.
- 4 Click to release.



To scale using grips, select the entity (A), click a grip (B), and scale the entity by dragging the grip to its new location (C).

Result.

Use the Move/Copy/Rotate command.

You can move, copy, rotate, and scale selected entities using a single command. Choose the Move/Copy/Rotate express tool or type *MOCORO*.


To scale selected text entities use the Scale Text express tool or type *SCALETEXT*.

Extending entities

You can extend entities so that they end at a boundary defined by other entities. You can also extend entities to the point at which they would intersect an implied boundary edge. When extending entities, you first select the boundary edges, and then specify the entities to extend, selecting them either one at a time, using the fence selection method, or the projection selection method.

You can extend arcs, lines, two-dimensional polylines, and rays. Arcs, circles, ellipses, lines, splines, polylines, rays, infinite lines, and viewports on a Layout tab can act as boundary edges.

To extend an entity


- 1 Do one of the following to choose Extend ():
 - On the ribbon, choose Edit > Extend (in Modify).
 - On the menu, choose Modify > Extend.
 - On the Modify toolbar, click the Extend tool.
 - Type *extend* and then press Enter.
- 2 Select one or more entities as boundary edges, and then press Enter.
- 3 Select the entity to extend.
- 4 Select another entity to extend, or press Enter to complete the command.



To extend entities, select the boundary edge (A), and then select the entities to extend (B).

Result.


To extend an entity to an implied boundary

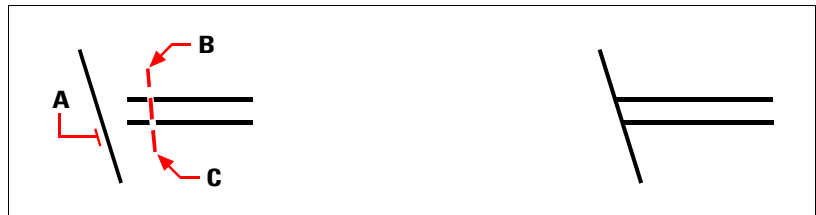
- 1 Do one of the following to choose Extend ():
 - On the ribbon, choose Edit > Extend (in Modify).
 - On the menu, choose Modify > Extend.
 - On the Modify toolbar, click the Extend tool.
 - Type *extend* and then press Enter.
- 2 Select one or more boundary edges, and then press Enter.
- 3 In the prompt box, choose Edge Mode.
- 4 In the prompt box, choose Extend.
- 5 Select the entity to extend.
- 6 Select another entity to extend, or press Enter to complete the command.



Select the boundary edge (A), and then select the entities to extend (B). Result.

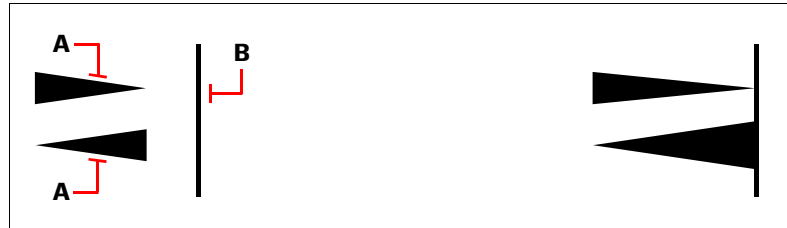
To extend several entities using the fence selection method

- 1 Do one of the following to choose Extend ():
 - On the ribbon, choose Edit > Extend (in Modify).
 - On the menu, choose Modify > Extend.
 - On the Modify toolbar, click the Extend tool.
 - Type *extend* and then press Enter.
- 2 Select one or more boundary edges, and then press Enter.
- 3 In the prompt box, choose Fence.
- 4 Specify the first point of the fence.
- 5 Specify the second point of the fence.
- 6 Specify the next fence point, or press Enter to complete the command.



Select the boundary edge (A), and then specify the first point (B) and second point (C) of the fence. Result.

When you extend a wide polyline, its centerline intersects the boundary edge. Because the end of the polyline is always cut at a 90-degree angle, part of the polyline may extend past the boundary edge. A tapered polyline continues to taper until it intersects the boundary edge. If this would result in a negative polyline width, the ending width changes to 0.



A tapered polyeline (A) continues to taper until it intersects the boundary edge (B).


Result.

Trimming entities

You can clip, or trim, entities so they end at one or more implied cutting edges defined by other entities. You can also trim entities to the point at which they would intersect an implied cutting edge. When trimming entities, you first select the cutting edges and then specify the entities to trim, selecting them either one at a time or using the fence selection method.

You can trim arcs, circles, lines, open two-dimensional and three-dimensional polylines, and rays. Arcs, circles, lines, polylines, rays, infinite lines, and viewports on a Layout tab can act as cutting edges. An entity can be both a cutting edge and one of the entities being trimmed.

To trim an entity


- 1 Do one of the following to choose Trim ():
 - On the ribbon, choose Edit > Trim (in Modify).
 - On the menu, choose Modify > Trim.
 - On the Modify toolbar, click the Trim tool.
 - Type *trim* and then press Enter.
- 2 Select one or more cutting edges, and then press Enter.
- 3 Select the entity to trim.
- 4 Select another entity to trim, or press Enter to complete the command.

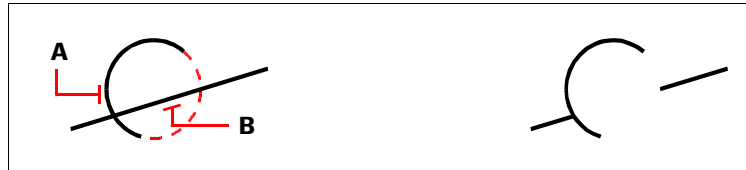


To trim entities, select the cutting edge (A), and then select the entities to trim (B).

Result.

To trim an entity to an implied boundary


- 1 Do one of the following to choose Trim ():
 - On the ribbon, choose Edit > Trim (in Modify).
 - On the menu, choose Modify > Trim.
 - On the Modify toolbar, click the Trim tool.
 - Type *trim* and then press Enter.
- 2 Select one or more cutting edges, and then press Enter.
- 3 In the prompt box, choose Edge Mode.
- 4 In the prompt box, choose Extend.
- 5 Select the entity to trim.
- 6 Select another entity to trim, or press Enter to complete the command.

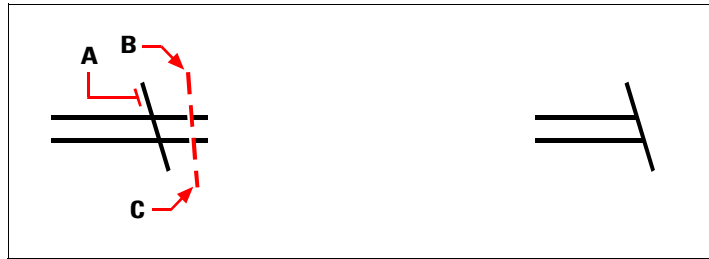


Select the implied boundary edge (A), and then select the entities to trim (B).

Result.

To trim several entities using the fence selection method

- 1 Do one of the following to choose Trim ():
 - On the ribbon, choose Edit > Trim (in Modify).
 - On the menu, choose Modify > Trim.
 - On the Modify toolbar, click the Trim tool.
 - Type *trim* and then press Enter.
- 2 Select one or more cutting edges, and then press Enter.
- 3 In the prompt box, choose Fence.
- 4 Specify the first point of the fence.
- 5 Specify the second point of the fence.
- 6 Specify the next fence point, or press Enter to complete the command.



Select the boundary edge (A), and then specify the first point (B) and second point (C) of the fence.

Result.

Use the Quick Trim command.

You can trim all entities to the edge of a cutting entity in a single operation. Choose the Quick Trim express tool or type `extrim`.


Editing the length of entities

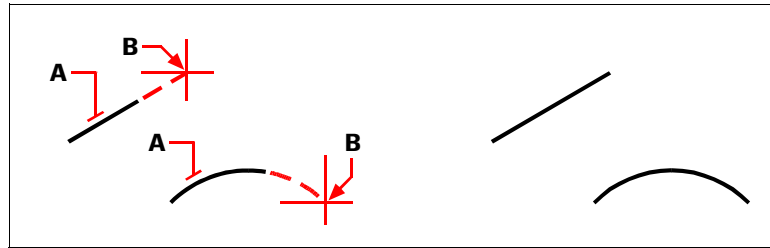
You can change the length of entities or the included angle of arcs. Use any of the following methods to change the length of an entity:

- Dynamically drag the endpoint or angle.
- Specify an incremental length or angle measured from an endpoint.
- Specify the new length as a percentage of the total length or angle.
- Specify a new length or included angle.

You can change the length of arcs, lines, and open polylines.

To change the length of an entity by dragging

- 1 Do one of the following to choose Edit Length ():
 - On the ribbon, choose Edit > Edit Length (in Modify).
 - On the menu, choose Modify > Edit Length.
 - On the Modify toolbar, click the Edit Length tool.
 - Type `editlen` and then press Enter.
- 2 In the prompt box, choose Dynamic.
- 3 Select the entity you want to change.
- 4 Specify the new endpoint or included angle.



Select the entity (A), and then select the new endpoint (B).

Result.

Splitting and combining entities

You can break and combine entities using the following methods:

- Break — Separate a single entity into two parts, removing a portion of the entity in the process.
- Break at Point — Separate a single entity into two parts at a single point, without removing a portion of the entity in the process.
- Join — Combine two entities into a single entity.
- Explode — Separate a complex entity, such as a block or polyline, into its various component parts.
- Group — Combine multiples entities to behave as a single unit.


Breaking entities

Use the Break command to split, or remove the ends from, lines, arcs, circles, ellipses, rays, infinite lines, polylines, and donuts. By default, the point you use to select the entity becomes the first break point; however, you can use the First option to select a break point different from the one that selects the entity.

When breaking entities, the original entity is split by removing the portion between the two points you specify. If the two points are the same or if you use the Split option, the entity is split without a gap between the resulting entities.

You can also use the Break at Point command to break an entity into two without removing a portion of the entity.

To break an entity by removing a portion of it


- 1 Do one of the following to choose Break ():
 - On the ribbon, choose Home > Break (in Modify) or Edit > Break (in Modify).
 - On the menu, choose Modify > Break.
 - On the Modify toolbar, click the Break tool.
 - Type *break* and then press Enter.
- 2 Select the entity.
- 3 Specify the second break point.

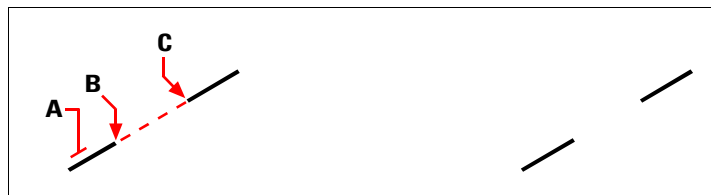


Select the entity (A), and then specify the second break point (B).

Result.

To select an entity and then specify the two break points


- 1 Do one of the following to choose Break ():
 - On the ribbon, choose Home > Break (in Modify) or Edit > Break (in Modify).
 - On the menu, choose Modify > Break.
 - On the Modify toolbar, click the Break tool.
 - Type *break* and then press Enter.
- 2 Select the entity.
- 3 In the prompt box, choose First.
- 4 Specify the first break point.
- 5 Specify the second break point.




Select the entity (A), and then specify the first (B) and second (C) break points.

Result.

To split an entity without removing a portion of it using the Break command

- 1 Do one of the following to choose Break ():
 - On the ribbon, choose Edit > Break (in Modify).
 - On the menu, choose Modify > Break.
 - On the Modify toolbar, click the Break tool.
 - Type *break* and then press Enter.
- 2 Specify the entity at the desired break point.
- 3 Do one of the following:
 - Choose Split.
 - Specify the same point for the second break point by typing the at sign (@) and pressing Enter.

To break an entity without removing a portion of it using the Break at Point command

- 1 Do one of the following to choose Break at Point ():
 - On the ribbon, choose Home > Break at Point (in Modify) or Edit > Break at Point (in Modify).
 - On the menu, choose Modify > Break at Point.
 - On the Modify toolbar, click the Break at Point tool.
 - Type *breakatpoint* and then press Enter.
- 2 Select an entity, such as a line, arc, polyline, infinite line, ray, or spline.
- 3 Select the point at which to break the selected entity.

Joining entities


You can join two entities into a single entity.

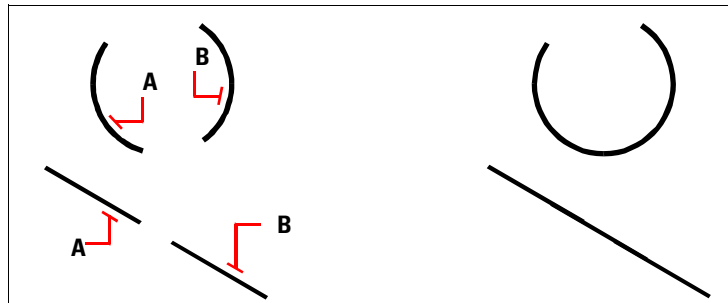
When you join two lines, they must be col-linear and the farthest endpoints remain at their existing locations; the program draws a new line between these points. Arcs must share the same center point and radius, and they are joined counterclockwise, from the first arc you select to the second.

Use the Edit Polyline command to join entities and polylines.

For more details, see “Joining polylines” on page 456 in this chapter.

To join two entities

- 1 Do one of the following to choose Join ():
 - On the ribbon, choose Edit > Join (in Modify).
 - On the menu, choose Modify > Join.
 - On the Modify toolbar, click the Join tool.
 - Type *join* and then press Enter.
- 2 Select the first entity.
- 3 Select the second entity.



Select the first arc or line (A), and then select the second arc or line (B).

Result.

The Overkill command can also combine entities.

If a drawing has overlapping or duplicate entities that you want to combine, for example polylines with too many vertices, you can use the Overkill command to clean up the drawing.

Exploding entities

You can convert a complex entity, such as a block or polyline, from a single entity into its component parts. Exploding a polyline, rectangle, donut, polygon, dimension, or leader reduces it to a collection of individual line and arc entities that you can then modify individually.

Blocks are converted to the individual entities, possibly including other, nested blocks that composed the original entity. When exploding blocks, you can also use the Explode Attributes command to ungroup blocks, creating separate entities for each element and converting attributes to text. The resulting exploded entities are placed on the block layer (not the layer of the original entities) while the exploded attributes are placed on the layer of the original attribute. The Explode Attributes command is best used if you want to explode a block with attributes and have it retain the same visual appearance for properties and text. However, if you want to explode the block to modify the block definition and redefine it, you should use the Explode command, not Explode Attributes.


Exploding remote text (rtxt) entities from drawings created in another CAD program converts them to multiline text entities.

Exploding multiline text or single-line text using the Text Explode command reduces the text to polylines.


With the following exceptions, exploding an entity usually has no visible effect on a drawing:

- If the original polyline had a width, the width information is lost when you explode it. The resulting lines and arcs follow the centerline of the original polyline.
- If you explode a block containing attributes, the attributes are lost, but the original attribute definitions remain.
- Colors, linetypes, lineweights, and print styles assigned BYBLOCK may be different after exploding an entity, because they will adopt the default color, linetype, lineweight, and print style until inserted into another block.


To explode an entity

- 1 Do one of the following to choose Explode (- On the ribbon, choose Home > Explode (in Modify) or choose Edit > Explode (in Modify).
- On the menu, choose Modify > Explode.
- On the Modify toolbar, click the Explode tool.
- Type *explode* and then press Enter.
- 2 Select the entities to explode.
- 3 Press Enter.


To explode multiline text or single-line text

- 1 Do one of the following to choose Explode Text ():
 - On the ribbon, choose Express Tools > Explode Text (in Text).
 - On the menu, choose Express Tools > Text > Explode Text.
 - Type *txexp* and then press Enter.
- 2 Select the text to explode.
- 3 Press Enter.

To explode a block and its attributes

- 1 Do one of the following to choose Explode Attributes ():
 - On the ribbon, choose Express Tools > Explode Attributes (in Blocks).
 - On the menu, choose Express Tools > Blocks > Explode Attributes.
 - Type *burst* and then press Enter.
- 2 Select the blocks that have attributes you want to explode.
- 3 Press Enter.

To explode an entity and specify properties of the resulting entities

- 1 Do one of the following to choose Xplode ():
 - On the ribbon, choose Edit > Xplode (in Modify).
 - On the menu, choose Modify > Xplode.
 - On the Modify toolbar, click the Xplode tool.
 - Type *xplode* and then press Enter.
- 2 Select the entities to explode.
- 3 Choose an option:
 - All — Displays prompts for specifying all available properties: color, layer, linetype, and lineweight.
 - Color — Enter a color. You can enter an index color, true color, or color from a colorbook.
 - Layer — Enter a layer for the resulting entities.
 - LType — Enter a linetype for the resulting entities.
 - LWeight — Enter a lineweight for the resulting entities.
 - Inherit — Explodes selected entities and assigns the same color, layer, linetype, and lineweight properties to sub-entities as the parent entity if the sub-entity layer is 0 and other properties are BYBLOCK.
 - Explode — Explodes selected entities in the same way as the Explode command.

Grouping entities

A group is a collection of entities saved together as one unit. After you select the entities that belong in the group, you can later add more entities, remove entities, and reorder the entities. If necessary, you can also ungroup the entities at any time to work with the entities separately.


Group selection can be customized.

You can control whether a group is selected when a sub-entity within the group is selected. On the ribbon if your program version includes one, choose Home > Group Selection (in Groups), on the menu choose Modify > Groups > Group Selection, or type `togglepickstyle`.

Creating groups

The easiest way to create a group is to use the Quick Group command and simply select the entities that you want to belong in the group. If you want to specify more options for the group such as a group name, description, and selectability, use the Group Manager command.


To create a group using the Quick Group command

- 1 Do one of the following to choose Quick Group ():
 - On the ribbon, choose Home > Quick Group (in Groups).
 - On the menu, choose Modify > Groups > Quick Group.
 - On the Groups toolbar, click the Quick Group tool.
 - Type `qgroup` and then press Enter.
- 2 Select the entities for the group, and then press Enter.
An unnamed group is created.

By default, groups created with Quick Group are unnamed.

If you later use the Group Manager command to work with unnamed groups, be sure to mark `Display unnamed groups`.

To create a group using the Group Manager command

- 1 Do one of the following to choose Group Manager ():
 - On the ribbon, choose Home > Group Manager (in Groups).
 - On the menu, choose Modify > Groups > Group Manager.
 - On the Groups toolbar, click the Group Manager tool.
 - Type `group` and then press Enter.
- 2 Under Create New Group, enter the name and description of the group.

- 3 Click Selectable if you want all entities in the group to be selected in the drawing when you select one entity of the group. If turned off for a group, the group selection setting has no affect.

Groups can also have grips.

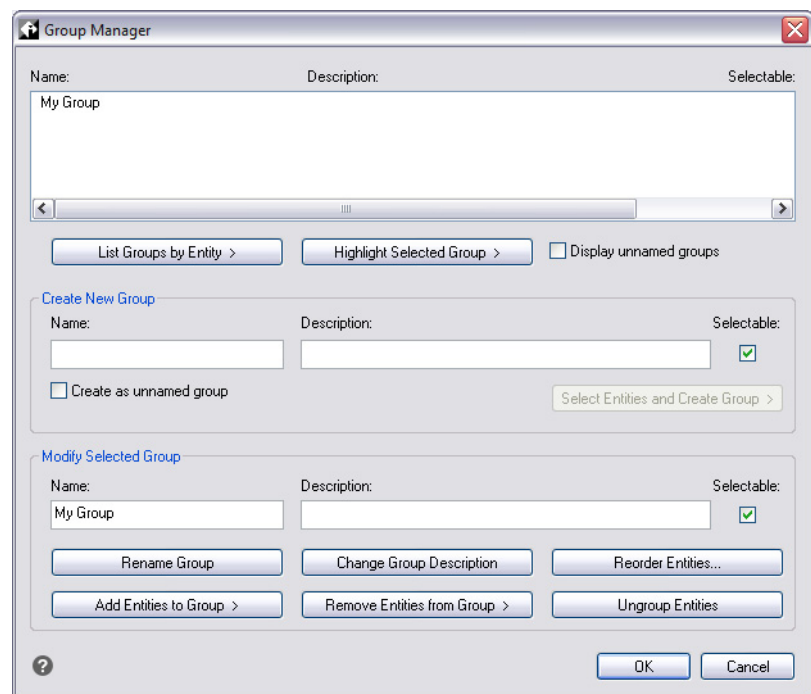
To set how grips work when selecting groups, choose Drawing Settings > Display > Grips and choose an option in Group grips. For more details, see “Using grips” on page 401 in this chapter.

- 4 Click Select Entities and Create Group.
- 5 Select the entities for the group, and then press Enter.
- 6 In the Group Manager dialog box, click OK.

You can select entities using groups.

Enter the name of a group in the command bar when selecting entities.

The Group Manager dialog box controls the settings for all groups in a drawing.




Modifying groups

There are two commands that allow you to modify a group:


- Group Edit command — Adds and removes entities from a group. You can also rename a group using this command.
- Group Manager command — Displays the Group Manager dialog box where you can work with more than one group, add and remove entities from a group, rename a group, determine selectability, change the entity order, and more.

You can also modify a group using the Ungroup command. For more details, see “Ungrouping entities” on page 453 in this chapter.

To modify a group and its entities using the Group Edit command

- 1 Do one of the following to choose Group Edit ():
 - On the ribbon, choose Home > Group Edit (in Groups).
 - On the menu, choose Modify > Groups > Group Edit.
 - On the Groups toolbar, click the Group Edit tool.
 - Type *groupedit* and then press Enter.
- 2 Select the group you want to modify.

To modify a group and its entities using the Group Manager command


- 1 Do one of the following to choose Group Manager ():
 - On the ribbon, choose Home > Group Manager (in Groups).
 - On the menu, choose Modify > Groups > Group Manager.
 - On the Groups toolbar, click the Group Manager tool.
 - Type *group* and then press Enter.
- 2 Select the group you want to modify.
- 3 Under Modify Selected Group, do one or more of the following:
 - Enter a new name, and then click Rename Group.
 - Enter a new description, and then click Change Group Description.
 - For Selectable, mark whether you want the group to be selected in the drawing when you select one entity of the group.

Groups can also have grips.

To set how grips work when selecting groups, choose Drawing Settings > Display > Grips and choose an option in Group grips. For more details, see “Using grips” on page 401 in this chapter.

- Click Add Entities to Group, select the entities to add to the group, and then press Enter.
 - Click Remove Entities from Group, select the entities to remove from the group, and then press Enter.
- 4 In the Group Manager dialog box, click OK.

To change the order of entities in a group


- 1 Do one of the following to choose Group (- On the ribbon, choose Home > Group (in Utilities).
- On the menu, choose Tools > Group.
- On the Tools toolbar, click the Group tool.
- Type *group* and then press Enter.
- 2 Under Modify Selected Group, click Reorder Entities.
- 3 In the Reorder Grouped Entities dialog box, select the group you want to reorder.
- 4 To see the order of entities in the group, click Highlight. Follow the prompts that display to view the entities one by one.
- 5 To reverse the order of all entities in the group, click Reverse Order.
- 6 To change the order of specific entities or a range of entities:
 - In Remove from Position, enter the current position of the entity.
 - In Place to Position, enter the new position of the entity.
 - In Number of Entities, enter the number of entities or range of entities to reorder. For example, if you are changing the order of only one entity, enter 1.
 - Click Reorder.

The entities in a group are numbered 0, 1, 2, 3, and so on.
- 7 Click OK, and then click OK again.

Ungrouping entities

When you ungroup entities, the group breaks apart into separate entities. The entities remain in the drawing but the group is removed.

To ungroup entities

- 1 Do one of the following to choose Ungroup (- On the ribbon, choose Home > Ungroup (in Groups).
- On the menu, choose Modify > Groups > Ungroup.
- On the Groups toolbar, click the Ungroup tool.
- Type *ungroup* and then press Enter.
- 2 Select the group that you want to break into separate source entities.
- 3 If the selected entity belongs to more than one group, do one of the following at the prompt that displays:
 - Choose Accept to break up the group at the highest level.
 - Choose Next to cycle through nested groups. When the group that you want to break up is highlighted, press Enter.

Editing polylines


You can modify any type of two-dimensional or three-dimensional polyline. Entities such as rectangles, polygons, and donuts, as well as three-dimensional entities such as pyramids, cylinders, and spheres, are all variations of polylines that you can edit.

You can edit a polyline by opening or closing it, by changing its overall width or the widths of individual segments, and by converting a polyline with straight line segments into a flowing curve or an approximation of a spline. In addition, you can use the Edit Polyline tool to edit individual vertices, adding, removing, or moving vertices. You can also add new segments to an existing polyline, change the linetypes of a polyline, and reverse the direction or order of the vertices.

Converting an entity to a polyline

To modify a polyline, you first select the polyline, and then select a polyline editing option. The available options vary depending on whether the selected polyline is a two-dimensional or three-dimensional entity. If the selected entity is not a polyline, the Edit Polyline tool provides the option of turning it into one. You can convert only arcs and lines into polylines. If several arcs or lines are joined endpoint to endpoint, they can all be selected and turned into one polyline.

To convert an entity into a polyline

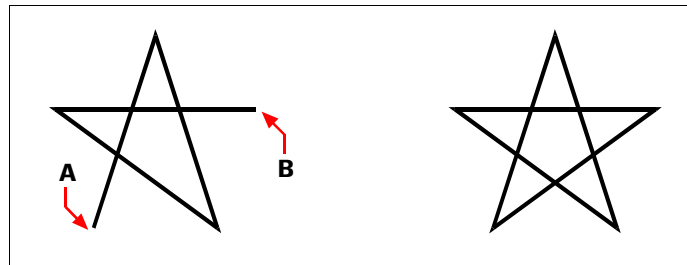
- 1 Do one of the following to choose Edit Polyline ():
 - On the ribbon, choose Edit > Edit Polyline (in Modify).
 - On the menu, choose Modify > Edit Polyline.
 - On the Modify toolbar, click the Edit Polyline tool.
 - Type *editpline* and then press Enter.
- 2 Select the entity.
- 3 In the prompt box, choose Yes-Turn Into Polyline.
- 4 In the prompt box, choose another option, or choose Done to complete the command.

Opening and closing polylines

When you close a polyline, the program draws a straight polyline segment from the last vertex of the polyline to the first vertex. Opening a polyline removes the closing segment. When you select a polyline for editing, the prompt box displays either the Open or Close option, depending on whether the polyline you select is closed or open.

To close an open polyline

- 1 Do one of the following to choose Edit Polyline (🔧):
 - On the ribbon, choose Edit > Edit Polyline (in Modify).
 - On the menu, choose Modify > Edit Polyline.
 - On the Modify toolbar, click the Edit Polyline tool.
 - Type *editpline* and then press Enter.
- 2 Select the polyline.
- 3 In the prompt box, choose Close.
- 4 In the prompt box, choose another option, or choose Done to complete the command.



Closing an open polyline adds a straight polyline segment between the first (A) and last (B) vertices.

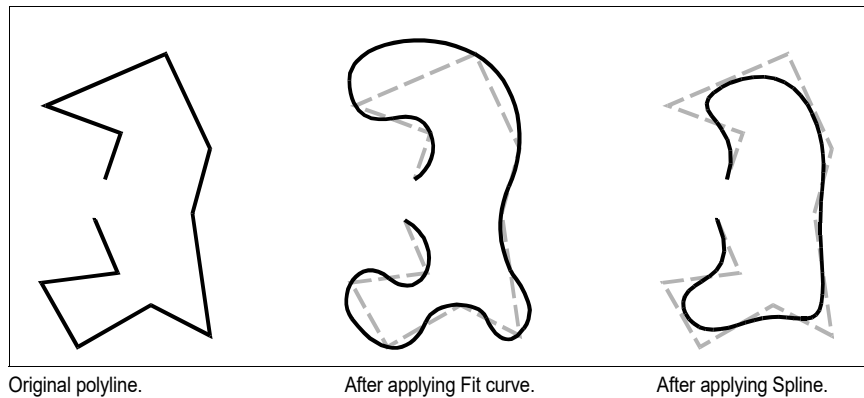
Result.

Curving and decurving polylines

You can convert a multisegment polyline into a smooth curve using either the Fit or Spline option. The Fit option creates a smooth curve connecting all the vertices. The Spline option computes a smooth curve that is pulled toward the vertices but passes through only the first and last vertices. The Decurve option removes Fit or Spline curves and arcs, leaving straight segments between the vertices.

To fit a curve to a polyline

- 1 Do one of the following to choose Edit Polyline (🔧):
 - On the ribbon, choose Edit > Edit Polyline (in Modify).
 - On the menu, choose Modify > Edit Polyline.
 - On the Modify toolbar, click the Edit Polyline tool.
 - Type *editpline* and then press Enter.
- 2 Select the polyline.
- 3 In the prompt box, choose Fit.
- 4 In the prompt box, choose another option, or choose Done to complete the command.



Joining polylines

You can add an arc, line, or polyline entity to an existing open polyline, forming one continuous polyline entity. To join an entity to a polyline, that entity must already share an endpoint with an end vertex of the selected polyline.


When you join an entity to a polyline, the width of the new polyline segment depends on the width of the original polyline and the type of entity you are joining to it:

- A line or an arc assumes the same width as the polyline segment for the end vertex to which it is joined.
- A polyline joined to a tapered polyline retains its own width values.
- A polyline joined to a uniform-width polyline assumes the width of the polyline to which it is joined.

Use the Join command to join arcs and lines.

For more details, see “Joining entities” on page 446 in this chapter.


To join an arc, line, or polyline to an existing polyline

- 1 Do one of the following to choose Edit Polyline ():
 - On the ribbon, choose Edit > Edit Polyline (in Modify).
 - On the menu, choose Modify > Edit Polyline.
 - On the Modify toolbar, click the Edit Polyline tool.
 - Type *editpline* and then press Enter.
- 2 Select the polyline.
- 3 In the prompt box, choose Join.
- 4 Select the arc, line, or polyline to join.
- 5 In the prompt box, choose another option, or choose Done to complete the command.

Changing the polyline width

You can change the width of an entire polyline, applying a uniform width to the entire entity or tapering the polyline uniformly along its entire length.

To apply a uniform width to an entire polyline

- 1 Do one of the following to choose Edit Polyline (

To taper a polyline uniformly along its length

- 1 Do one of the following to choose Edit Polyline (

Editing polyline vertices

You can use the Edit Vertices option to modify individual polyline vertices. When you select this option, the program switches into a special vertex editing mode and places an *x* on the first vertex. The *x* indicates the vertex you are editing. The Next and Previous options move the *x* to the next or previous vertex. You can edit only one vertex at a time.


When editing vertices, you can modify the polyline in the following ways:

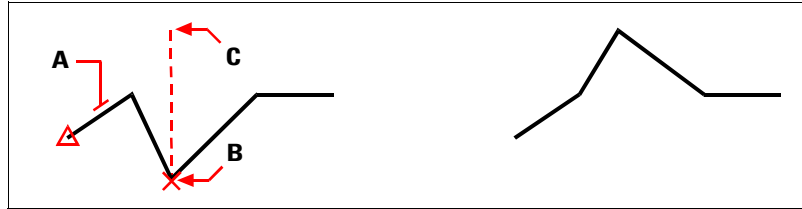
- Convert a polyline segment into a curve by specifying a new tangent angle.
- Break a polyline into two separate polylines.
- Insert a new vertex after the current vertex.
- Move the current vertex.
- Straighten the polyline segment between two vertices.
- Change the width of the polyline segment between two vertices.

You can also reverse polyline vertices.

To reverse the direction of polyline segments, use the Reverse command. For more details, see “Reversing the direction of entities” on page 435 in this chapter.

To move a polyline vertex


- 1 Do one of the following to choose Edit Polyline (- On the ribbon, choose Edit > Edit Polyline (in Modify).
- On the menu, choose Modify > Edit Polyline.
- On the Modify toolbar, click the Edit Polyline tool.
- Type *editpline* and then press Enter.
- 2 Select the polyline.
- 3 In the prompt box, choose Edit Vertices.
- 4 In the prompt box, choose Next Vertex.
Repeat until the *x* reaches the vertex you want to move.
- 5 In the prompt box, choose Move.
- 6 Specify the new location for the vertex.
- 7 In the prompt box, choose another option, or choose Exit to stop editing vertices.
- 8 In the prompt box, choose another option, or choose Done to complete the command.

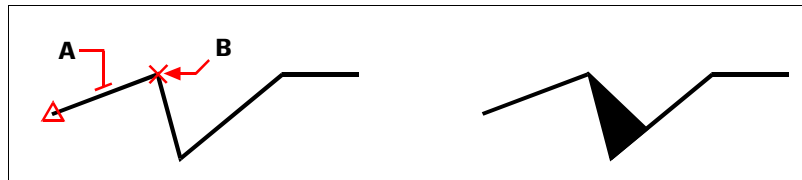


Select the polyline (A), move the current vertex marker to the vertex you want to move (B), and then specify the new vertex location (C).

Result.

To taper the width of an individual polyline segment

- 1 Do one of the following to choose Edit Polyline ():
 - On the ribbon, choose Edit > Edit Polyline (in Modify).
 - On the menu, choose Modify > Edit Polyline.
 - On the Modify toolbar, click the Edit Polyline tool.
 - Type *editpline* and then press Enter.
- 2 Select the polyline.
- 3 In the prompt box, choose Edit Vertices.
- 4 In the prompt box, choose Next Vertex.
Repeat until the *x* reaches the first vertex of the segment you want to taper.
- 5 In the prompt box, choose Width.
- 6 Specify the starting width.
- 7 Specify the ending width.
- 8 In the prompt box, choose another option, or choose Exit to stop editing vertices and update the display.
- 9 In the prompt box, choose another option, or choose Done to complete the command.



Select the polyline (A), move the current vertex marker to the first vertex of the segment you want to taper (B), and then specify the new starting and ending widths for that segment.

Result.


Chamfering and filleting entities

You can chamfer or fillet entities. A chamfer connects two nonparallel entities with a line to create a beveled edge. A fillet connects two entities with an arc of a specified radius to create a rounded edge. If both entities you are working with are on the same layer, the chamfer or fillet is drawn on that layer. If they are on different layers, the chamfer or fillet is drawn on the current layer.

Modifying the chamfer and fillet settings

The Chamfer/Fillet settings in the Drawing Settings dialog box control the chamfer and fillet settings. The portions of the entities that extend beyond the chamfer or fillet are normally deleted when you create the chamfer or fillet. You can retain these original entities, however, by changing the settings in the dialog box.

To modify the chamfer and fillet settings

- 1 Do one of the following to choose Drawing Settings ():
 - On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
 - On the menu, choose Tools > Drawing Settings.
 - On the Tools toolbar, click the Drawing Settings tool.
 - Type *dsettings* and then press Enter.
- 2 In the Drawing Settings dialog box, click the Entity Modification tab.
- 3 Click the Chamfer/Fillet tab and choose from the following:
 - Corners — Select to remove or retain portions of entities that extend beyond the chamfer or fillet.
 - Fillet radius — Specify the fillet radius or click Select to specify it by selecting two points in the drawing.
 - Chamfer Distances and Angles — Choose to use the distance-distance method or distance-angle method when creating chamfers. The distance-distance method creates a chamfer using two chamfer distances that you specify. The distance-angle method creates a chamfer using a chamfer length and angle that you specify.
- 4 Click OK.


Chamfering entities

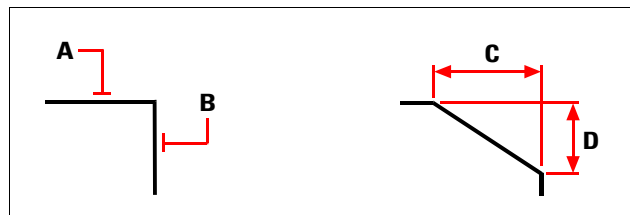
You can connect two nonparallel entities by extending or trimming them and then joining them with a line to create a beveled edge. You can chamfer lines, polylines, rays, and infinite lines. When creating a chamfer, you can specify how far to trim the entities back from their intersection (distance-distance method), or you can specify the length of the chamfer and the angle it forms along the first entity (distance-angle method).

When chamfering a polyline, you can chamfer multiple segments between two selected polyline segments, or you can chamfer the entire polyline.

Chamfering two entities using the distance-distance method

To chamfer two entities using the distance-distance method


- 1 Do one of the following to choose Chamfer ():
 - On the ribbon, choose Modify > Chamfer (in Modify).
 - On the menu, choose Modify > Chamfer.
 - On the Modify toolbar, click the Chamfer tool.
 - Type *chamfer* and then press Enter.
- 2 In the prompt box, choose Chamfer Settings.
- 3 In the Drawing Settings dialog box, click the Entity Modification tab.
- 4 Under Chamfer Distances And Angles, click Distance-Distance.
- 5 Under Chamfer Distances And Angles, specify the first and second chamfer distances.
- 6 Click OK.
- 7 Select the first entity.
- 8 Select the second entity.

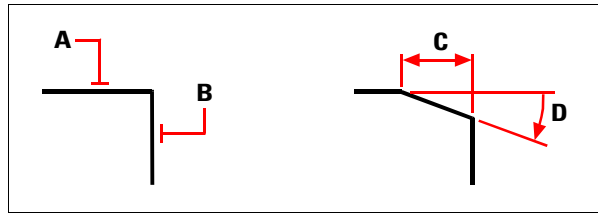


Select the first (A) and second (B) entities. The chamfer is drawn, based on the first (C) and second (D) chamfer distances.

Chamfering two entities using the distance-angle method

To chamfer two entities using the distance-angle method


- 1 Do one of the following to choose Chamfer ():
 - On the ribbon, choose Modify > Chamfer (in Modify).
 - On the menu, choose Modify > Chamfer.
 - On the Modify toolbar, click the Chamfer tool.
 - Type *chamfer* and then press Enter.
- 2 In the prompt box, choose Chamfer Settings.
- 3 In the Drawing Settings dialog box, click the Entity Modification tab.
- 4 Under Chamfer Distances And Angles, click Distance-Angle.
- 5 Under Chamfer Distances And Angles, specify the chamfer distance and angle.
- 6 Click OK.
- 7 Select the first entity.
- 8 Select the second entity.

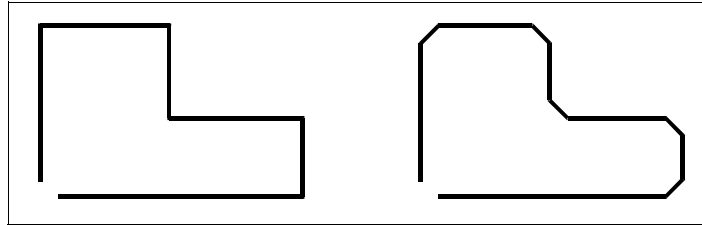


Select the first (A) and second (B) entities. The chamfer is drawn, based on the distance measured along the first entity (C) and the angle (D) formed with the first entity.

Chamfering all vertices in a polyline

To chamfer all vertices in a polyline

- 1 Do one of the following to choose Chamfer ():
 - On the ribbon, choose Modify > Chamfer (in Modify).
 - On the menu, choose Modify > Chamfer.
 - On the Modify toolbar, click the Chamfer tool.
 - Type *chamfer* and then press Enter.
- 2 In the prompt box, choose Polyline.
- 3 Select the polyline.




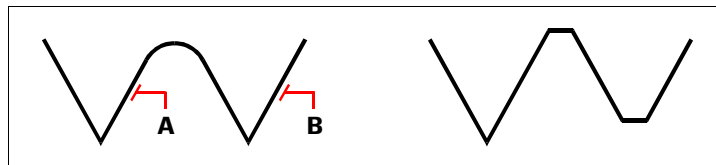
Original polyline.

Result after chamfering.

Chamfering selected vertices in a polyline

To chamfer selected vertices in a polyline

- 1 Do one of the following to choose Chamfer ():
 - On the ribbon, choose Modify > Chamfer (in Modify).
 - On the menu, choose Modify > Chamfer.
 - On the Modify toolbar, click the Chamfer tool.
 - Type *chamfer* and then press Enter.
- 2 Select the polyline along the segment where you want to begin the chamfer.
- 3 Select the polyline along the segment where you want to end the chamfer.



Select the polyline along the (A) and (B) segments.


Result after chamfering.

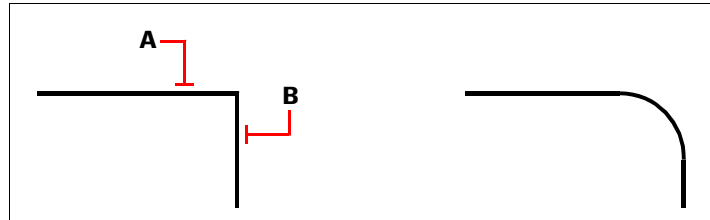
Filleting entities

You can connect two entities with an arc of a specified radius to create a rounded edge. You can fillet pairs of line segments, straight polyline segments, arcs, circles, rays, and infinite lines. You can also fillet parallel lines, rays, and infinite lines. When filleting a polyline, you can fillet multiple segments between two selected segments, or you can fillet the entire polyline.

Filleting two entities

To fillet two entities

- 1 Do one of the following to choose Fillet ():
 - On the ribbon, choose Modify > Fillet (in Modify).
 - On the menu, choose Modify > Fillet.
 - On the Modify toolbar, click the Fillet tool.
 - Type *fillet* and then press Enter.
- 2 In the prompt box, choose Fillet Settings.
- 3 In the Drawing Settings dialog box, specify the fillet radius.
- 4 Click OK.
- 5 Select the first entity.
- 6 Select the second entity.




Select the first (A) and second (B) entities.

Result after filleting.

Filleting an entire polyline

To fillet an entire polyline

- 1 Do one of the following to choose Fillet ():
 - On the ribbon, choose Modify > Fillet (in Modify).
 - On the menu, choose Modify > Fillet.
 - On the Modify toolbar, click the Fillet tool.
 - Type *fillet* and then press Enter.
- 2 In the prompt box, choose Polyline.
- 3 Select the polyline.




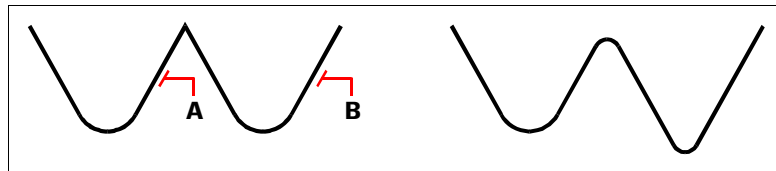
Select the polyline (A).

Result after filleting.

Filleting selected vertices in a polyline

To fillet selected vertices in a polyline

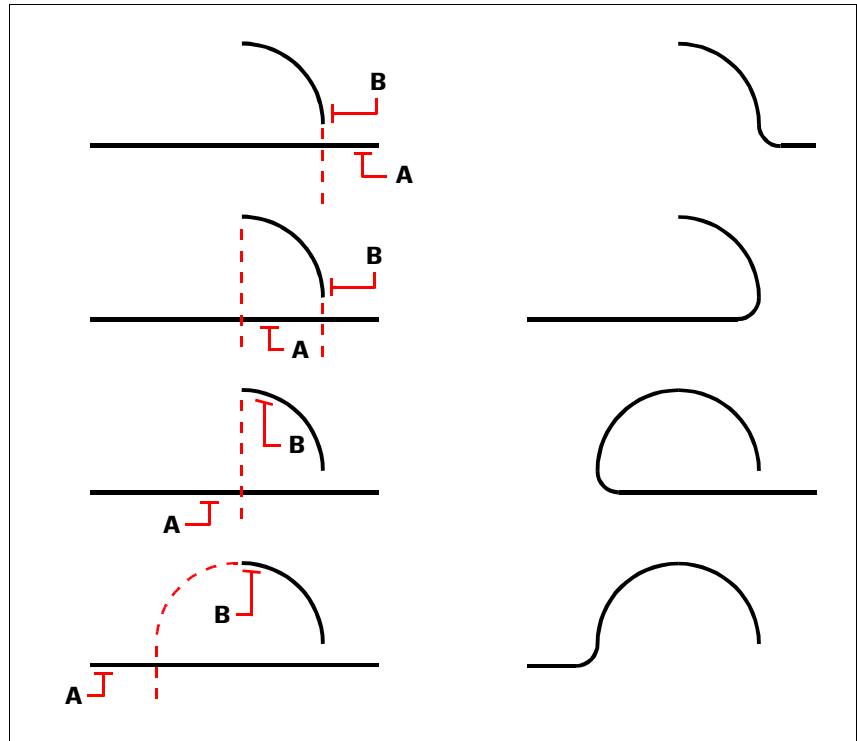
- 1 Do one of the following to choose Fillet ():
 - On the ribbon, choose Modify > Fillet (in Modify).
 - On the menu, choose Modify > Fillet.
 - On the Modify toolbar, click the Fillet tool.
 - Type *fillet* and then press Enter.
- 2 Select the polyline along the segment where you want to begin the fillet.
- 3 Select the polyline along the segment where you want to end the fillet.



Select the polyline along the (A) and (B) segments.

Result after filleting.

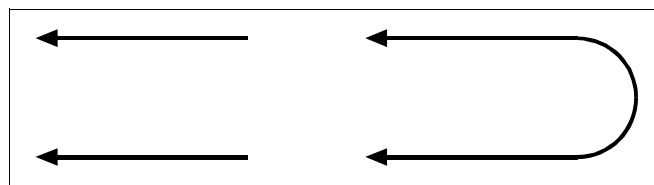
When you fillet circles and arcs, more than one fillet can exist between the entities. The point at which you select the entities determines the fillet.



Select entities (A and B).

Results after filleting.

You can fillet parallel lines, rays, and infinite lines. The first entity must be a line or ray; the second entity can be a line, ray, or infinite line. The diameter of the fillet arc is always equal to the distance between the parallel entities. The current fillet radius is ignored.



Original entities.

Result after filleting.

Working with text

You can insert text into your drawing and control its appearance, allowing you to provide additional information for your PTCAD drawings. This section explains how to:

- Create line text, including along an arc.
- Create paragraphs.
- Create text styles.
- Format text.
- Change line text and paragraph text.
- Convert line text to paragraph text.
- Working with fields.
- Check the spelling of text.
- Use an alternate text editor.
- Working with text written in different languages.
- Working with missing fonts.

Topics in this chapter

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Creating line text

Most commonly, lines of text are created using the Text command which creates single-line text entities. A line of text can also be created along an arc using the Arc-Aligned Text command.

When creating and editing single-line text, the text is checked for spelling automatically and incorrectly spelled words display with a dashed red underline.

Creating a line of text

When you create single-line text, you end each line of text by pressing Enter. Each line of text is created as a separate entity that you can modify.

To create a line of text


- 1 Do one of the following to choose Text (**A**):
 - On the ribbon, choose Home > Text (in Annotation) or choose Annotate > Text (in Text).
 - On the menu, choose Draw > Text.
 - On the Draw toolbar, click the Text tool.
 - Type *text* and then press Enter.
- 2 Specify the insertion point for the first character.
- 3 Specify the height of the text.
- 4 Specify the text rotation angle.
- 5 Type the text, and then press Enter at the end of each new line.
- 6 To complete the command, press Enter again.

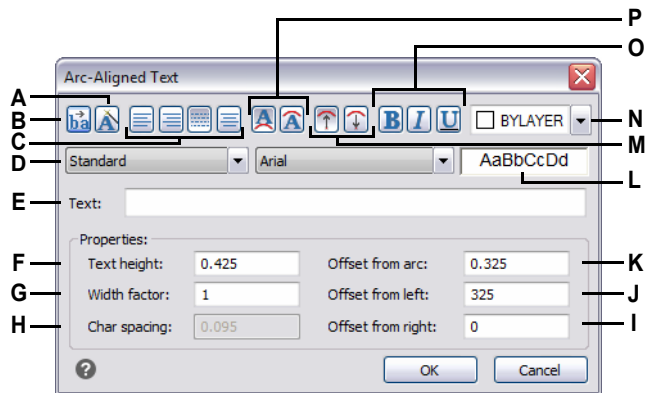
If you've already created text, new text can appear immediately below the previous text. Choose Insert > Text. When prompted for an insertion point, press Enter. The new text will keep the same height and rotation angle as the previous text.

Creating a single line of text along an arc

You can create a single line of text positioned along an arc. Arc-aligned text is its own special type of text entity; it is not exactly the same as a single-line text entity.

To create a line of text along an arc

- 1 Do one of the following to choose Arc-Aligned Text ():
 - On the ribbon, choose Express Tools > Arc-Aligned Text (in Text).
 - On the menu, choose Express Tools > Text > Arc-Aligned Text.
 - Type *arctext* and then press Enter.
- 2 Select the arc along which you want to align text.
- 3 In Text, type the text.
- 4 Select arc-aligned text options.
- 5 Click OK.



- | | |
|---|---|
| <p>A Click to maintain the arrangement of text relative to the arc while moving the arc.</p> <p>B Click to reverse the text flow direction.</p> <p>C Click to align text to the left, right, fit (text fills the length of the arc), or center.</p> <p>D Select a text style.</p> <p>E Type the text to display along the arc.</p> <p>F Type the text font height.</p> <p>G Type the font width factor.</p> <p>H Type the spacing between characters.</p> <p>I Enter the spacing between text and the right edge of the arc. This option is not available when alignment is set to the right and to the center of the arc.</p> | <p>J Enter the spacing between text and the left edge of the arc. This option is not available when alignment is set to the left and to the center of the arc.</p> <p>K Enter the spacing between arc and text.</p> <p>L Displays a sample of the text.</p> <p>M Click to orient text characters either outward from or inward to the center of the arc.</p> <p>N Click to set the text color to BYLAYER, BYBLOCK, one of the standard colors, or choose Select Color for more options.</p> <p>O Click to apply bold, italics, and underline to the text.</p> <p>P Click to place text either on the convex side of the arc or on the concave side of the arc.</p> |
|---|---|

Creating paragraph text

Paragraph text consists of one or more lines of text or paragraphs that fit within a boundary width that you specify. Each paragraph text entity you create is treated as a single entity regardless of the number of individual paragraphs or lines of text it contains.

When you create paragraph text, you first determine the paragraph's boundary width by specifying the opposite corners of a rectangle. The paragraph text automatically wraps so that it fits within this rectangle. The first corner of the rectangle determines the default attachment point of the paragraph text. You can determine the direction in which text flows within the rectangle, and you can also select the text and paragraph format, text style, text height, and the rotation angle of the entire paragraph text entity.

When creating and editing paragraph text, the text can be checked for spelling automatically and incorrectly spelled words display with a dashed red underline.

To create paragraph text

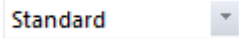







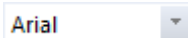
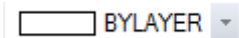

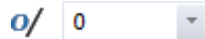
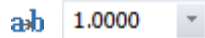
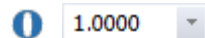





- 1 Do one of the following to choose Multiline Text (**A**):
 - On the ribbon, choose Home > Multiline Text (in Annotation) or Annotate > Multiline Text (in Text).
 - On the menu, choose Draw > Multiline Text.
 - On the Draw toolbar, click the Multiline Text tool.
 - Type *mtext* and then press Enter.
- 2 Select the first corner of the text area.
- 3 In the command bar or context menu, optionally choose the properties you want to set, or proceed directly to the next step. You can also choose these properties later for the resulting multiline text entity.
- 4 Select the second corner of the text area.
- 5 Type the text you want.














To create paragraphs, press Enter and continue typing.
- 6 On the ribbon (or floating toolbar if not viewing the ribbon), use the Text Editor tools to make your selections for highlighted text, or if no text is highlighted, the changes affect new text when you type it.

Text Editor tools display on the ribbon or a floating toolbar.

By default, the floating toolbar displays only if the ribbon is turned off, however, some users prefer to always use the floating toolbar. To enable the floating toolbar even when the ribbon displays, type cui, and on the Customize tab, select AcTextEditorInProgress in Contextual Ribbon States. On the right, choose True for the Disabled option, then click OK.

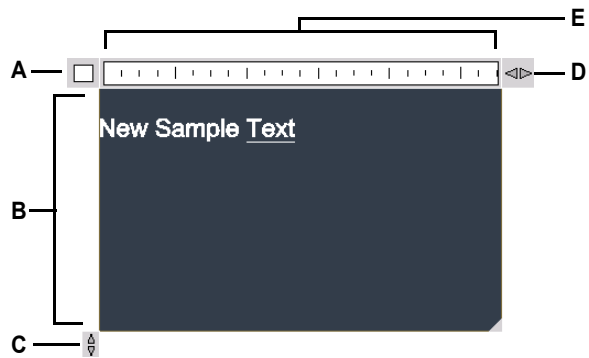
7 Click Close Editor to accept your changes and close the editor.

Tool	Name	Description
	Text Style	Select a text style.
	Text Height	Select or type the text font height.
	Annotative	Select or type the text font height.
	Bold and Italics	Click to bold and/or italicize text.
	Strikethrough	Click to strikethrough text.
	Underline and Overline	Click to underline and/or overline text.
	Uppercase and Lowercase	Click to make text uppercase or lowercase.
	Superscript and Subscript	Click to make text superscript or subscript.
	Text Font	Select the text font.
	Text Color	Select BYBLOCK, BYLAYER, the color of your choice, or choose Select Color to select from additional colors.
	Background Mask	Click to set the background settings for text. For more details, click Help in the dialog box that displays.
	Oblique Angle	Select or type the angle of text characters.
	Tracking	Select or type the factor representing spacing between characters.
	Width Factor	Select or type the font width factor.
	Clear Formatting	Choose whether to remove character formatting from selected characters, paragraph formatting or all formatting from a selected paragraph.
	Combine Paragraphs	Click to combine selected paragraphs into a single paragraph.
	Justification	Select the text box justification.
	List	Select the type of list for the text. You can also specify to continue or restart numbered lists.
	Line Spacing	Select the spacing between lines.

Tool	Name	Description
	Paragraph	Click to specify tab, indent, alignment, and line spacing settings. For more details, click Help in the dialog box that displays.
	Default, Left, and Center	Click to set the paragraph justification to default, left, or center.
	Right, Justify, and Distribute	Click to set the paragraph justification to right, justify, or distribute.
	Undo and Redo	Click to redo or undo the previous action.
	Symbol	Click to insert a symbol character where the cursor is located in the text editor.
	Insert Field	Click to insert a field where the cursor is located in the text editor. For more details, click Help in the dialog box that displays.
	Columns	Click to specify columns for the text, and options for the columns. For more details, click Help in the program.
	Find and Replace	Click to find and replace text in the text editor.
Import Text	Import Text	Click to select a file that contains text to import.
AutoCAPS	AutoCAPS	Click to turn on or off capitalizing text automatically as you type.
	Check Spelling	Click to check text for spelling automatically. Incorrectly spelled words display with a dashed red underline; right-click to choose a correctly spelled word.
	Ruler	Click to show or hide the ruler in the text editor.
	Stacked Text	Click to stack or unstack selected text, for example, fractions can display as stacked text.
	More	Click to display a menu of additional options, including the character set, settings for the text editor, and online help.
	Done	Accepts your changes and closes the editor.

Access more options by right-clicking the text editor.

Right-click the text editor to access additional options, including *Select All*, *Cut*, *Copy*, *Paste*, and *Paste Special*. You can also use typical shortcuts such as *Ctrl + V* to paste text from the clipboard.



- A** Click to accept your changes and close the editor.
- B** Type and select text.
- C** Click and drag to resize the editor vertically.
- D** Click and drag to resize the editor horizontally.
- E** Displays the ruler.

There are two multiline text editors.

To switch to a dialog box version of the multiline text editor, set the `MTEXTED` system variable to “`olddeditor`”.

Working with text styles

When you add text to a drawing, it uses the current text style. Text style determines the font, size, angle, orientation, whether the text is annotative by default, and other characteristics.

Every drawing has a default text style, named Standard, which initially uses the Arial font. You cannot delete the Standard style, but you can rename it or modify it. You can change the font, the size of the font, and the obliquing angle applied to it. If you change the font or orientation of an existing style, all existing text entities created using that style are automatically updated to reflect the new font or orientation. Changing any other characteristic has no effect on existing text. You also can create and use an unlimited number of additional text styles.

The text style determines the characteristics shown in the following table.

Style characteristic	Default	Description
Style name	Standard	The name of the style, up to 255 characters.
Font file	Arial	The font file on which the style is base, and whether Asian language big font files are used (for .SHX file fonts only).
Language	Western	The character height. A value of 0 prompts for text height upon insertion.
Annotative	No	Determines whether text is annotative by default and therefore its display and print is affected by annotation scaling. If annotative by default, also determines whether the text, when displayed in paper space, is oriented automatically according to the layout viewport.
Text height	0	The character height. A value of 0 prompts for text height upon insertion. The text height is the paper text height if the text style is annotative.
Width factor	1	The horizontal expansion or compression of the text. Values less than 1 compress the text; values greater than 1 expand the text.
Obliquing angle	0	The slant of the text, in degrees. Negative values slant the text to the left; positive values slant the text to the right.
Backward	No	Determines whether text appears backward.
Upside down	No	Determines whether text appears upside down.
Vertical	No	Determines whether text has a vertical orientation.

To create a text style

- Do one of the following to choose Text Styles Manager (

- 3 Type a new text style name, then click OK.
- 4 Under Text Font, select the name, style, and language of the font you want to use.
- 5 Under Text Measurements, select the check boxes if you want annotative text; you can also enter a Fixed Text Height (Paper Text Height if annotative), Width Factor, or Oblique Angle measurement.
- 6 Under Text Generation, select the check boxes you want to indicate the direction for printed text to appear.
- 7 Click OK.
- 8 To begin using the new style, choose Draw > Text.
- 9 In the prompt box, select Use Defined Style.

Formatting text

When you create text, you choose the text style and set the alignment. The style determines the font characteristics for the text. For single-line text, the alignment point determines how the text aligns with the text insertion point. For paragraph text, the alignment point determines the location of the attachment point in relation to the paragraph text boundary and the direction in which text flows within the boundary.

You can also use the Express Tools > Text area for commands that scale text, justify text, enclose text in shapes, and more. All of these options are described in the online help.

Setting the line text style

You can set the text style before you specify the insertion point. You select the text style by typing the name of a previously defined style.

To specify a line text style

- 1 Do one of the following to choose Text (**A**):
 - On the ribbon, choose Home > Text (in Annotation) or choose Annotate > Text (in Text).
 - On the menu, choose Draw > Text.
 - On the Draw toolbar, click the Text tool.
 - Type *dtext* and then press Enter.
- 2 In the prompt box, choose Use Defined Style.
- 3 Type the name of a previously defined text style, and then press Enter.
To display a list of available text styles, type a question mark (?), and then press Enter.
- 4 Specify the text insertion point.

- 5 Specify the text height.
- 6 Specify the rotation angle.
- 7 Type the text line, and then press Enter.
- 8 To complete the command, press Enter again.

Setting the paragraph text style

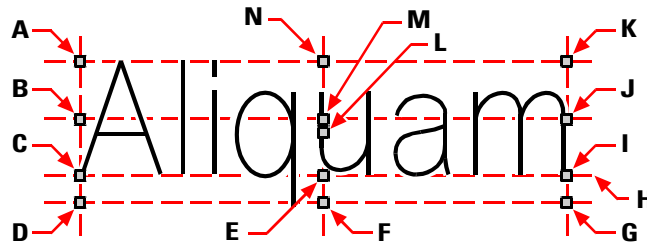
You can set the paragraph text style before you specify the insertion point. You select the text style by typing the name of a previously defined style.

To specify a paragraph text style

- 1 Do one of the following to choose Multiline Text (**A**):
 - On the ribbon, choose Home > Multiline Text (in Annotation) or Annotate > Multiline Text (in Text).
 - On the menu, choose Draw > Multiline Text.
 - On the Draw toolbar, click the Multiline Text tool.
 - Type *mtext* and then press Enter.
- 2 Specify the first point of the text box.
- 3 In the prompt box, choose Text Style.
- 4 Type the name of a previously defined text style, and then press Enter.
- 5 Specify the text height.
- 6 Specify the rotation angle.
- 7 Specify the opposite corner of the text box.
- 8 Type the text.
- 9 To complete the command, click Close Editor.

Setting the line text alignment

When you create text, you can set the text alignment before you specify the insertion point. You set the alignment by choosing it in the prompt box. By default, text is left justified. You can align text at the left, center, or right and at the top, middle, or base-line of the text or at the bottom of descending letters.



A Top left	H Baseline
B Middle left	I Right
C Left	J Middle right
D Bottom left	K Top right
E Center	L Middle
F Bottom center	M Middle center
G Bottom right	N Top center

You can also align text so that it fits or aligns between two points. The Align option creates text that scales up or down while maintaining a constant height/width ratio; the Fit option expands or compresses the text to fit between the two points.

Ut wisi enim ad minim veniam

Text aligned between two points maintains a constant height/width ratio.

Ut wisi enim


Text fit between two points expands or compresses to fit.

To specify the line text alignment before entering text


- Do one of the following to choose Text (**A**):
 - On the ribbon, choose Home > Text (in Annotation) or choose Annotate > Text (in Text).
 - On the menu, choose Draw > Text.
 - On the Draw toolbar, click the Text tool.
 - Type *dtext* and then press Enter.
- In the prompt box, choose an alignment option, or choose Justification Options to display all the justification options.

- 3 Specify the text insertion point, and then continue creating the line text.

To specify the justification for existing line text entities

- 1 Do one of the following to choose Justify Text (- On the ribbon, choose Express Tools > Justify Text (in Text > Modify Text).
- On the menu, choose Express Tools > Text > Justify Text.
- Type *tjust* and then press Enter.
- 2 Select text entities. In addition to single-line text, you can also choose multiline text and attribute text.
- 3 Choose one of the following:
 - **Center** Justification point of the text is center aligned.
 - **Middle** Justification point of the text is middle aligned (left/right and top/bottom).
 - **Right** Justification point of the text is right aligned.
 - **TL** Justification point of the text is top-left aligned.
 - **TC** Justification point of the text is top-center aligned.
 - **TR** Justification point of the text is top-right aligned.
 - **ML** Justification point of the text is middle-left aligned.
 - **MC** Justification point of the text is middle-center aligned.
 - **MR** Justification point of the text is middle-right aligned.
 - **BL** Justification point of the text is bottom-left aligned.
 - **BC** Justification point of the text is bottom-center aligned.
 - **BR** Justification point of the text is bottom-right aligned.


To fit existing line text within two points

- 1 Do one of the following to choose Text Fit (- On the ribbon, choose Express Tools > Fit Text (in Text > Modify Text).
- On the menu, choose Express Tools > Text > Fit Text.
- Type *textfit* and then press Enter.
- 2 Select single-line text entities.
- 3 Specify a new desired ending point or choose Start Point to specify a new starting point and then specify the ending point.

The Text Fit command works with single-line text only.

To fit multiline text within two points, first use the Explode command to convert it to single-line text.

To change the scale of existing line text

- 1 Do one of the following to choose Scale Text (- On the ribbon, choose Express Tools > Scale Text (in Text > Modify Text).
- On the menu, choose Express Tools > Text > Scale Text.
- Type *scaletext* and then press Enter.
- 2 Select single-line text entities.
- 3 Choose a base point option for scaling.
- 4 Choose one of the modes of scaling text and follow the prompts. Refer to online help for more details.

Setting the paragraph text alignment

When you create paragraph text, you can set the text alignment by specifying the direction in which text flows within the boundary. You can set the paragraph text alignment either in the prompt box displayed after you specify the first corner of the paragraph text boundary, from the Properties pane, or using the Justify Text express tool command. You can specify the attachment point at the top left, top center, top right, middle left, middle center, middle right, bottom left, bottom center, or bottom right. The paragraph text can flow left to right, right to left, top to bottom, or bottom to top. You can change the scale of existing paragraph text using the Scale Text express tool command.

Including special text characters

You can use control codes to overscore or underscore text or to include special characters. Both overscore and underscore can be active at the same time. To include control codes, as you type text, type two percent symbols (%%) followed by the special control code or character. A single percent sign is treated as a normal text character. A triple-percent control code is provided for those instances where a control-code sequence must follow a percent sign in the text.

Special text characters

Control code	Function
%%o	Toggles overscore mode on and off.
%%u	Toggles underscore mode on and off.
%%d	Draws a degree symbol (°).
%%p	Draws the plus-or-minus symbol (±).
%%c	Draws the circle diameter symbol (Ø).
%%%	Forces a single percent sign.
%%nnn	Draws special character number <i>nnn</i> .


Ut wisi enim° ad± minim Ø veniam

Text created using special text characters: Ut wisi %%uenim%%d ad%%p%%u minim %%c %%oveniam.

Creating a frame around text

You can draw a frame around each selected line text, paragraph text or attribute text, that is enclose text with shape. Frames can be in the shape of a circle, rectangle or a slot.

To draw a frame around text

- 1 Do one of the following to choose Enclose Text with Shape ():
 - On the ribbon, choose Express Tools > Enclose Text with Shape (in Text > Modify Text).
 - On the menu, choose Express Tools > Text > Enclose Text with Shape.
 - Type *tcircle* and then press Enter.
- 2 Select line text, paragraph text or attribute text.
- 3 Specify which shape you want to draw around text. Refer to online help for more details.


Changing text

You can change line and paragraph text in your drawing.

Changing line text

You can edit and modify the text as you would any other drawing entity. That is, you can delete, move, rotate, and scale text.

To edit text and its properties

- 1 Do one of the following to choose Edit Text (- On the ribbon, choose Home > Edit Text (in Annotation) or Annotate > Edit Text (in Text).
- On the menu, choose Modify > Entities > Text.
- On the Modify toolbar, click the Edit Text tool.
- Type *textedit* and then press Enter.
- 2 Select the text entity.
- 3 In the Text dialog box, edit the text.
- 4 Change the text properties you want.


The Text dialog box displays when TEXTED is set to 3 (the default).

If the TEXTED system variable is set to 1 or 2, line text is edited in the command bar or Properties pane respectively instead of the Text dialog box.

Changing paragraph text

You can modify the text as you would any other drawing entity. That is, you can delete, move, rotate, and scale text.

To edit paragraph text and its properties



- 1 Do one of the following to choose Edit Text (- On the ribbon, choose Home > Edit Text (in Annotation) or Annotate > Edit Text (in Text).
- On the menu, choose Modify > Entities > Text.
- On the Modify toolbar, click the Edit Text tool.
- Type *textedit* and then press Enter.
- 2 Select the text entity.
- 3 In the text editor, edit the text.
- 4 Change the text properties you want.
- 5 Click Close Editor.

Finding and replacing text

You can search and optionally replace text throughout a drawing, within a layout, or within selected entities.

PTCAD searches and displays a list of matching text. Each found text item has an associated entity type to help you identify its location in the drawing. Entity types can include single-line text, multiline text, dimension leaders, hyperlinks, alternate text, paperspace, and modelspace. Note that sometimes dimension text is listed as multiline text because of the way dimensions are created.

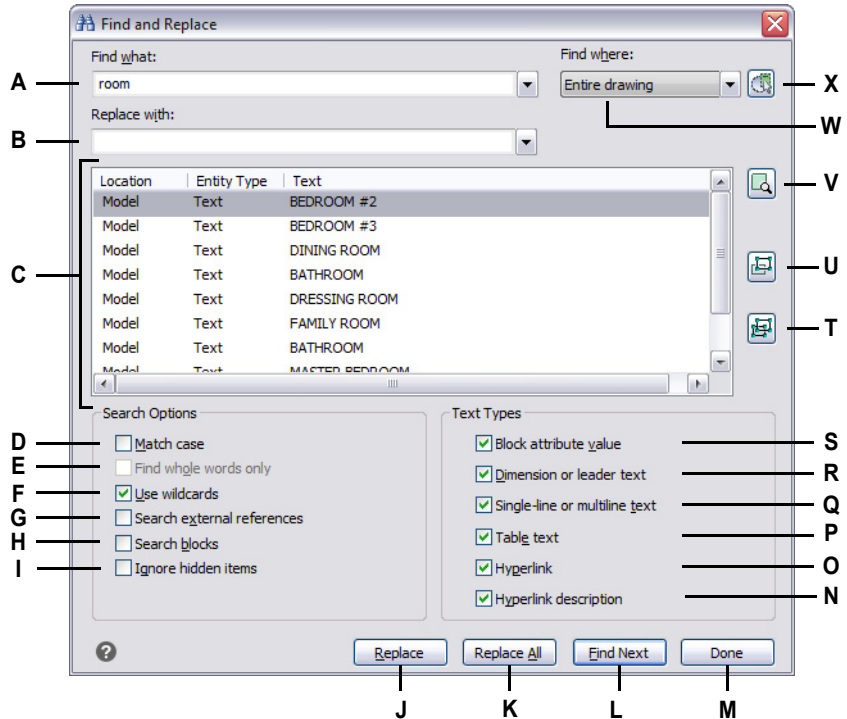
To find and optionally replace text

- 1 Do one of the following to choose Find and Replace ():
 - Choose Edit > Find and Replace.
 - On the Standard toolbar, click the Find and Replace tool.
 - Type *find* and then press Enter.
- 2 In Find What, type the text you want to search for, or select previously entered text from the list.
- 3 In Find Where, select the location where you want to search:
 - **Entire drawing** Searches the entire drawing.
 - **Current layout** Searches the current layout only.
 - **Selected entities** Searches selected entities only. You can click () to switch to the drawing temporarily and select entities.
- 4 In Text Types, select the text types you want to include in the search.
- 5 In Search Options, select the desired search options.
- 6 Click Find to display a list of all matching text.
- 7 To replace text, do the following:
 - In Replace With, enter or select the new text.
 - In the list of found text, select the text to replace.
 - Click Replace.

Use a shortcut for replacing text.

Click Replace All to replace all matching text without finding it first.

- 8 Click Done.




- A Type or select the text to find.
- B Type or select the text to replace found text with (optional).
- C Displays all found text.
- D Select to find only text that matches the upper and lower case specified in Find What.
- E Select to find only whole words that match the text in Find What.
- F Select to allow the use of wildcards in the search.
- G Select to include external references in the search.
- H Select to include blocks in the search.
- I Select to exclude hidden items (such as text on layers that are frozen or turned off and text in block attributes created in invisible mode) from the search.
- J Click to replace selected text in the Location list with text from Replace With.
- K Click to replace all matching text without searching first.
- L Click to find matching text.
- M Click when finished.
- N Select to search hyperlink descriptions.
- O Select to search hyperlinks.
- P Select to search tables.
- Q Select to search both single-line and multiline text.
- R Select to search both dimension and leader text.
- S Select to search block attribute values.
- T Click to zoom to the selected found text in the drawing.
- U Click to create a selection set that includes the selected found text in model space or a single layout.
- V Click to create a selection set that includes all found text in model space or a single layout.
- W Select where to search for text: the entire drawing, the current layout, or selected entities.
- X Click to switch to the drawing and select entities to search.

Changing capitalization of letters

You can quickly change the capitalization of single-line and multiline text entities to these capitalization methods: sentence case, all lower case, all uppercase, title case, and toggle case.


To set capitalization formatting

- 1 Do one of the following to choose Change Text Case ():
 - On the ribbon, choose Express Tools > Change Text Case (in Text > Modify Text).
 - On the menu, choose Express Tools > Text > Change Text Case.
 - Type *tcase* and then press Enter.
- 2 Select text entities.
- 3 Choose one of the following:
 - **Sentence case** The first letter of the first text character is upper case.
 - **lowercase** All text characters are lower case letters.
 - **UPPERCASE** All text characters are upper case letters.
 - **Title** The first letters of all text characters are upper case.
 - **TOGGLE cASE** The first letters of all text characters are lower case and the rest of the letters are upper case.


Converting line text to paragraph text

When converting line text to paragraph text, one or more text entities created with the Text command are combined into one multiline text entity. During the conversion, the selected text entities are removed from the drawing and a multiline text entity is created.

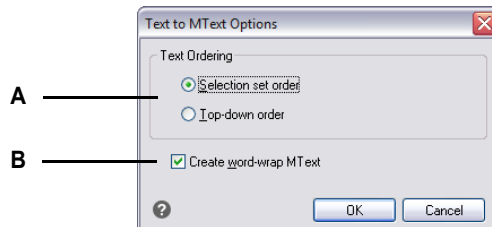
To convert line text to paragraph text

- 1 Do one of the following to choose Text to Multiline Text ():
 - On the ribbon, choose Home > Text to Multiline Text (in Annotation), Annotate > Text to Multiline Text (in Text) or Express Tools > Text to Multiline Text (in Text).
 - On the menu, choose Draw > Text to Multiline Text or choose Express Tools > Text > Text to Multiline Text.
 - On the Text toolbar, click the Text to Multiline Text tool.
 - Type *txt2mtxt* and then press Enter.
- 2 Select one or more text entities.
- 3 When finished with selection, press Enter.

To customize options while converting line text to paragraph text

- 1 Do one of the following to choose Text to Multiline Text ():
 - On the ribbon, choose Home > Text to Multiline Text (in Annotation), Annotate > Text to Multiline Text (in Text) or Express Tools > Text to Multiline Text (in Text).
 - On the menu, choose Draw > Text to Multiline Text or choose Express Tools > Text > Text to Multiline Text.
 - On the Text toolbar, click the Text to Multiline Text tool.
 - Type *txt2mtxt* and then press Enter.
- 2 Press Enter.
- 3 Select one of the following:
 - **Selection set order** Select to add single-line text to multiline text in the order that you select.
 - **Top-down order** Select to add single-line text to multiline text in the order that they appear in the drawing, from the top downwards.
- 4 Mark Create Word-Wrap MText to add spaces within individual lines of text, which helps balance multiple lines of text.
- 5 Click OK.
- 6 Select one or more text entities.
- 7 When finished with selection, press Escape.

All text entities are combined into a single multiline text entity.



A Choose to add text in the order selected or in top-down order.

B Click to add spaces within text lines to balance multiple lines of text.

Working with fields


Fields contain expressions that can be updated automatically to generate new field content as drawing values change. For example, a the *filename* field displays text with the name of drawing file. If the name of the drawing file changes, you can use the Update Field command to automatically update the field with the correct filename.

Inserting a field

Fields can be inserted as multiline text entities directly in the drawing.

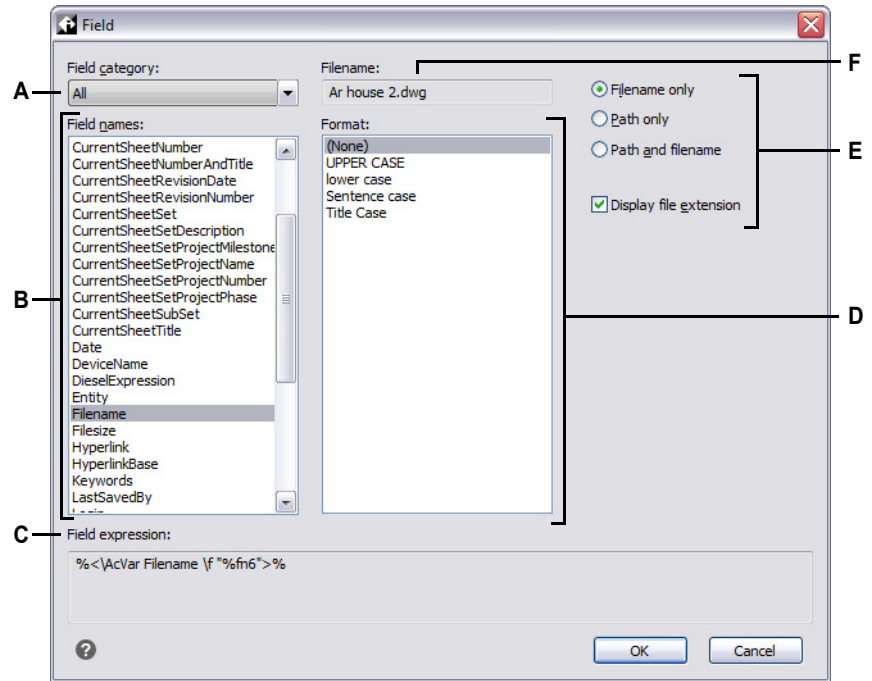
Fields also can be inserted in existing paragraph text and table cells. For more details about editing text, see “Changing paragraph text” on page 481 in this chapter. For more details about editing text in tables, see “Editing table text” on page 182.

To insert a field directly in the drawing

- 1 Do one of the following to choose Field (

Fields display with a gray background color.

To turn off the background color, set the *FIELDDISPLAY* system variable to off (0).




- A** Select a category to filter the Field Names list, or select All to display all available fields.
- B** Select the field you want to insert.
- C** Displays the expression, similar to a formula, that is used to generate content for the selected field.
- D** Select format options for the selected field (varies according to the selected field).
- E** Displays additional options, if applicable for the selected field.
- F** Displays a preview of the field content, if available.

Updating fields

Use the Update Field command to automatically regenerate field content in all entities you select.

To update fields

- Do one of the following to choose Update Field ():
 - On the ribbon, choose Insert > Update Field (in Data).
 - On the menu, choose Insert > Field > Update Field.
 - Type *updatefield* and then press Enter.
- Select entities that contain fields, then press Enter.

Checking the spelling

You can check the spelling of text in your drawing, customize the dictionary to recognize new words, and use a different spelling dictionary that supports another language.

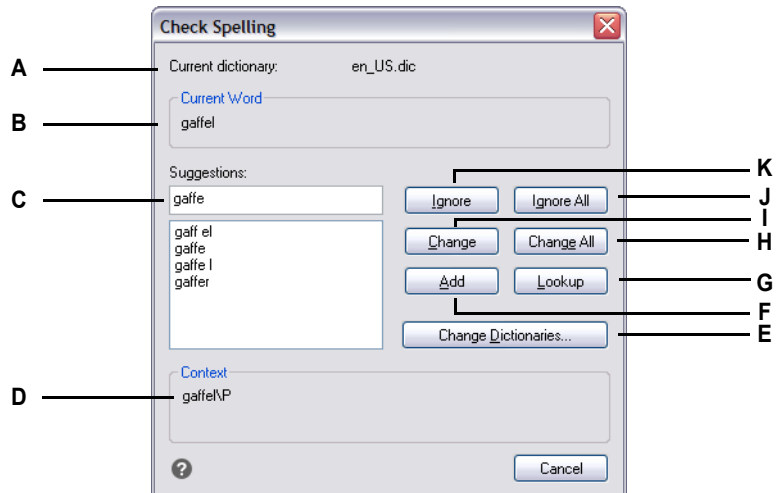
When editing text, the text is checked for spelling automatically and incorrectly spelled words display with a dashed red underline.

Checking the spelling of text

You can use the Check Spelling command to check the spelling of single-line text, multiline text, paragraph text, attributes, attribute definitions, and dimension text.

To check the spelling of text

- 1 Do one of the following to choose Check Spelling (



- A** Displays the current dictionary.
- B** Displays the misspelled word and its context in the drawing.
- C** Displays (or you can type) a new word to replace the misspelled word in the drawing.
- D** Click to select a new word to replace the currently found misspelled word.
- E** Click to select a new dictionary or modify a custom dictionary.
- F** Click to add the suggested word to the custom dictionary.
- G** Click to suggest additional words based on the selected word in the Suggestions list.
- H** Click to change all instances of the misspelled word to the suggested word.
- I** Click to change a single instance of the misspelled word to the suggested word.
- J** Click to skip all instances of the misspelled word.
- K** Click to skip a single instance of the misspelled word without changing it.

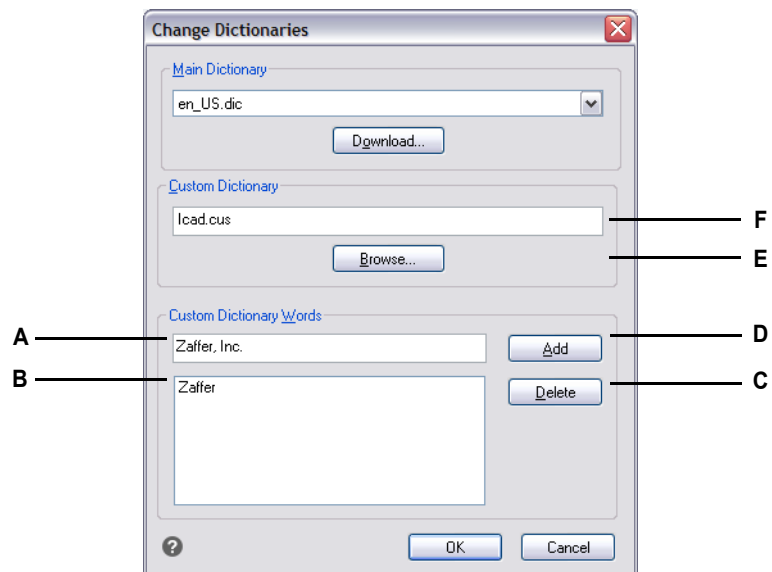
Customizing the spelling words

Most drawings contain text that is not recognized as spelled correctly, even though it is spelled correctly. For example, if your company name Zaffer, Inc. appears in all of your drawings, the company name will appear as misspelled every time you check the spelling of drawing text. You can easily add words to a custom dictionary so that any word in the custom dictionary is recognized as spelled correctly.

The custom dictionary is independent of any spelling dictionary you have chosen to use. You can check the spelling of text using one dictionary, check the spelling of the same or other text using a different dictionary, and in both cases your custom dictionary is used.

To create and manage a custom dictionary

- 1 Do one of the following to choose Check Spelling (ABC):
 - On the ribbon, choose Annotate > Check Spelling (in Text).
 - On the menu, choose Tools > Check Spelling.
 - On the Standard toolbar, click the Check Spelling tool.
 - Type *spell* and then press Enter.
- 2 Select one or more text entities.
- 3 In the Check Spelling dialog box, click Change Dictionaries.
- 4 To add a custom word, type a word in Custom Dictionary Words, then click Add.
- 5 To delete a custom word, select a word in Custom Dictionary Words, then click Delete.
- 6 Click OK.




- | | |
|---|--|
| <p>A Type a word for the custom dictionary.</p> <p>B Displays the words in the custom dictionary.</p> <p>C Click to delete the currently selected word from the custom dictionary.</p> <p>D Click to add the current word in the edit box (A) to the custom dictionary.</p> | <p>E Click to select a different custom dictionary.</p> <p>F Displays the custom dictionary filename. You can type a new path and filename or click Browse to select a file.</p> |
|---|--|

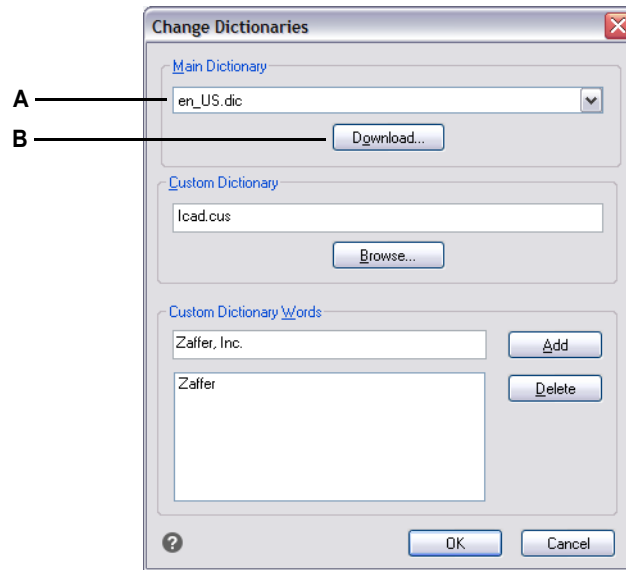
You can also add custom words to the custom dictionary during spell checking of text.
In the Check Spelling dialog box, click Add to add the currently found word to the custom dictionary.

Changing the dictionary

The Check Spelling command compares text found in the drawing with correctly spelled words in an installed dictionary (.dic file). There are many different dictionaries that can be used with PTCAD, including dictionaries in different languages. You can download and install a new dictionary or use a dictionary already installed on your computer.

To download and install a new dictionary

- 1 Do one of the following to choose Check Spelling ():
 - On the ribbon, choose Annotate > Check Spelling (in Text).
 - On the menu, choose Tools > Check Spelling.
 - On the Standard toolbar, click the Check Spelling tool.
 - Type *spell* and then press Enter.
 - On the ribbon, choose Annotate > Check Spelling (in Text).
- 2 Select one or more text entities, then in the Check Spelling dialog box, click Change Dictionaries.
- 3 Click Download.
- 4 From the web page that displays (or from a different web page), download the desired dictionary.
- 5 Unzip the contents of the downloaded file to \MyDocuments\Spelling, or the location where you installed spelling dictionaries for PTCAD.
- 6 Click Change Dictionaries again, which will load the newly installed dictionary.
- 7 In Main Dictionary, select the desired dictionary.
- 8 Click OK.



- A** Select the desired spelling dictionary.
- B** Click to open the default Internet browser to a web page where you can download spelling dictionaries in different languages.

To use a dictionary already installed on your computer

- 1 Do one of the following:
 - Copy the existing dictionary (.dic file) to \MyDocuments\Spelling, or to the location where you install spelling dictionaries for PTCAD.
 - Add the folder of the existing dictionary to the list of folders where PTCAD searches for dictionaries. Choose Tools > Options, click Paths/Files and add the folder location to the Dictionary paths. For more details, see “Changing the options on the Paths/Files tab” on page 829.
- 2 Choose Tools > Check Spelling, select one or more text entities, then in the Check Spelling dialog box, click Change Dictionaries.
- 3 In Main Dictionary, select the desired dictionary.
- 4 Click OK.

Using an alternate text editor

PTCAD includes a built-in text editor for creating paragraph text using the Multiline Text command. You can also specify an alternate text editor for the Multiline Text command.

Selecting an alternate text editor

Before you can use an alternate text editor, you must specify the editor by setting the MTEXTED system variable.

To select an alternate text editor

- 1 Type *mtextd* and then press Enter.
- 2 Enter the path and name of the executable file for the text editor you want to use to create or edit multiline text. For example, to use Microsoft® Wordpad, you would type something similar to the following (adjusting the path name as necessary):
C:\Program Files\Windows\Accessories\Wordpad.exe

Creating paragraph text in an alternate text editor

After you set up PTCAD to use an alternate text editor, you can start using it to include text in your drawings.

To use an alternate text editor

- 1 Do one of the following to choose Multiline Text (**A**):
 - On the ribbon, choose Home > Multiline Text (in Annotation) or Annotate > Multiline Text (in Text).
 - On the menu, choose Draw > Multiline Text.
 - On the Draw toolbar, click the Multiline Text tool.
 - Type *mtext* and then press Enter.
- 2 Select the first and second corners of the text area.
- 3 In the text editor, type the text you want, using the special characters from the table shown next to achieve special formatting. Enter \P to end a paragraph and start a new paragraph on the next line. For example, to use an alignment value of 1 and stack two numbers to display them as a fraction:


```
{\A1;\S1/2;}\P
```
- 4 When your text is complete, save the changes and exit the text editor.

Special format character	Function
\O...o	Toggles overscore mode on and off.
\L...l	Toggles underscore mode on and off.
\~	Inserts a nonbreaking space.

Special format character	Function
\\	Inserts a backslash.
\{...\}	Inserts an opening and closing brace.
\Cvalue;	Sets the color to a specified value.
\File name;	Sets the font based on a specified font file name.
\Hvalue;	Sets the text height to a specified value.
\Hvaluex;	Sets the text height to a multiple of the current text height.
\S...^...;	Stacks the subsequent text at the /, #, or ^ symbol.
\Tvalue;	Adjusts the space between characters, from 0.75 to 4 times.
\Qangle;	Changes obliquing angle.
\Wvalue;	Changes width factor to produce wide text.
\A	Sets the alignment value.
\P	Ends paragraph.
\p[i],[l],[t][X1,X2,...,X32]	Formats paragraph: i = first line indent; l = paragraph offset; t = tab positions.

Working with text written in different languages

You can include text in your drawings that is written in different languages.

Using Unicode characters

PTCAD supports the Unicode character encoding standard, which enables you to display and write text in different languages using different letters. Unicode fonts contain many more characters than typically defined in a system. The following table describes only a small set that is available

Unicode control code	Function
\U+00B0	Draws a degree symbol (°).
\U+00B1	Draws the plus-or-minus symbol (±).
\U+0394	Draws the delta symbol (Δ).
\U+2205	Draws the circle diameter symbol (∅).
\U+03A9	Draws the omega symbol (Ω).
\U+2260	Draws the not equal symbol (≠).

For details about using other special characters, see “Including special text characters” on page 479 in this chapter. You can also use a different text editor; see “Using an alternate text editor” on page 493 in this chapter.

Specifying character sets for drawings

The character set used to display text in drawings typically depends on the character set specified by your operating system, for example, ANSI_1252 is Latin 1, ANSI_1253 is Greek, and ANSI_1254 is Turkish. This is also called the *code page*.

PTCAD allows you to manage which code page is assigned to a drawing. This unique feature is usually not necessary if you distribute drawings within the same geographic region, but it can be very helpful when sharing or distributing drawings between different geographic regions. Changing the code page is highly desirable especially in geographic regions where several languages are used that require different character sets to display text properly.

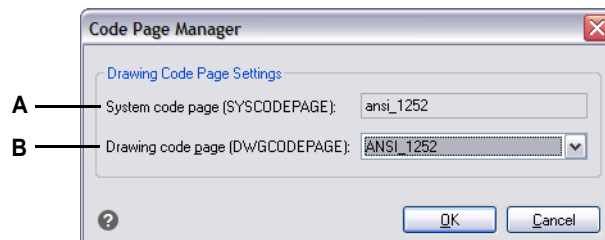
Setting the code page doesn’t change the language of your text; instead it specifies the character set in which the text displays. Some fonts can display its characters several different ways depending on the code page setting.

There are two system variables that relate to the code page of a drawing:

- SYSCODEPAGE — Character set defined for your operating system.
- DWGCODEPAGE — Character set defined for an individual drawing that overrides the operating system code page.

To select a code page for the current drawing

- 1 Type *codepage* and then press Enter.
The Code Page Manager dialog box opens.
- 2 If you want to convert the drawing to use a code page that is different from the operating system code page, in Drawing Code Page, select the new code page for the drawing.
- 3 Click OK.



- A** Displays the code page for the operating system. **B** Select the code page for the current drawing.


Working with missing fonts

When you open a drawing, the program displays text using fonts that are available on your computer: TrueType fonts installed when you started the program and SHX font files located, by default, in the \Fonts folder of your program installation.

If you open a drawing that contains text assigned a font that cannot be found, the program attempts to substitute an alternate font for displaying the text. The following table describes the order in which the program searches for an alternate font, depending on the font type.

TrueType font	SHX font
Font mapping file	Font mapping file
Font defined for the text style	Font defined for the text style
Similar font determined by Windows	FONTALT system variable
	User prompted to select font

To add a search folder for where additional SHX files are stored

- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options.
 - Type *options* and then press Enter.
- 2 Click the Paths/Files tab.
- 3 In the upper half of the dialog box, in Support Paths > Fonts, click Add.
- 4 Enter a path for the folder where SHX font files are located, then press Enter. Or click Browse, select the folder, and then click Open.
- 5 Click OK.


To edit the default font mapping file

- 1 Open the program's .fmp file in a text editor, such as Notepad.
By default, the .fmp file is located in the folder where you installed the program.
- 2 Enter font file names in pairs on a separate line using the following format:


```
missing_font.shx;alternate_font.shx
```

 Font files can be .ttf or .shx font files.
- 3 When finished editing, click Save and close the text editor.

To specify a different font mapping file

- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options.
 - Type *options* and then press Enter.
- 2 Click the Paths/Files tab.
- 3 In the lower half of the dialog box, in Font Mapping, do one of the following:
 - Enter a path and filename for the font map file, then press Enter.
 - Click Browse, select the font map file you want, then click Open.
- 4 Click OK.

To change the FONTALT system variable

- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options.
 - Type *options* and then press Enter.
- 2 Click the Paths/Files tab.
- 3 In the lower half of the dialog box, in Alternate Font, do one of the following:
 - Enter the SHX font file name, then press Enter.
 - Click Browse, select the SHX font file you want, and then click Open.
- 4 Click OK.

Dimensioning your drawing

The PTCAD dimensioning tools let you add measurements to a drawing. You can quickly add dimensions by simply pointing to entities. You can also add tolerance symbols to your drawings.

The program's many dimensioning variables let you control the appearance of the dimensions. With dimension styles, you can save dimension variable settings so you can reuse them without having to re-create them.

This section explains how to:

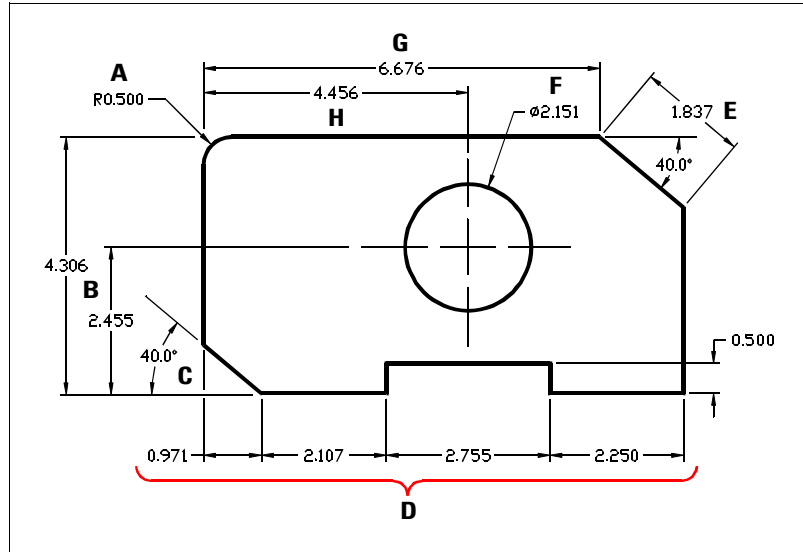
- Create dimensions as linear, angular, arcs, diametral, radial, and ordinate.
- Create leaders and annotations.
- Create center marks and center lines.
- Edit dimensions.
- Use dimension styles and variables.
- Add geometric tolerances.
- Control dimension tolerance.
- Control alternate dimension units.

Topics in this chapter

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<i>Creating dimensions</i>	502
<i>Creating center marks and center lines</i>	514
<i>Editing dimensions</i>	516
<i>Using dimension styles and variables</i>	523
<i>Adding geometric tolerances</i>	538

Understanding dimensioning concepts

You can create five basic types of dimensions: linear, angular, radial, diametral, and ordinate. You can create dimensions for existing entities by selecting them, or you can create dimensions by selecting points within a drawing. For example, you can create a linear dimension either by selecting the entity to be dimensioned or by specifying the first and second extension line origins.



A Radial dimension.

B Vertical linear dimension.

C Angular dimension.

D Linear continued dimensions.

E Aligned dimension.

F Diametral dimension.

G Horizontal linear dimension.

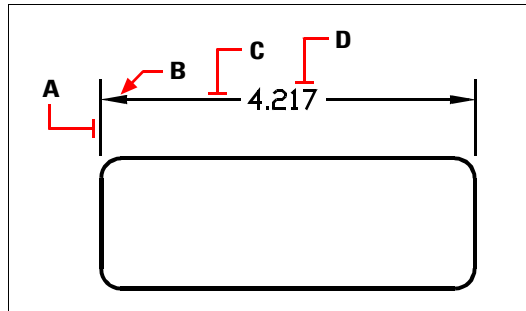
H Linear baseline dimension.

When you create a dimension, the program draws it on the current layer, using the current dimension style. Each dimension has a corresponding dimension style, which controls the appearance of the dimension, such as the types of arrowheads, text style, and colors of various components. You can modify existing dimension styles by changing one of the dimension variable settings and then updating the dimension style to reflect the new settings.

Each dimension you create consists of several parts. A dimension line shows where a dimension begins and ends. When you create an angular dimension, the dimension line is a dimension line arc that subtends the measured angle.

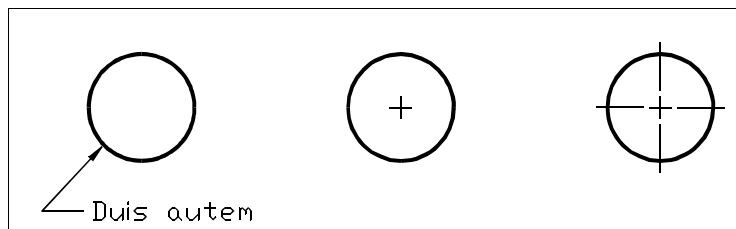
Extension lines, also called projection lines, are lines that extend away from the entity for which you are creating a dimension, so that you can place the dimension line away from the entity. Arrowheads form the termination at each end of the dimension line.

Dimension text contains the measured dimension and can also include prefixes, suffixes, tolerances, and other optional text. As you insert dimensions, you can control the dimension text and specify its position and orientation.



A Extension line.	C Dimension line.
B Arrowhead.	D Dimension text.

Dimensions can also contain other optional components. A leader is a line leading from a feature of the drawing to an annotation. Leaders begin with an arrowhead, and you can use them to place a dimension away from the dimension line or to add notes. When you create a radial dimension, using its dimension style you can add a non-associative center mark, which is a small cross that marks the center of a circle or an arc, or you can add center lines, which are crossing lines that extend out from the center of a circle or an arc. Center marks and center lines can also be created as associative entities.



Leader.	Center mark.	Center mark with centerlines.
---------	--------------	-------------------------------

Dimensions can be one of three types:

- **Associative** — A dimension is linked with the entities it measures. If the entities being measured by the dimension are modified, the dimension is updated automatically. Newly created dimensions are associative when DIMASSOC is set to 2 (default) and created using entity snaps.

- Non-associative — A dimension is not linked with the entities it measures. If the entities being measured by the dimension are modified, the dimension is not updated automatically. Newly created dimensions are non-associative when DIMASSOC is set to 1.
- Exploded — Dimensions are created as separate entities, not a single dimension entity. Newly created dimensions are exploded when DIMASSOC is set to 0.

Creating dimensions

You can create dimensions by:

- Selecting the entity to dimension and specifying the dimension line location.
- Specifying the extension line origins and the dimension line location.

When you create dimensions by selecting an entity, the program automatically places the extension line origins at the appropriate definition points based on the type of entity you select. For example, the definition points are located at the endpoints of arcs, lines, and polyline segments. When you create dimensions by specifying the extension line origins, the points you specify determine the definition points. To establish these points precisely, use entity snaps.

You can create dimensions in model space or paper space.

There are many commands for creating dimensions, described next. Additionally, you can use the Quick Dimension command to quickly create a series of dimensions of the same type, simply follow the prompts.


Creating linear dimensions

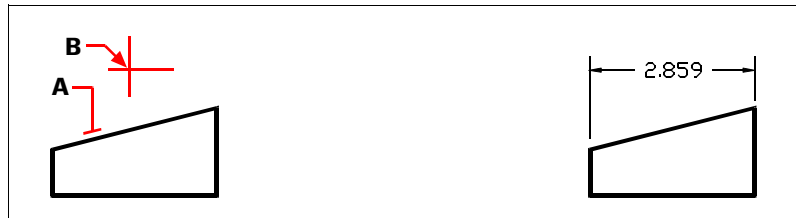
Linear dimensions annotate linear distances or lengths and can be oriented horizontally, vertically, or aligned parallel to an existing entity or to the selected extension origin points. After you create a linear dimension, you can add a baseline dimension or a continued dimension. A linear baseline dimension inserts an additional dimension from a common first extension line origin of a previous linear dimension. A linear continued dimension continues a linear dimension from the second extension line of a previous linear dimension.

Selecting exact points is important when creating dimensions.

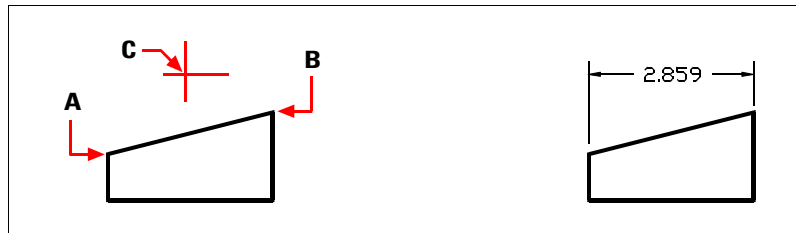
Use entity snaps to select precise ordinate points.

To create a horizontal or vertical dimension

- 1 Do one of the following to choose Linear (- On the ribbon, choose Home > Linear (in Annotation) or Annotate > Linear (in Dimensions).
- On the menu, choose Dimensions > Linear.
- On the Dimensioning toolbar, click the Linear tool.
- Type *dimlinear* and then press Enter.
- 2 Press Enter, and then select the entity to dimension.
Or, you can insert the dimension by specifying the first and second extension line origins.
- 3 Specify the dimension line location.




To insert a linear dimension by selecting the entity, select the entity (A) to dimension, and then specify the dimension line location (B). Result.



To insert a linear dimension by selecting the extension line origins, select the first extension origin (A), select the second extension origin (B), and then specify the dimension line location (C). Result.

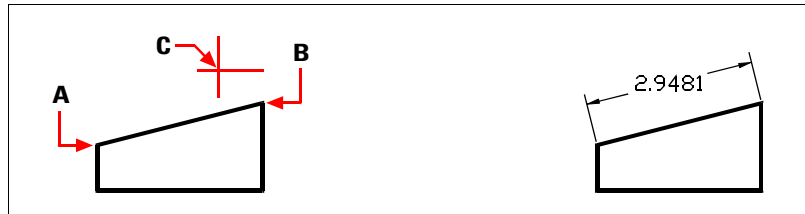
To create an aligned dimension

- 1 Do one of the following to choose Aligned ():
 - On the ribbon, choose Home > Aligned (in Annotation) or Annotate > Aligned (in Dimensions).
 - On the menu, choose Dimensions > Aligned.
 - On the Dimensioning toolbar, click the Aligned tool.
 - Type *dimaligned* and then press Enter.
- 2 Press Enter, and then select the entity to dimension.
Or, you can insert the dimension by specifying the first and second extension line origins.
- 3 Specify the dimension line location.



To insert an aligned dimension by selecting the entity, select the entity (A) to dimension, and then select the dimension line location (B).


Result.



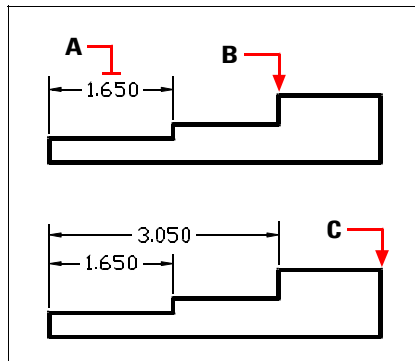
To insert an aligned dimension by selecting the extension line origins, select the first extension origin (A), select the second extension origin (B), and then specify the dimension line location (C).

Result.

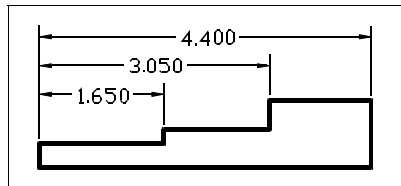
To create a linear baseline dimension

- 1 Create a dimension.
- 2 Do one of the following to choose Baseline (- On the ribbon, choose Annotate > Baseline (in Dimensions).
- On the menu, choose Dimensions > Baseline.
- On the Dimensioning toolbar, click the Baseline tool.
- Type *dimbaseline* and then press Enter.
- 3 To select a starting dimension, press Enter.
- 4 Select the next extension line origin, and then press Enter.
 Or press Enter, and then select an existing dimension for the baseline. Select the origin of the next extension line, and then press Enter.

The program automatically places the new baseline dimension above or below the previous dimension line. The distance between the two dimension lines is determined by the Baseline Offset value in the Dimension Styles dialog box.




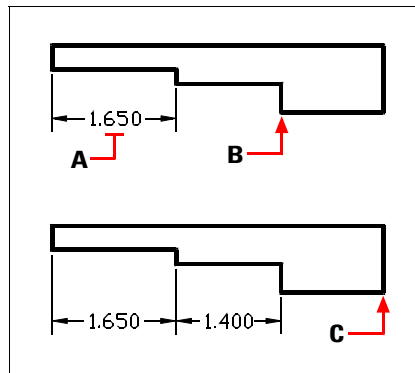
To add a baseline dimension to an existing linear dimension, select the existing dimension (A), select the next extension line origin (B), and select as many additional points as you want (C).



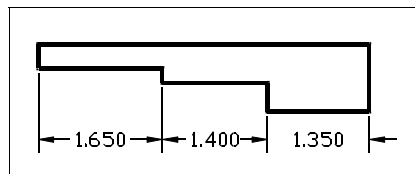
Result.

To create a linear continued dimension

- 1 Create a dimension.
- 2 Do one of the following to choose Continue (- On the ribbon, choose Annotate > Continue (in Dimensions).
- On the menu, choose Dimensions > Continue.
- On the Dimensioning toolbar, click the Continue tool.
- Type *dimcontinue* and then press Enter.
- 3 To select a starting dimension, press Enter.
- 4 Select the next extension line origin, and then press Enter.
Or press Enter, and then select an existing dimension to continue.
- 5 To add continued dimensions, continue selecting extension line origins.
- 6 To end the command, press Enter twice.



To add a continued dimension to an existing linear dimension, select the existing dimension (A), select the next extension line origin (B), and select another extension line origin (C).




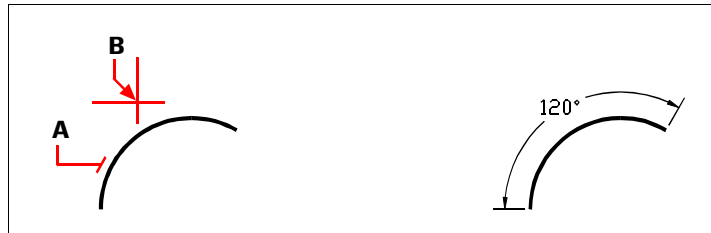
Result.

Creating angular dimensions

Angular dimensions annotate the angle measured between two lines. You can also dimension an angle by selecting an angle vertex and two endpoints. After you create an angular dimension, you can add a baseline dimension or a continued dimension. An angular baseline dimension inserts an additional dimension from a common first extension line origin of a previous angular dimension. An angular continued dimension continues an angular dimension from the second extension line of a previous angular dimension.

To dimension an angle encompassed by an arc


- 1 Do one of the following to choose Angular ():
 - On the ribbon, choose Home > Angular (in Annotation) or Annotate > Angular (in Dimensions).
 - On the menu, choose Dimensions > Angular.
 - On the Dimensioning toolbar, click the Angular tool.
 - Type *dimangular* and then press Enter.
- 2 Select the arc.
- 3 Specify the dimension arc location.

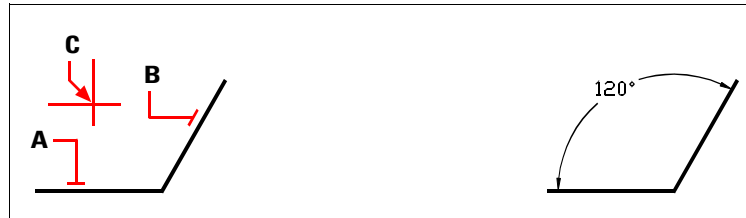


To dimension the angle subtended by an arc, select the arc (A), and then specify the dimension arc location (B).

Result.

To dimension an angle between two lines

- 1 Do one of the following to choose Angular ():
 - On the ribbon, choose Annotate > Angular (in Dimensions).
 - On the menu, choose Dimensions > Angular.
 - On the Dimensioning toolbar, click the Angular tool.
 - Type *dimangular* and then press Enter.
- 2 Select one line.
- 3 Select the other line.
- 4 Specify the dimension line location.




Select one line (A), select the other line (B), and then specify the dimension line location (C).

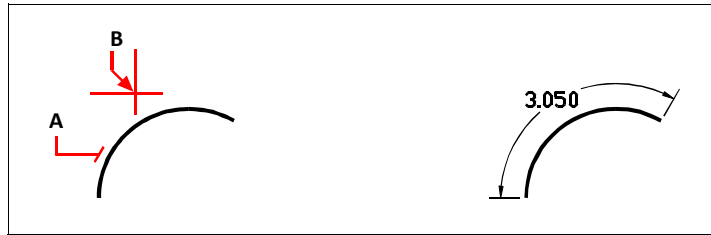
Result.

Creating arc dimensions

Arc dimensions annotate the length of an arc or arc segment. You can also dimension a portion of an arc by selecting two points. After you create an arc dimension, you can change its text to the arc angle or to any other text.


To dimension an arc length

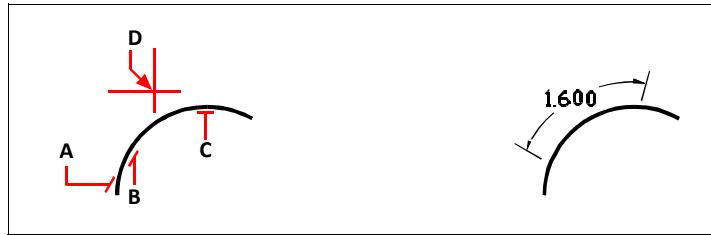
- 1 Do one of the following to choose Arc ():
 - On the ribbon, choose Home > Arc (in Annotation) or Annotate > Arc (in Dimensions).
 - On the menu, choose Dimensions > Arc.
 - On the Dimensioning toolbar, click the Arc tool.
 - Type *dimarc* and then press Enter.
- 2 Select the arc or arc segment.
- 3 Specify the dimension arc location.



To dimension the length of an arc, select the arc (A), and then specify the dimension arc location (B). Result.

To dimension part of an arc length

- 1 Do one of the following to choose Arc ():
 - On the ribbon, choose Home > Arc (in Annotation) or Annotate > Arc (in Dimensions).
 - On the menu, choose Dimensions > Arc.
 - On the Dimensioning toolbar, click the Arc tool.
 - Type *dimarc* and then press Enter.
- 2 Choose Partial.
- 3 Select the start point of the arc length you want to measure.
- 4 Select the end point.




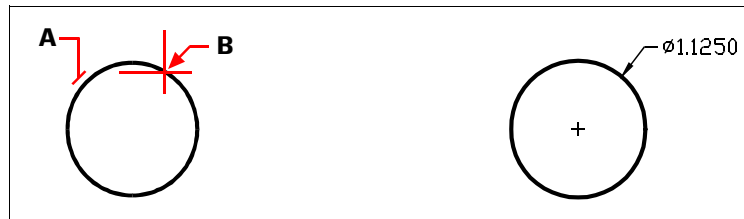
To dimension part of an arc length, select the arc (A), select the start point (B) and end point (C) of the portion to measure, then specify the dimension arc location (D). Result.

Creating diametral and radial dimensions

Diameter and radius dimensions annotate the radii and diameters of arcs and circles. You can optionally include centerlines or center marks.


To create a diametral dimension

- 1 Do one of the following to choose Diameter (- On the ribbon, choose Home > Diameter (in Annotation) or Annotate > Diameter (in Dimensions).
- On the menu, choose Dimensions > Diameter.
- On the Dimensioning toolbar, click the Diameter tool.
- Type *dimdiameter* and then press Enter.
- 2 Select the arc or circle.
- 3 Specify the dimension line location.



Select the circle (A), and then specify the dimension line location (B). Result.

To create a radial dimension

- 1 Do one of the following to choose Radius (- On the ribbon, choose Home > Radius (in Annotation) or Annotate > Radius (in Dimensions).
- On the menu, choose Dimensions > Radius.
- On the Dimensioning toolbar, click the Radius tool.
- Type *dimradius* and then press Enter.
- 2 Select the arc or circle.
- 3 Specify the dimension line location.

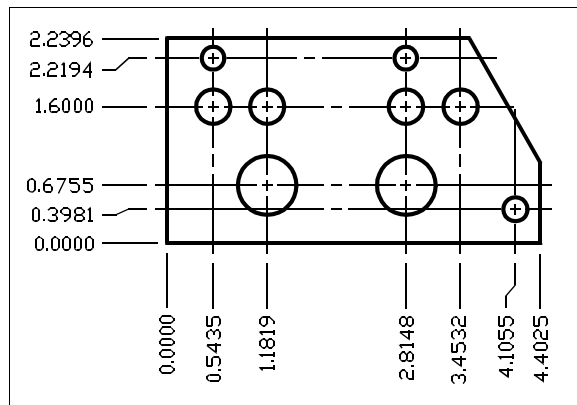


Select the circle (A), and then specify the dimension line location (B). Result.

Creating ordinate dimensions


An ordinate dimension annotates the perpendicular distance from an origin or base point (the origin of the current user coordinate system [UCS]). Ordinate dimensions consist of an x- or y-coordinate and a leader. An x-ordinate dimension measures distances along the x-axis; a y-ordinate dimension measures distances along the y-axis.

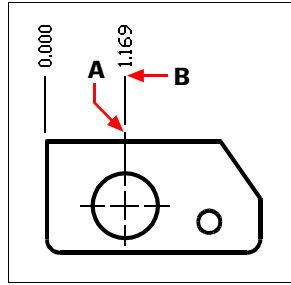
As you select ordinate points, the program automatically determines whether the point is an x- or y-ordinate based on which direction you drag the second point. You can also specify whether the ordinate represents an x- or y-ordinate. Ordinate dimension text is always aligned with the ordinate leader lines, regardless of the text orientation specified by the current dimension style.



Ordinate dimensions measure the distance along the x- or y-axis from an origin to a selected ordinate point.

To create an ordinate dimension

- 1 Do one of the following to choose Ordinate (- On the ribbon, choose Annotate > Ordinate (in Dimensions).
- On the menu, choose Dimensions > Ordinate.
- On the Dimensioning toolbar, click the Ordinate tool.
- Type *dimordinate* and then press Enter.
- 2 Select the point for ordinate dimension.
- 3 Specify the ordinate leader endpoint.



Select the ordinate point (A), and then specify the ordinate leader endpoint (B).


Selecting exact points is important when creating dimensions.

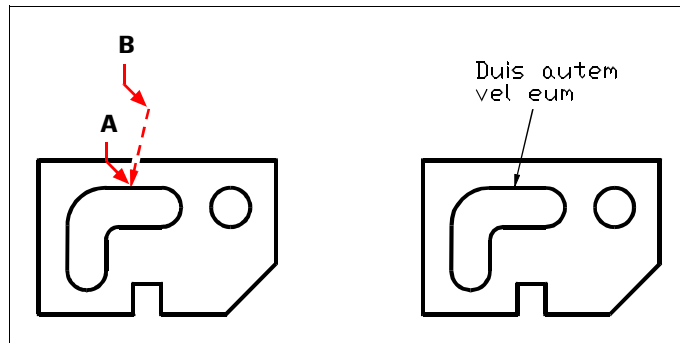
Use entity snaps to select precise ordinate points.

Creating leaders and annotations

Leaders consist of a line or series of lines that connects a feature in a drawing to an annotation. Generally, you place an arrowhead at the first point. An annotation, created as dimension text, is placed immediately adjacent to the last point. By default, the text placed at the end of the leader line consists of the most recent dimension. You can also type an annotation as a single line of text.

To create a leader and an annotation

- 1 Do one of the following to choose Leader ():
 - On the ribbon, choose Annotate > Leader (in Dimensions).
 - On the menu, choose Dimensions > Leader.
 - On the Dimensioning toolbar, click the Leader tool.
 - Type *dimleader* and then press Enter.
- 2 Specify the starting point of the leader.
- 3 Specify the endpoint of the leader line segment.
- 4 Specify additional leader line segment endpoints.
- 5 After you specify the last endpoint, press Enter.
- 6 Type the annotation, or press Enter to accept the most recent dimension as the default annotation.



Specify the starting point of the leader (A) and the endpoint of the leader line segment (B).

Result.

Dimensioning model space entities in paper space

To increase efficiency, you can separate your drawing model from annotations using the Model and Layout tabs.

It takes time to display dimensions, title blocks, keynotes, and other annotations. If you draw these on a Layout tab, display-time and visual clutter are reduced when you work on your model (on the Model tab). PTCAD allows you to dimension model space entities on either the Model tab or a Layout tab — you can make the choice depending on the method that works best for your needs.

To dimension model space entities in paper space

- 1 Click a Layout tab.
- 2 Create at least one layout viewport. For details, see “Creating layout viewports” on page 623.
- 3 Select the edge of the layout viewport that you want to use for creating dimensions and lock the layout viewport by doing one of the following:
 - Click Viewport Lock/Unlock in the status bar.
 - Right-click the edge of the layout viewport, choose Properties, then choose Lock Viewports.

Locking the viewport is not required, but it is extremely helpful when you zoom or pan in the layout viewport; it prevents the viewport scale and view center from changing.

You can work in a layout viewport without having it clutter your display or selections. Place layout viewports on their own layer, and after locking the layout viewports, hide the layer that contains them.

- 4 Make sure you are working in paper space by verifying that Model or Paper Space in the status bar begins with “P.” If necessary, switch to paper space by double-clicking Model or Paper Space in the status bar.
- 5 Create a dimension. You can select the model space entities directly, specify definition points, or use entities snaps to help accurately select the definition points. The dimension is created in paper space.

For more details about using paper space and model space, see “Understanding paper space and model space” on page 616.

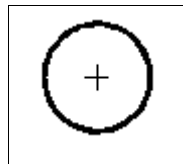
Creating center marks and center lines

Center marks and center lines are associative dimensioning references that do not contain measurement data. A center mark is a small cross that marks the center of a circle or an arc. Center lines are linear entities drawn between two lines or two linear segments of polylines.

Both are associative, meaning that when the associated entity changes, so does the center mark or center line entity. This is different than the non-associative center marks and lines that can be drawn with dimension styles.


Creating center marks

Center marks can be added to a circle or arc, for example, marking a hole in a mechanical part. When you move or resize the associated entity, the center mark entity also moves or resizes.



Center mark.

To create a center mark

- 1 Do one of the following to choose Center Mark (- On the ribbon, choose Annotate > Center Mark (in Center Lines).
- On the menu, choose Dimensions > Center Mark.
- On the Dimensioning toolbar, click the Center Mark tool.
- Type *centermark* and then press Enter.
- 2 Select the circle or arc that you want to add a center mark to.
- 3 Continue to select circles or arcs, or press Enter to end the command.


You can change the appearance of center marks.

To change the appearance of existing center marks, use the Properties pane or grips. To change the appearance of new center marks, before you draw them modify the CENTERCROSSIZE, CENTERCROSSGAP, CENTEREXE, CENTERLTYPE, CENTERLTSCALE, and CENTERMARKEXE system variable values.

Creating center lines

A center line is an associative linear entity drawn between two lines or two linear segments of polylines. The center line is drawn using the apparent midpoints between the two start points and the two endpoints of the selected lines or segments. If selected lines or segments are not parallel, the center line is drawn using the intersection point—real or imaginary—and the midpoint between the endpoints of the selected lines.

To create a center line

- 1 Do one of the following to choose Center Line (- On the ribbon, choose Annotate > Center Line (in Center Lines).
- On the menu, choose Dimensions > Center Line.
- On the Dimensioning toolbar, click the Center Line tool.
- Type *centerline* and then press Enter.
- 2 Select the first line or linear polyline segment.
- 3 Select the second line or linear polyline segment.

If specifying linear polyline segments, only the segment you pick, not the whole polyline, is used to calculate the location of the center line.

You can change the appearance of center lines.

To change the appearance of existing center lines, use the Properties pane or grips. To change the appearance of new center lines, before you draw them modify the CENTERLTYPE, CENTERLTSCALE, and CENTEREXE system variable values.


Editing dimensions

The easiest way to edit dimensions is by using grips. Some tasks require an additional command though, such as the Dimension Break command for adding space around dimension and extension lines where they intersect other entities. This helps prevent dimensions from looking as if they are part of drawn entities. You might also want to quickly distribute equal space between the dimension lines of several dimensions. You can also use the Edit Dimension Text command to modify the dimension text and reposition it anywhere along the dimension line or the Make Oblique command to rotate dimension lines and dimension text at any angle.

Spacing dimension lines

If your drawing contains several dimensions, you can distribute equal space between their dimension lines. The dimension must be linear or angular with dimension lines that are parallel.

To distribute equal space between dimension lines

- 1 Do one of the following to choose Adjust Spacing ():
 - On the ribbon, choose Annotate > Adjust Spacing (in Dimensions).
 - On the menu, choose Dimensions > Adjust Spacing.
 - On the Dimensioning toolbar, click the Adjust Spacing tool.
 - Type *dimspace*, then press Enter.
- 2 Select a linear or angular dimension from which dimension lines of other dimensions are to be equally spaced.
- 3 Select the parallel linear or angular dimensions to adjust, and press Enter when done.
- 4 Enter a value for the spacing distance, or press Enter to adjust the distance automatically to two times the text height of the base dimension.

Breaking dimensions

A dimension break is a space on either side of an entity that intersects a dimension or extension line. Adding a break keeps the dimension from looking as if it is a part of drawn entities, and it can add clarity where dimension, extension, or leader lines intersect.


Dimension breaks can be applied to multileaders too.

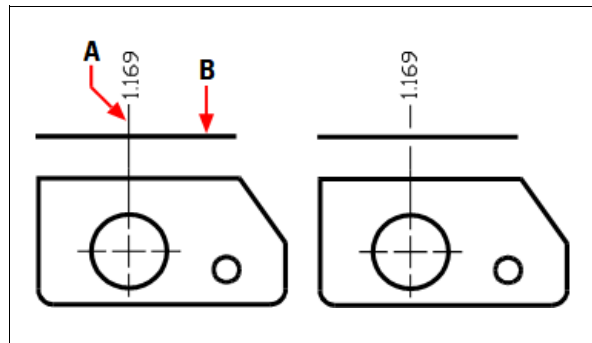
After a break is added, it is updated automatically when the dimension, multileader, or intersecting entity is modified.

The size of a break gap is determined by the assigned dimension style.

To specify the break size, use the Dimension Styles Manager command. For more details, see “Using dimension styles and variables” on page 523 in this chapter.

To break a dimension (or multileader)


- 1 Do one of the following to choose Dimension Break (- On the ribbon, choose Annotate > Dimension Break (in Dimensions).
- On the menu, choose Dimensions > Dimension Break.
- On the Dimensioning toolbar, click the Make Oblique tool.
- Type *dimbreak*, then press Enter.
- 2 Select the dimension or multileader that you want to break. To break or restore more than one dimension or multileader, choose Multiple, then select them.
- 3 Do one of the following:
 - Select the entity that crosses the dimension or multileader where the break is needed.
 - Choose Auto to place breaks at each point where the selected dimension intersects with other entities.
 - Choose Manual to add a break to the selected dimension or multileader manually, then specify two break points (not available when Multiple is selected).
- 4 When finished, press Enter.



Specify the dimension (A) and the intersecting entity (B).

Result.


To remove breaks from a dimension (or multileader)

- 1 Do one of the following to choose Dimension Break (- On the ribbon, choose Annotate > Dimension Break (in Dimensions).
- On the menu, choose Dimensions > Dimension Break.
- On the Dimensioning toolbar, click the Make Oblique tool.
- Type *dimbreak*, then press Enter.
- 2 Select the dimension or multileader that you want to break. To break or restore more than one dimension or multileader, choose Multiple, then select them.
- 3 Choose Remove to remove all existing breaks from the selected dimensions or multileaders.

Making dimensions oblique

Extension lines are normally created at a perpendicular angle to the dimension line. You can change the angle of the extension lines, however, so that they tilt relative to the dimension line.

To make oblique extension lines

- 1 Do one of the following to choose Make Oblique ():
 - On the ribbon, choose Annotate > Make Oblique (in Dimensions).
 - On the menu, choose Dimensions > Make Oblique.
 - On the Dimensioning toolbar, click the Make Oblique tool.
 - Type *dimedit*, press Enter, and then in the prompt box, choose Oblique Lines.
- 2 Select the linear dimension, and then press Enter.
- 3 Type the obliquing angle, and then press Enter.



Select the dimension to be made oblique (A), and then type the obliquing angle.

Result.

You can align the oblique angle if you don't know the exact measurement.

Use entity snaps to pick two points on the entity.


Editing dimension text

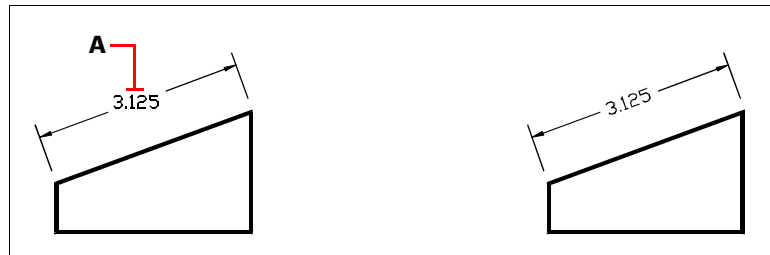
You can rotate the text of an existing dimension, move the dimension text to a new position, or replace selected dimension text with new text. You can also restore dimension text to its original position as defined by the current dimension style.

When you rotate or replace selected text, you specify the change first, and then select one or more dimensions to which to apply the change. All the selected dimensions are updated simultaneously.

To rotate dimension text

Advanced experience level

- 1 Do one of the following to choose Rotate Dimension Text (- On the ribbon, choose Annotate > Rotate Dimension Text (in Dimensions).
- On the menu, choose Dimensions > Rotate Dimension Text.
- On the Dimensioning toolbar, click the Rotate Dimension Text tool.
- Type *dimedit*, press Enter, and then in the prompt box, choose Rotate Text.
- 2 Type the new dimension text angle, and then press Enter.
- 3 Select the dimension to be rotated, and then press Enter.



Select the dimension to be rotated (A), and then type the rotation angle.


Result.

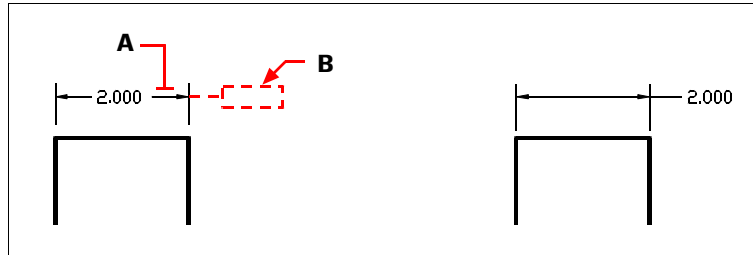
The dimension text angle is relative to the dimension line.

If the dimension text rotation is set to zero, the text angle is defined by the dimension type and the dimension style.

To move dimension text

Advanced experience level


- 1 Do one of the following to choose Reposition Dimension Text ():
 - On the ribbon, choose Annotate > Reposition Dimension Text (in Dimensions).
 - On the menu, choose Dimensions > Reposition Dimension Text.
 - On the Dimensioning toolbar, click the Reposition Dimension Text tool.
 - Type *dimtedit* and then press Enter.
- 2 Select the dimension to reposition text.
- 3 Select the new text position.



Select the dimension to be moved (A), and then select the new text position (B). Result.


To restore dimension text to its home position

Advanced experience level


- 1 Do one of the following to choose Restore Text Position ():
 - On the ribbon, choose Annotate > Restore Text Position (in Dimensions).
 - On the menu, choose Dimensions > Restore Text Position.
 - On the Dimensioning toolbar, click the Restore Text Position tool.
 - Type *dimedit*, press Enter, and then in the prompt box, choose Restore Text.
- 2 Select the dimension text to restore, and then press Enter.

To replace existing dimension text with new text


Advanced experience level

- 1 Do one of the following to choose Edit Dimension Text ():
 - On the ribbon, choose Annotate > Edit Dimension Text (in Dimensions).
 - On the menu, choose Dimensions > Edit Dimension Text.
 - On the Dimensioning toolbar, click the Edit Dimension Text tool.
 - Type *dimedit* and then in the prompt box, choose Edit Text.
- 2 Type the new dimension text, and then press Enter.
- 3 Select the dimension to be replaced, and then press Enter.

To restore dimension text to its original text value

- 1 Do one of the following to choose Reset Dimension Text (- On the ribbon, choose Express Tools > Reset Dimension Text (in Dimension).
- On the menu, choose Express Tools > Dimensions > Reset Dimension Text.
- Type *dimreassoc* and then press Enter.
- 2 Select the dimension text to restore, and then press Enter.

To reassociate dimension text with a new entity or point on an entity

- 1 Do one of the following to choose Reassociate Dimensions (- On the ribbon, choose Annotate > Reassociate Dimensions (in Dimensions).
- On the menu, choose Dimensions > Reassociate Dimensions.
- Type *dimreassociate* and then press Enter.
- 2 Select the dimensions you want to reassociate with a different entity or point, and then press Enter.
- 3 Follow the prompts to select a new entity or point on an entity. The prompts that display vary according to the type of dimensions you selected.

Using dimension styles and variables

Dimensions that you insert are created using the current dimension style. You can create, save, restore, and delete named dimension styles.

To display information about the current dimension style and compare it with other style names, you can use the *dimstyle* command.

Creating a dimension style


Dimension styles provide a way for you to change various settings that control the appearance of dimensions. You can then save those settings for reuse. If you don't define a dimension style before creating dimensions, the program uses the Standard dimension style, which stores the default dimension variable settings. Each option in the Dimension Styles dialog box relates to a variable that you can set manually. See the online Command Reference for more information.

To create a dimension style

- 1 Do one of the following to choose Dimension Styles Manager (

Selecting a dimension style

To select a dimension style

- 1 Do one of the following to choose Dimension Styles Manager (


Renaming a dimension style

To rename a dimension style

- 1 Do one of the following to choose Dimension Styles Manager (

Deleting a dimension style

To delete a named dimension style

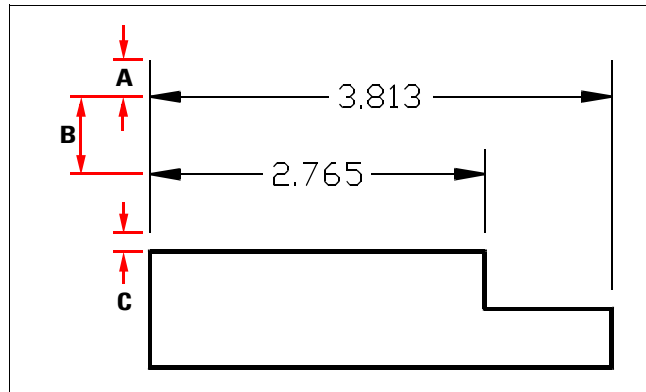
- 1 Do one of the following to choose Dimension Styles Manager ():
 - On the ribbon, choose Annotate > Dimension Styles Manager (in Dimensions).
 - On the menu, choose Dimensions > Dimension Styles Manager or choose Format > Dimension Styles Manager.
 - On the Dimensioning or Styles toolbar, click the Dimension Styles Manager tool.
 - Type *setdim* and then press Enter.
- 2 In the Dimension Styles dialog box, select the dimension style to delete.
- 3 Click Delete.
- 4 To confirm the deletion, click OK.
- 5 Click OK.

To display information about the current style

- 1 Type *dimstyle* and then press Enter.
- 2 Type *v* and then press Enter to display information about the variables.
- 3 Type the dimension style name, and press Enter.

Controlling line settings

You can control settings affecting dimension lines, extension lines, and center marks. Any changes you make affect the current dimension style. The image tile on the right side of the Dimension Styles dialog box shows the appearance of the dimensions based on the current dimension style settings.




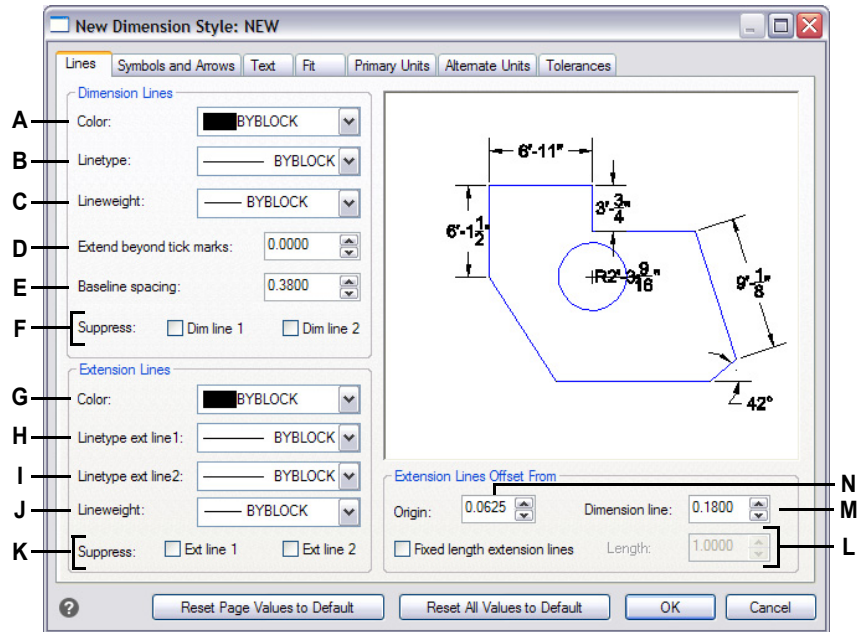
A Extend past dimension.

C Offset from origin.

B Baseline offset.

To set the color for dimension lines

- 1 Do one of the following to choose Dimension Styles Manager (- On the ribbon, choose Annotate > Dimension Styles Manager (in Dimensions).
- On the menu, choose Dimensions > Dimension Styles Manager or choose Format > Dimension Styles Manager.
- On the Dimensioning or Styles toolbar, click the Dimension Styles Manager tool.
- Type *setdim* and then press Enter.
- 2 Click the Lines tab.
- 3 Make your selections.
- 4 Click OK.



- A Select the dimension line color.
- B Select the dimension linetype.
- C Select the dimension lineweight.
- D Type or select the distance that dimension lines extend beyond tick marks.
- E Type or select the baseline offset distance (the distance to offset successive dimension lines when creating baseline dimensions) and the offset from origin (the distance extension lines are offset from their origin points).
- F Click to prevent the creation of the first or second dimension line.
- G Select the extension line color.
- H Select the first extension line linetype.
- I Select the second extension line linetype.
- J Select the extension lineweight.
- K Click to prevent the creation of the first or second extension line.
- L Click to set the length of extension lines to always be a set length, then type or select the extension line length.
- M Type or select the distance to offset extension lines from dimension lines.
- N Type or select the distance to offset extension lines from their origin.


Controlling dimension arrows

You can control the appearance and size of arrowheads or tick marks placed at the ends of dimension lines. Any changes you make affect the current dimension style. The arrowheads you choose display in the image tile on the right side of the Dimension Styles dialog box.

You can choose from a number of arrowhead types. You can specify different arrowheads for each end of a dimension line and for leader lines. The Starting arrow corresponds to the first extension line; the Ending arrow corresponds to the second extension line. Blocks defined in the drawing also display in the three Arrowhead lists as user-defined arrows. You can use these blocks to create and assign your own arrowheads.

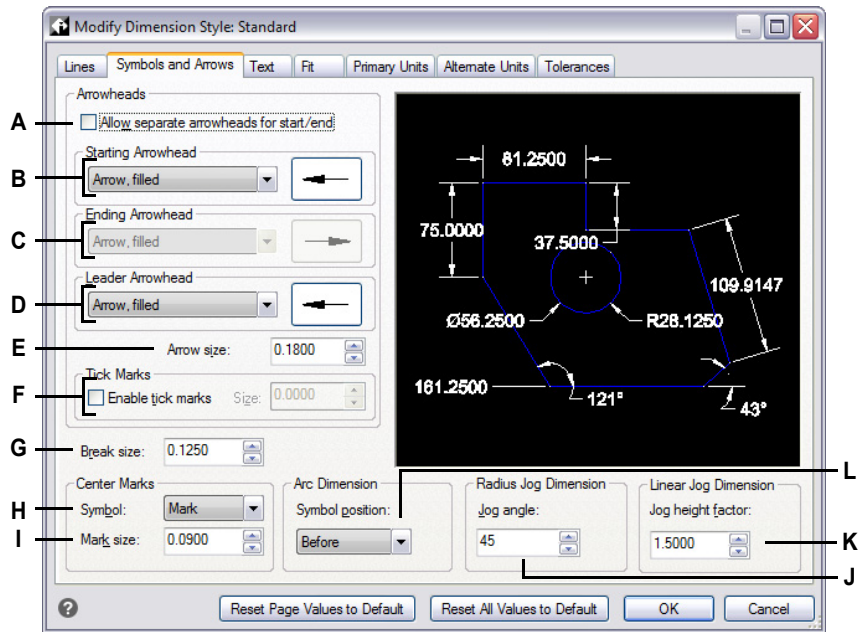
The Arrow Size value determines the size of the arrowhead, measured in drawing units. You can also use tick marks instead of arrowheads.

To choose an arrowhead

- 1 Do one of the following to choose Dimension Styles Manager ():
 - On the ribbon, choose Annotate > Dimension Styles Manager (in Dimensions).
 - On the menu, choose Dimensions > Dimension Styles Manager or choose Format > Dimension Styles Manager.
 - On the Dimensioning or Styles toolbar, click the Dimension Styles Manager tool.
 - Type *setdim* and then press Enter.
- 2 Click the Symbols and Arrows tab.
- 3 In the Starting Arrowhead or Ending Arrowhead list, click to select the starting or ending arrowhead, respectively. If necessary, mark Allow Separate Arrowheads if you want to use different starting and ending arrowheads.
- 4 In the Leader Arrowhead list, click to select a leader arrowhead for leader lines.
- 5 Click OK.

Use the system variable.

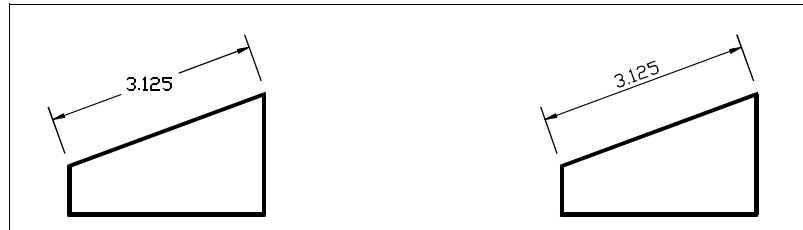
The DIMLDRBLK system variable also specifies leader arrow types.



- A Select to enable the ability to use different starting and ending arrowheads.
- B Click to select a starting arrowhead, or click the arrowhead picture to scroll the list automatically.
- C Click to select an ending arrowhead, or click the arrowhead picture to scroll the list automatically.
- D Click to select a leader arrowhead, or click the arrowhead picture to scroll the list automatically.
- E Type or select arrowhead size.
- F Select to enable tick marks instead of arrowheads, then type or select tick mark size.
- G Enter the size of dimension breaks.
- H Select the symbol for center marks.
- I Type or select center mark size. Positive values create a center mark. Negative values create centerlines.
- J Type or select the angle for the jog of radius dimensions.
- K Type or select the scale factor for the jog height of linear dimensions.
- L Select where to position arc symbols for arc dimensions.

Controlling dimension text


You can control the settings affecting the appearance of dimension text. Any changes you make affect the current dimension style. The image tile on the right side of the Dimension Styles dialog box shows the appearance of the dimensions based on the current dimension style settings.

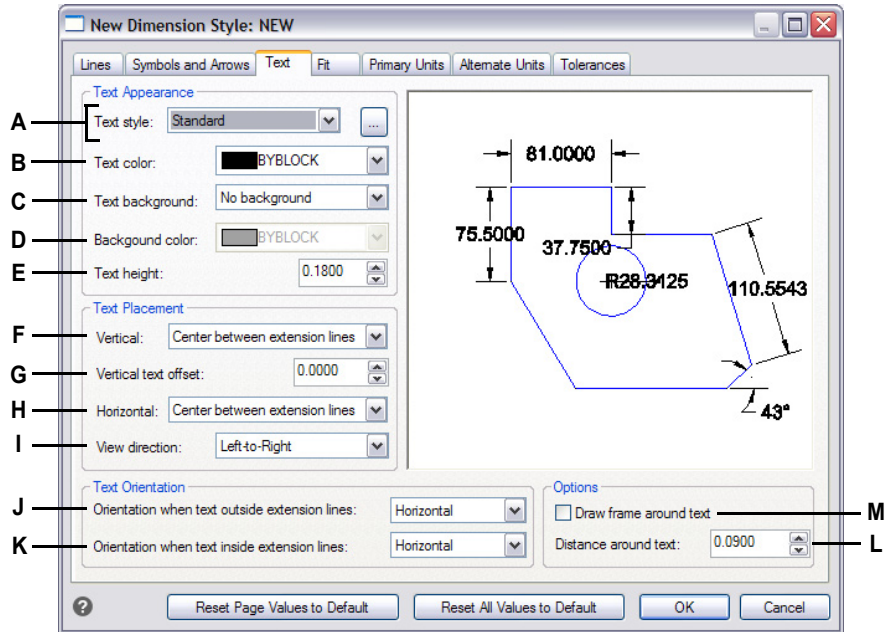


Text between extension lines aligned horizontally.

Text between extension lines aligned with dimension line.

To align dimension text with the dimension line

- 1 Do one of the following to choose Dimension Styles Manager ():
 - On the ribbon, choose Annotate > Dimension Styles Manager (in Dimensions).
 - On the menu, choose Dimensions > Dimension Styles Manager or choose Format > Dimension Styles Manager.
 - On the Dimensioning or Styles toolbar, click the Dimension Styles Manager tool.
 - Type *setdim* and then press Enter.
- 2 Click the Text tab.
- 3 Make your selections.
- 4 Click OK.

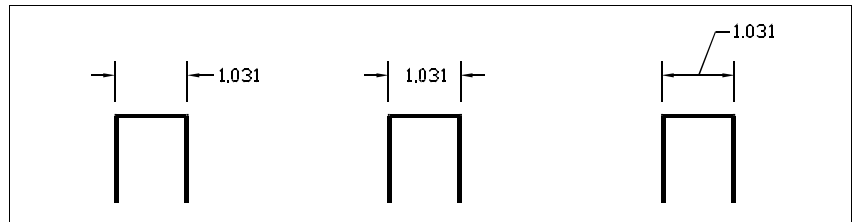


- A** Click to select the text style used for dimension text. Click [...] to manage text styles.
- B** Select the dimension text color.
- C** Select none, drawing color, or user color for the dimension text background color.
- D** Select the color for dimension text backgrounds (available if User Color is selected for Text Background).
- E** Type or select the text height, measured in drawing units.
- F** Select the vertical justification of dimension text.
- G** Type or select the vertical text offset distance.
- H** Select the horizontal justification of dimension text.
- I** Select the view direction of dimension text.
- J** Select the alignment when dimension text is outside extension lines.
- K** Select the alignment when dimension text is inside extension lines.
- L** Type or select the distance around the dimension text.
- M** Click to include a frame around dimension text.

Controlling dimension fit

You can control the way dimension text and arrowheads are placed in relation to the dimension lines. You can also control how the dimension scales by choosing whether it is annotative by default or whether it scales according to a specific scale or according to the layout. Any changes you make affect the current dimension style. The image tile on the right side of the Dimension Styles dialog box shows the appearance of dimensions based on the current dimension style settings.

The program determines whether both dimension text and arrowheads will fit between the extension lines by comparing the distance between the extension lines to the size of the dimension text, the size of the arrowheads, and the amount of space required around dimension text. The program applies the best fit method based on the available space. If possible, both the dimension text and arrowheads are placed between the extension lines. If both will not fit between the extension lines, you can determine how text and arrowheads are placed using the Fit Method settings on the Dimension Styles dialog box.




Text and arrowheads placed outside extension lines.

Text placed between extension lines and arrowheads outside extension lines.

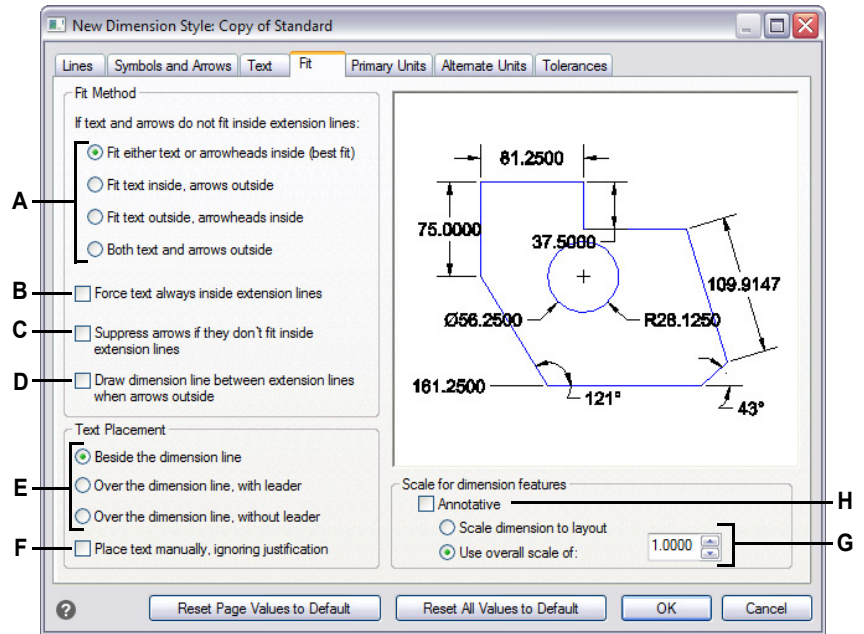
Text placed above dimension line with a leader connecting the text to the dimension line.

To format the fit of dimensions

- 1 Do one of the following to choose Dimension Styles Manager ():
 - On the ribbon, choose Annotate > Dimension Styles Manager (in Dimensions).
 - On the menu, choose Dimensions > Dimension Styles Manager or choose Format > Dimension Styles Manager.
 - On the Dimensioning or Styles toolbar, click the Dimension Styles Manager tool.
 - Type *setdim* and then press Enter.
- 2 Click the Fit tab.
- 3 Click the options that you want.
- 4 Click OK.

Use the system variables.

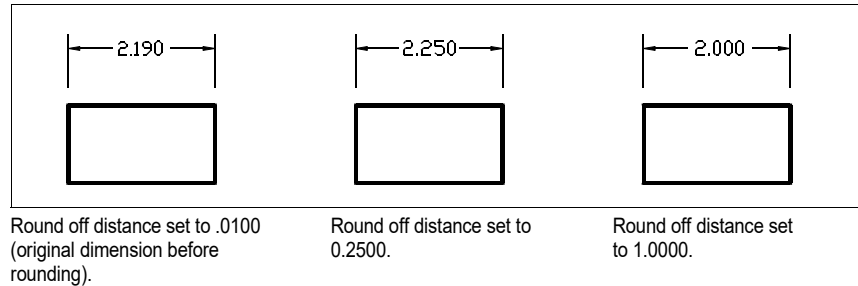
The *DIMATFIT* system variable specifies how dimension text and arrows are arranged. The *DIMTMOVE* system variable specifies how dimension text is moved.




- A Select how to fit text and arrows if they both do not fit inside extension lines.
- B Select to always place text inside extension lines.
- C Select to prevent the creation of arrows if they don't fit inside extension lines.
- D Select to draw dimension lines between extension lines when text and arrows are placed outside extension lines.
- E Select how to position text relative to dimension lines and whether to include a leader.
- F Select to be prompted for text placement when creating dimensions.
- G Select to scale the dimension according to the layout or enter a specific scale to apply to all dimension style settings (available only if Annotative is not selected.)
- H Select to make the dimension support annotative scaling by default.

Controlling primary dimension units

You can determine the appearance and format of the primary dimension units. The image tile on the right side of the Dimension Styles dialog box shows the appearance of the dimensions based on the current dimension style settings.

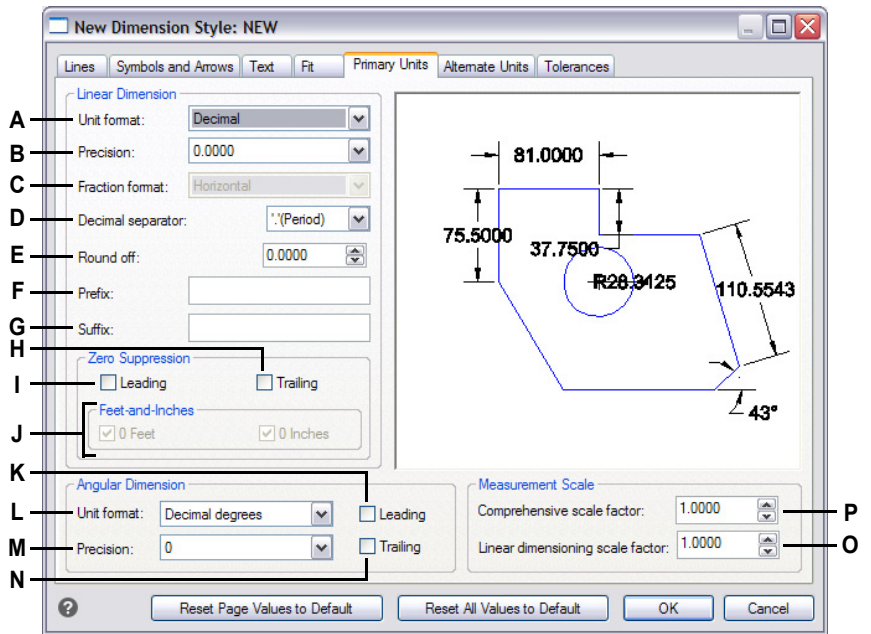


To set the primary units of dimensions

- 1 Do one of the following to choose Dimension Styles Manager ():
 - On the ribbon, choose Annotate > Dimension Styles Manager (in Dimensions).
 - On the menu, choose Dimensions > Dimension Styles Manager or choose Format > Dimension Styles Manager.
 - On the Dimensioning or Styles toolbar, click the Dimension Styles Manager tool.
 - Type *setdim* and then press Enter.
- 2 Click the Primary Units tab.
- 3 Make your selections.
- 4 Click OK.

Use the system variables.

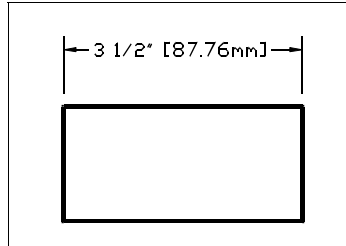
The DIMLUNIT system variable specifies units for linear dimensions. The DIMFRAC system variable specifies fraction formats.



- A Select the linear dimension unit format.
- B Type or select the number of decimal places for text of linear dimensions.
- C Select the format for text fractions of linear dimensions.
- D Enter the marker symbol used for decimals.
- E Type or select the nearest value to round to for linear distances.
- F Type a prefix to be appended to linear dimension text.
- G Type a suffix to be appended to linear dimension text.
- H Select to prevent the inclusion of trailing zeros for linear dimension text.
- I Select to prevent the inclusion of leading zeros for linear dimension text.
- J Select to prevent the inclusion of inches or feet in dimension text when the corresponding number of inches or feet is zero.
- K Select to prevent the inclusion of leading zeros for angular dimensions.
- L Select the angular dimension unit format.
- M Type or select the number of decimal places for angular dimensions.
- N Select to prevent the inclusion of trailing zeros for angular dimensions.
- O Type or select the linear scale factor applied to all lengths measured by dimensioning commands.
- P Type or select the scale factor applied to all dimensions.


Controlling alternate dimension units

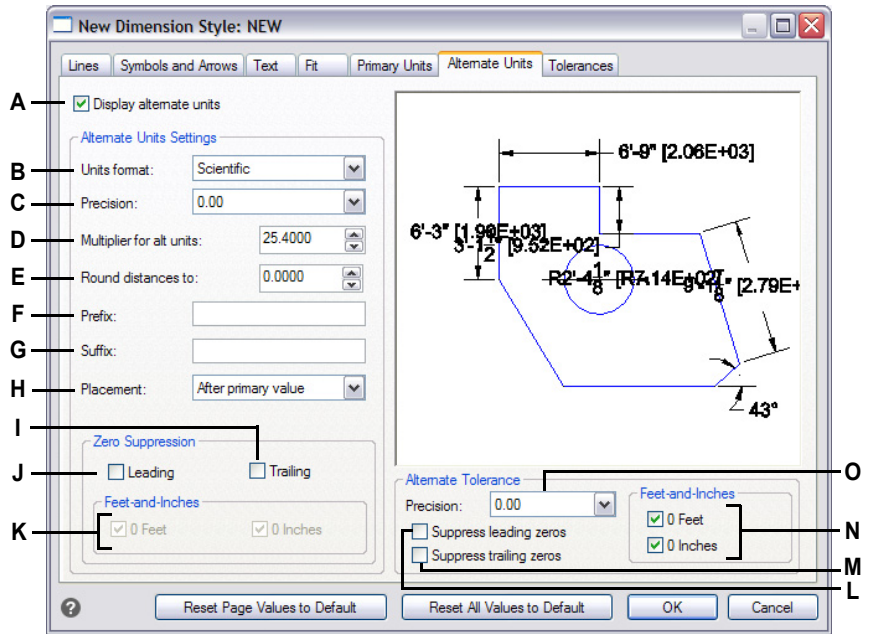
You can include alternate dimensions in addition to the primary dimension text. You can also determine the appearance and format of the alternate dimensions, including the scale factor applied to generate alternate dimensions. The image tile on the right side of the Dimension Styles dialog box shows the appearance of the dimensions based on the current dimension style settings.



Alternate dimension created using a scale factor of 25.4, with an appended suffix.

To control alternate dimension units

- 1 Do one of the following to choose Dimension Styles Manager ():
 - On the ribbon, choose Annotate > Dimension Styles Manager (in Dimensions).
 - On the menu, choose Dimensions > Dimension Styles Manager or choose Format > Dimension Styles Manager.
 - On the Dimensioning or Styles toolbar, click the Dimension Styles Manager tool.
 - Type *setdim* and then press Enter.
- 2 Click the Alternate Units tab.
- 3 Select Display Alternate Units.
- 4 Make your selections.
- 5 Click OK.



- A Select to include alternate units with dimension text.
- B Click to select the format for alternate dimension text.
- C Type or select the number of decimal places displayed in alternate dimension text.
- D Type or select the scale factor applied to measured dimensions to generate the alternate dimension text.
- E Type or select any rounding for alternate dimension text.
- F Type a prefix to be appended to alternate dimension text.
- G Type a suffix to be appended to alternate dimension text.
- H Select the placement of alternate dimension text.
- I Select to prevent the inclusion of trailing zeros for alternate dimension text.
- J Select to prevent the inclusion of leading zeros for alternate dimension text.
- K Select to prevent the inclusion of inches or feet in alternate dimension text when the corresponding number of inches or feet is zero.
- L Select to prevent the inclusion of leading zeros for tolerances included as part of alternate dimensions.
- M Select to prevent the inclusion of trailing zeros for tolerances included as part of alternate dimensions.
- N Select to prevent the inclusion of inches or feet for tolerances included as part of alternate dimensions when the corresponding number of inches or feet is zero.
- O Type or select the number of decimal places displayed in limits or tolerances included as part of alternate dimensions.















Adding geometric tolerances

Geometric tolerances indicate the maximum allowable variations in the geometry defined by a drawing. PTCAD draws geometric tolerances using a feature control frame, which is a rectangle divided into compartments.

Understanding geometric tolerances

Each feature control frame consists of at least two compartments. The first compartment contains a geometric tolerance symbol that indicates the geometric characteristic to which the tolerance is applied, such as location, orientation, or form. For example, a form tolerance may indicate the flatness or roundness of a surface. The geometric tolerance symbols and their characteristics are shown in the following table.

Geometric tolerance symbols

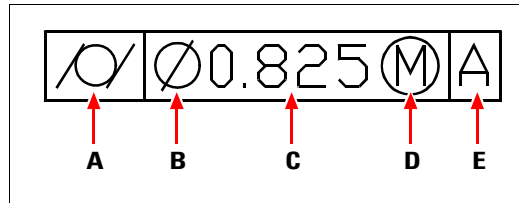
Symbol	Characteristic	Type
	Position	Location
	Concentricity or coaxiality	Location
	Symmetry	Location
	Parallelism	Orientation
	Perpendicularity	Orientation
	Angularity	Orientation
	Cylindricity	Form
	Flatness	Form
	Circularity or roundness	Form
	Straightness	Form
	Profile of a surface	Profile
	Profile of a line	Profile
	Circular runout	Runout
	Total runout	Runout

The second compartment contains the tolerance value. When appropriate, the tolerance value is preceded by a diameter symbol and followed by a material condition symbol. The material conditions apply to features that can vary in size. The material condition symbols and their meanings are shown in the following table.

Material conditions

Symbol	Definition
Ⓜ	At maximum material condition (MMC), a feature contains the maximum amount of material stated in the limits.
Ⓛ	At least material condition (LMC), a feature contains the minimum amount of material stated in the limits.
Ⓢ	Regardless of feature size (RFS) indicates that the feature can be any size within the stated limits.

The tolerance value can then be followed by primary, secondary, and tertiary datum reference letters, along with the material conditions of each datum. Datum reference letters are generally used as reference tolerances to one of up to three perpendicular planes from which a measurement is made, although datum reference letters can also indicate an exact point or axis.




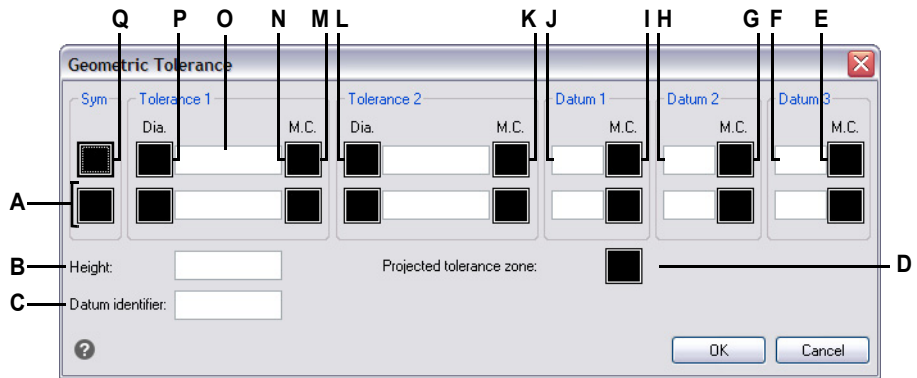
- A Geometric characteristic symbol.
- B Diameter symbol.
- C Tolerance value.
- D Material condition symbol.
- E Datum reference.

When two tolerances apply to the same geometry, you can also add a composite tolerance consisting of a primary tolerance value followed by a secondary tolerance value. To make a tolerance even more specific, it can also contain a projected tolerance consisting of a height value followed by a projected tolerance symbol. For example, you can use a projected tolerance to indicate the perpendicularity of an embedded part.

Adding a geometric tolerance

To add a geometric tolerance

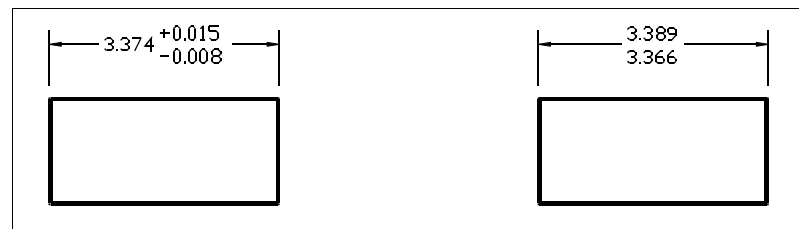
- 1 Do one of the following to choose Tolerance (- On the ribbon, choose Annotate > Tolerance (in Dimensions).
- On the menu, choose Dimensions > Tolerance.
- On the Dimensioning toolbar, click the Tolerance tool.
- Type *tolerance* and then press Enter.
- 2 To display the geometric tolerance symbols, on the first line, click the Sym box.
- 3 Click to select a geometric tolerance symbol.
- 4 Under Tolerance 1, click the Dia box to add a diameter symbol.
- 5 In the field, type the first tolerance value.
- 6 To display the material condition symbols, click the M.C. box.
- 7 Click to select a material condition.
- 8 Under Tolerance 2, repeat steps 4 through 7 to add a secondary tolerance value, if appropriate.
- 9 Under Datum 1, type the primary datum reference letter.
- 10 To display the material condition symbols, click the M.C. box.
- 11 Click to select a material condition.
- 12 Repeat steps 9 through 11 to add secondary and tertiary datum, if appropriate.
- 13 In the second row, repeat steps 2 through 12 to add composite tolerances, if appropriate.
- 14 In the Height box, type a projected tolerance zone height value, if appropriate.
- 15 To insert a projected tolerance zone symbol, click the Projected Tolerance Zone box, if appropriate.
- 16 Click OK.
- 17 In the drawing, specify the location of the feature frame.



- A Use this row to include composite tolerances.
- B Type a projected tolerance height value.
- C Type a datum identifier.
- D Click to include a projected tolerance symbol.
- E Click to select the material condition symbol for tertiary data.
- F Type the tertiary datum reference letter.
- G Click to select the material condition symbol for secondary datum.
- H Type the secondary datum reference letter.
- I Click to select the material condition symbol for primary datum.
- J Type the primary datum reference letter.
- K Click to select the material condition symbol for the second tolerance value.
- L Type the second tolerance value.
- M Click to include a diameter symbol for the second tolerance value.
- N Click to select the material condition symbol for the first tolerance value.
- O Type the first tolerance value.
- P Click to include a diameter symbol for the first tolerance value.
- Q Click to select a geometric tolerance symbol.

Controlling dimension tolerances

You can create dimensions as either tolerance or limits dimensions. When creating a tolerance dimension, you can control the upper and lower tolerance limits as well as the number of decimal places of the dimension text. The image tile on the right side of the Dimension Styles dialog box shows the appearance of tolerance and limits dimensions based on the current dimension style settings.




Tolerance dimension.

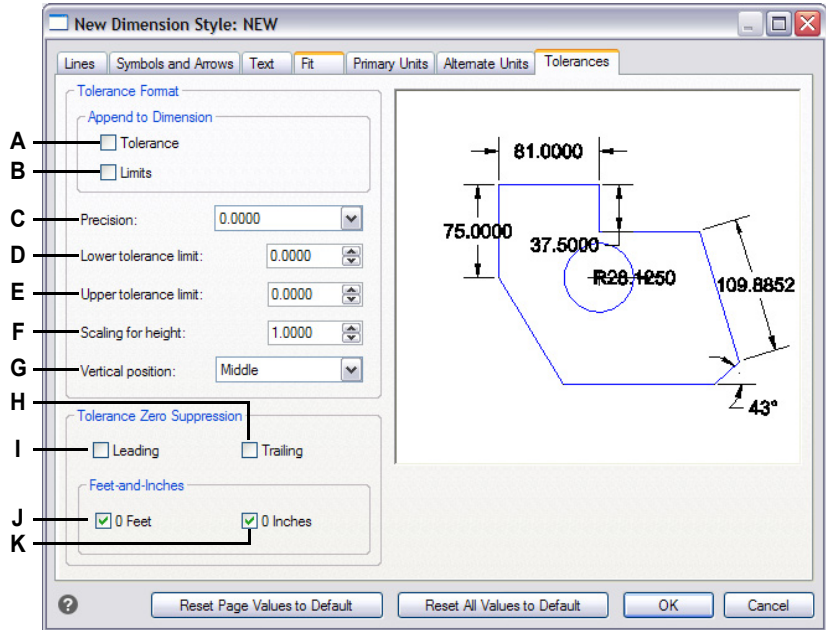
Limits dimension.

The display of zero tolerance values depends on the measurement units.

Metric drawings follow the Metric ISO Dimension Standards and display zero tolerance values without a plus or minus sign (0.0). For Imperial drawings, zero tolerance values display with a plus or minus sign (+0.0 or -0.0). To check the drawing measurement type, use the MEASUREMENT system variable.

To control dimension tolerances

- 1 Do one of the following to choose Dimension Styles Manager (- On the ribbon, choose Annotate > Dimension Styles Manager (in Dimensions).
- On the menu, choose Dimensions > Dimension Styles Manager or choose Format > Dimension Styles Manager.
- On the Dimensioning or Styles toolbar, click the Dimension Styles Manager tool.
- Type *setdim* and then press Enter.
- 2 Click the Tolerance tab.
- 3 Make your selections.
- 4 Click OK.



- A Select to insert a plus and minus tolerance value with dimension text.
- B Select to insert upper and lower tolerance limits with dimension text.
- C Type or select the number of decimal places displayed in limits or tolerance dimension text.
- D Type or select the minimum tolerance or lower limit value.
- E Type or select the maximum tolerance or upper limit value.
- F Type or select the scale factor applied to the height of limits or tolerance dimension text.
- G Select the vertical justification of limits or tolerance dimension text.
- H Select to prevent the inclusion of trailing zeros in limits or tolerance dimension text.
- I Select to prevent the inclusion of leading zeros in limits or tolerance dimension text.
- J Select to prevent the inclusion of inches or feet in limits or tolerance dimension text when the corresponding number of inches or feet is zero.

Working with other files in your drawings

Blocks, attributes, and external references provide mechanisms for managing entities in your drawings and for including additional information with the standard drawing entities. With blocks, you can combine numerous entities into a single entity, and then reuse it, inserting multiple copies. With attributes, you can associate text, such as part numbers or prices, with blocks, and then extract the text-attribute information to a separate file, such as a database, for further analysis. With external references, you can link separate reference drawing files to a drawing to combine information without adding the contents of the reference drawings to the current drawing. If you make changes to the referenced file, all references are updated automatically.

This section explains how to:

- Create, insert, and redefine blocks.
- Create, edit, and insert attributes.
- Extract attribute data to a separate file.
- Attach and work with external references and underlays.
- Link and update data between drawings and spreadsheets.
- Use images in your drawings.
- Use a geographic location in your drawings.

Topics in this chapter

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Working with blocks

Usually, blocks are several entities combined into one that you can insert into a drawing and manipulate as a single entity. A block can consist of visible entities such as lines, arcs, and circles, as well as visible or invisible data called attributes. Blocks are stored as part of the drawing file.

Understanding blocks

Blocks can help you better organize your work, quickly create and revise drawings, and reduce drawing file size. Using blocks, you can create a library of frequently used symbols. Then you can insert a symbol as a block rather than redraw the symbol from scratch.

After you create a block from multiple entities, you save it once, which also saves disk space. You insert only multiple references to a single block definition. You can change the block definition to quickly revise a drawing, and then update all instances of the block.

If you insert a block that contains entities originally drawn on layer 0 and assigned color and linetype `BYLAYER`, it is placed on the current layer and assumes the color and linetype of that layer. If you insert a block that contains entities originally drawn on other layers or with explicitly specified colors or linetypes, the block retains the original settings.

If you insert a block that contains entities originally assigned color and linetype `BYBLOCK`, and the block itself has the color and linetype `BYLAYER`, those entities adopt the color and linetype of the layer onto which they are inserted. If the block is assigned an explicit color or linetype, such as red or dashed, those entities adopt those qualities.

A procedure called nesting occurs when you include other blocks in a new block that you are creating. Nesting is useful when you want to combine and include small components, such as nuts and bolts, into a larger assembly and you need to insert multiple instances of that assembly into an even larger drawing.

Some blocks are called dynamic and have associated parameters and actions. You can't create dynamic blocks using `PTCAD`, but you can insert any dynamic block that was created using a different CAD application.




The quickest way to create and save blocks is using the Block Editor. It allows you to draw and save the block in-place, directly in the drawing area. You can also create blocks using individual block commands. A third way of creating blocks is using `PTCAD`'s Explorer. For details about creating blocks using the Explorer, see "Working with blocks" on page 345.

Creating and saving a block using the Block Editor

When you create a block using the Block Editor, you specify its name and then draw the entities that compose the block directly in the drawing area. While creating the block in-place, you can save the block with its existing name or a new name.

The insertion point of the created block is automatically defined relative to the UCS.

To create a block using the Block Editor

- 1 Do one of the following to choose Block Editor ():
 - On the ribbon, choose Home > Block Editor (in Block Definition) or choose Insert > Block Editor (in Block Definition).
 - Type *bedit* and then press Enter.
- 2 In the Edit Block Definition dialog box, enter a name and description for the new block.
- 3 Click OK.
- 4 Draw the entities that make up the block.
- 5 Do one of the following to choose Save Block ():
 - On the ribbon, choose Block Editor > Save Block.
 - On the floating toolbar (if the ribbon is not displayed), choose Save Block.
 - Type *bsave* and then press Enter.
- 6 Do one of the following to choose Close Block Editor ():
 - On the ribbon, choose Block Editor > Close Block Editor.
 - On the floating toolbar (if the ribbon is not displayed), choose Close Block Editor.
 - Type *bclose* and then press Enter.

The program adds the new block to the blocks list, with the name you entered for it.

If you often include blocks in your drawings, you can create a tool palette that contains the blocks you use most often. For details, see “Creating tool palettes and adding commands and blocks” on page 864.

An existing block instance can easily be used to create a new block.

Use the Rename Block command to create a new block by renaming a single block instance. Use the Change Block command to change the name of a single block instance and then open the Block Editor where you can modify it.

Creating and saving a block using individual commands





When you create a block using the Block command, you specify its name, its insertion point, and the entities that compose the block. The insertion point is the base point for the block and serves as the reference point when you later insert the block into a drawing.

The new block you create exists only in the current drawing, unless you save it separately using the Save Block to Disk command.

Creating a block for use within the current drawing

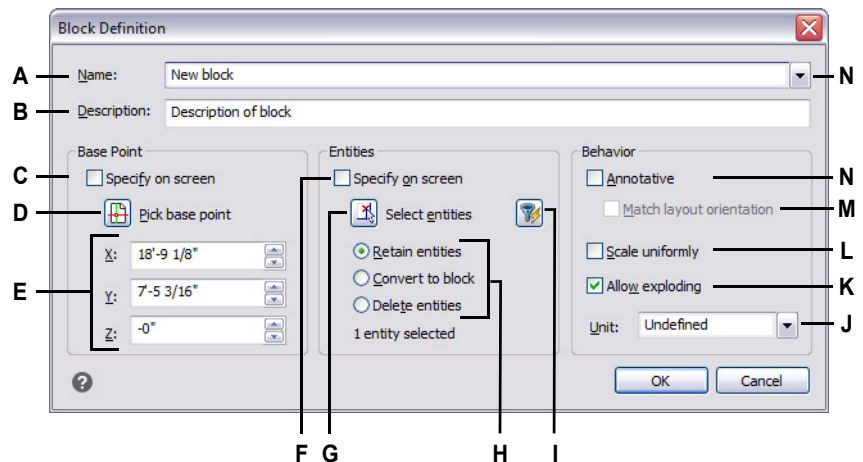
To create a block for use within a current drawing

Advanced experience level

- 1 Do one of the following to choose Create Block ():
 - On the ribbon, choose Home > Create Block (in Block) or choose Insert > Create Block (in Block Definition).
 - On the menu, choose Draw > Block > Create Block.
 - On the Tools toolbar, click the Create Block tool.
 - Type *block* and then press Enter.
- 2 In the Block definition dialog box, enter a name and description for the new block.
- 3 Specify the insertion point for the block by doing one of the following:
 - **Specify on Screen** Mark this check box to select the base point in the drawing after you click OK.
 - **Pick Base Point** Click () to temporarily close the dialog box immediately, select the base point in the drawing, then return to the dialog box. This option is available only if Specify on Screen is not marked.
 - **X, Y, and Z** Enter the x-, y-, and z-coordinates of the base point. This option is available only if Specify on Screen is not marked.
- 4 Select the entities to be combined into the block by doing one of the following:
 - **Specify on Screen** Mark this check box to select the entities in the drawing after you click OK.
 - **Select entities** Click () to temporarily close the dialog box immediately, select the entities in the drawing, then return to the dialog box. Or you can click () to select entities by type or value. This option is available only if Specify on Screen is not marked.
- 5 Select what to do with the entities after the block is created:
 - **Retain entities** Entities selected for the block remain in the drawing.
 - **Convert to block** Entities selected for the block are converted to the block, which remains in the drawing.
 - **Delete entities** Entities selected for the block are removed from the drawing.

- 6 Select any of the following options for the block:
- **Annotative** Determines whether the block is annotative by default. The display and printing of annotative blocks is affected by annotation scaling. If annotative by default, you can determine whether the block, when located in paper space, is oriented automatically according to the layout viewport.
 - **Scale uniformly** Mark this check box to retain the aspect ratio if the block is scaled. Annotative blocks must be scaled proportionately.
 - **Allow exploding** Mark this check box to allow the block to be exploded into separate entities.
 - **Unit** Defines the unit of the block, for example inches or millimeters.
- 7 Click OK.

The program adds a new block to the blocks list, with the name you entered for it.



- | | |
|--|---|
| <p>A Enter the name of the block.</p> <p>B Enter a description of the block.</p> <p>C Select to pick the base point in the drawing after you click OK.</p> <p>D Select to pick the base point now in the drawing.</p> <p>E Displays the base point coordinates or you can enter them manually.</p> <p>F Select to select entities to include in the block after you click OK.</p> <p>G Click to select entities now in the drawing.</p> <p>H Select whether to keep, convert, or delete entities after the block is created.</p> | <p>I Click to select entities by type or value.</p> <p>J Choose block units.</p> <p>K Select to allow the block to be exploded into entities.</p> <p>L Select to maintain aspect ratio if the block is scaled.</p> <p>M Select to make the block, when located in paper space, oriented according to the layout viewport (available only if Annotative is marked).</p> <p>N Mark to make the block annotative by default.</p> |
|--|---|

If you often include blocks in your drawings, you can create a tool palette that contains the blocks you use most often. For details, see “Creating tool palettes and adding commands and blocks” on page 864.

Some users frequently restore original entities after defining a block.

To restore the original entities to the drawing while retaining the new block, type `undelete` or `oops`. You might also want to add the `Undelete` command to a menu or toolbar by choosing `Tools > Customize`.


An existing block instance can easily be used to create a new block.

Use the `Rename Block` command to create a new block by renaming a single block instance. Use the `Change Block` command to change the name of a single block instance and then open the `Block Editor` where you can modify it.

Saving blocks


You can create a block as a separate drawing file that you can insert into other drawings. The `Save Block to Disk` command includes the most options for saving blocks, and the `Quick Block Save` command includes only the essential steps: selecting entities and naming the block.

To quickly save entities as a separate drawing file

- 1 Do one of the following to choose `Quick Block Save` ():
 - On the ribbon, choose `Insert > Quick Block Save` (in `Block Definition`).
 - On the menu, choose `Tools > Quick Block Save`.
 - On the `Tools` toolbar, click the `Quick Block Save` tool.
 - Type `quickwblock` and then press `Enter`.
- 2 Select the entities you want to save as a block, and then press `Enter`.
- 3 Enter a name for the destination drawing file.
- 4 Click `Save`.

To save a block as a separate drawing file

Advanced experience level


- 1 Do one of the following to choose `Save Block To Disk` ():
 - On the ribbon, choose `Insert > Save Block to Disk` (in `Block Definition`).
 - On the menu, choose `Tools > Save Block To Disk`.
 - On the `Tools` toolbar, click the `Save Block To Disk` tool.
 - Type `wblock` and then press `Enter`.
- 2 In `Source`, choose `Block`, then select the desired block from the list.
- 3 In `File Name and Path`, type the name and path of the destination drawing file you want to create, or click `[...]` to browse for it.
- 4 In `Insert Units`, select the units used for inserting the separate drawing file.
- 5 Click `OK`.

The program assigns the 0,0,0 coordinate as the insertion base point.

You can change the base point by opening the drawing and redefining the block.

To save the current drawing as a separate drawing file

Advanced experience level





- 1 Do one of the following to choose Save Block to Disk (- On the ribbon, choose Insert > Save Block to Disk (in Block Definition).
- On the menu, choose Tools > Save Block To Disk.
- On the Tools toolbar, click the Save Block To Disk tool.
- Type *wblock* and then press Enter.
- 2 In Source, choose Entire Drawing.
- 3 In File Name and Path, type the name and path of the destination drawing file you want to create, or click [...] to browse for it.
- 4 In Insert Units, select the units used for inserting the separate drawing file.
- 5 Click OK.

The program assigns the 0,0,0 coordinate as the insertion base point.

You can change the base point by opening the drawing and redefining the block.

To select entities and save them as a separate drawing file

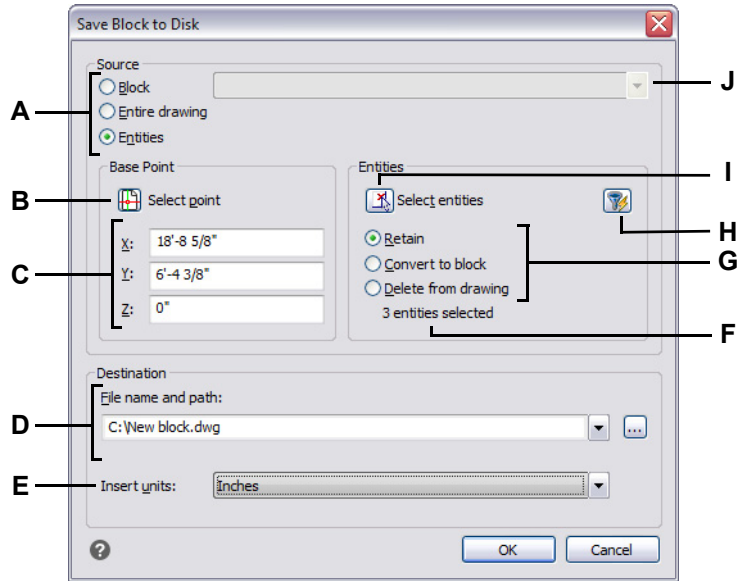
Advanced experience level

- 1 Do one of the following to choose Save Block to Disk (- On the ribbon, choose Insert > Save Block to Disk (in Block Definition).
- On the menu, choose Tools > Save Block To Disk.
- On the Tools toolbar, click the Save Block To Disk tool.
- Type *wblock* and then press Enter.
- 2 In Source, choose Entities.
- 3 In Base Point, enter the x,y,z coordinates of the base point to save with the separate drawing file, or click () to select it directly in the current drawing.
- 4 Click () and then select the source entities directly in the current drawing. Or you can click () to select entities by type or value.
- 5 Choose what action to take with selected entities in the current drawing after the separate drawing file is created:
 - **Retain** Keeps the selected source entities in the current drawing.
 - **Convert to block** Converts the selected source entities to a block in the current drawing.
 - **Delete from drawing** Deletes the selected source entities from the current drawing.

If no entities are selected, a separate drawing file is not created.

- 6 In File Name and Path, type the name and path of the destination drawing file you want to create, or click [...] to browse for it.
- 7 In Insert Units, select the units used for inserting the separate drawing file.
- 8 Click OK.

If you often include blocks in your drawings, you can create a tool palette that contains the blocks you use most often. For details, see “Creating tool palettes and adding commands and blocks” on page 864.



- A** Select what content to save to a separate drawing file. If Entities is selected, the Base Point and Entities options are available.
- B** Click to select in the current drawing the x,y,z coordinates of the base point to save with the separate drawing file.
- C** Enter the x,y,z coordinates of the base point to save with the separate drawing file.
- D** Enter the path and filename of the new drawing file, or click [...] to browse to it.
- E** Select the insertion units for the new drawing file.
- F** Displays the number of entities currently selected to save to a separate drawing file. If no entities are selected, a separate drawing file is not created.
- G** Select what action to take with selected entities in the current drawing after the separate drawing file is created.
- H** Click to select entities by type or value.
- I** Click to select in the current drawing the entities to save to a separate drawing file.
- J** Select the block to save to a separate drawing file. Available only if Block is selected as the source.

Inserting blocks

You can insert blocks and other drawings into the current drawing. When you insert a block, it is treated as a single entity. When you insert a drawing, it is added to the current drawing as a block. You can then insert multiple instances of the block without reloading the original drawing file. If you change the original drawing file, those changes have no effect on the current drawing unless you redefine the block by reinserting the changed drawing.


You can insert dynamic blocks that were created in a different CAD program. When inserting a dynamic block, press Ctrl to cycle through the block insertion points defined by its dynamic parameters.

When you insert a block or drawing, you must specify the insertion point, scale, and rotation angle. The block's insertion point is the reference point specified when you created the block. When you insert a drawing as a block, the program takes the specified insertion point as the block insertion point. You can change the insertion point, however, by first opening the original drawing and redefining the block.

Block settings can also be set before block insertion.


You can specify the insertion point, scale factors, and rotation angle before inserting a block by unmarking Specify on Screen for any of these options in the Insert Block dialog box, and then specifying the appropriate details. You can also control whether a block is exploded into its original component entities immediately after insertion by marking Explode.

To insert a block

- 1 Do one of the following to choose Insert Block (- On the ribbon, choose Home > Insert Block (in Block) or choose Insert > Insert Block (in Block).
- On the menu, choose Insert > Block.
- On the Draw toolbar, click the Block tool.
- Type *insert* or *ddinsert* and then press Enter.
- 2 Choose Block Name, then select the name of the block you want to insert.
- 3 Click Insert.
- 4 Specify the insertion point for the block.

If inserting a dynamic block, you can press Ctrl while left-clicking the mouse to cycle through block insertion points defined by its dynamic parameters.
- 5 Specify the x, y, and z scale factors and the rotation angle, or press Enter to accept the default values.

To insert an entire drawing into the current drawing

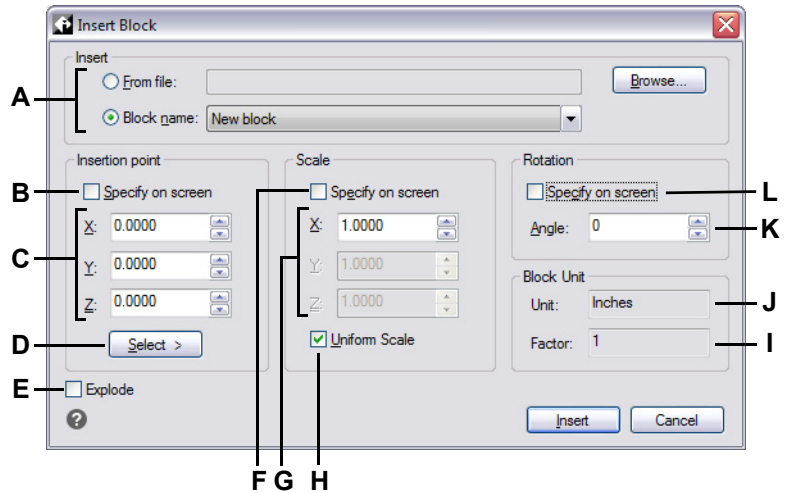
- 1 Do one of the following to choose Insert Block (- On the ribbon, choose Home > Insert Block (in Block) or choose Insert > Insert Block (in Block).
- On the menu, choose Insert > Block.
- On the Draw toolbar, click the Block tool.
- Type *insert* or *ddinsert* and then press Enter.
- 2 Choose From File, then type or click Browse to specify the path and the drawing file name.
- 3 Click Insert.
- 4 Specify the insertion point for the block.
- 5 Specify the x, y, and z scale factors and the rotation angle, or press Enter to accept the default values.

The scale can easily be changed after insertion.

To change the scale of a block after insertion, choose Express Tools > Blocks > Block Scale. You can also change the position, scale, and rotation using the Properties pane.

You can also insert drawings while browsing files on your computer.

If the DRAGOPEN system variable is set to 0, you can drag a .dwg file to the drawing area in PTCAD to insert it as a block. If DRAGOPEN is set to 1 (the default), the drawing opens in PTCAD.



- A** Choose to insert a block from a file and type or click Browse to specify the path and file name; or choose to insert a block from the current drawing and select its name from the list.
- B** Click to position the block when inserting and optionally insert multiple block instances.
- C** Specify the x-, y-, and z-coordinates of the insertion point (if Specify on screen is unmarked for insertion point).
- D** Click to pick an insertion point directly in the drawing area and return to the dialog box to specify additional options.
- E** Click to explode the block on insertion.
- F** Click to scale the block when inserting and optionally insert multiple block instances.
- G** Specify the scale factor for the x, y, and z directions (if Specify on screen is unmarked for scale).
- H** Mark to scale the block uniformly in the x, y, and z directions.
- I** Displays the scale factor defined for the block.
- J** Displays the insertion units defined for the block.
- K** Enter the rotation angle for the block (available if Specify on screen is unmarked for rotation).
- L** Click to rotate the block when inserting and optionally insert multiple block instances.

Modifying blocks


There are several ways you can modify a block. After you modify it, all blocks in the drawing are updated automatically.

Editing blocks using the Block Editor

You can redefine all instances of a block within the current drawing. To redefine a block that was created in the current drawing, you create a new block using the same name. You can update all the blocks in the current drawing by redefining the block. If the block was inserted from a separate drawing file that was subsequently updated, reinsert that block to update all other instances in the current drawing.


To redefine a block in the current drawing



Advanced experience level

- 1 Do one of the following to choose Block Editor ():
 - On the ribbon, choose Home > Block Editor (in Block Definition) or choose Insert > Block Editor (in Block Definition).
 - Type *bedit* and then press Enter.

Use a shortcut.

*Double-click a block to open it in the Block Editor. Note that the **BLOCKEDITLOCK** system variable controls the double-click action for blocks: if set to 0, the Block Editor opens; if set to 1, the Properties pane opens.*

- 2 In the Edit Block Definition dialog box, select the block you want to edit.
- 3 Click OK.
- 4 In the drawing area, make changes to the block.
- 5 Do one of the following to choose Save Block ():
 - On the ribbon, choose Block Editor > Save Block.
 - On the floating toolbar (if the ribbon is not displayed), choose Save Block.
 - Type *bsave* and then press Enter.

You can also choose Save Block As () to save the block with a new name.
- 6 Do one of the following to choose Close Block Editor ():
 - On the ribbon, choose Block Editor > Close Block Editor.
 - On the floating toolbar (if the ribbon is not displayed), choose Close Block Editor.
 - Type *bclose* and then press Enter.

The program updates all instances of the block.


You can update all instances of a block inserted from a separate drawing by reinserting the drawing.

Redefining blocks

You can redefine all instances of a block within the current drawing. To redefine a block that was created in the current drawing, you create a new block using the same name. You can update all the blocks in the current drawing by redefining the block. If the block was inserted from a separate drawing file that was subsequently updated, reinsert that block to update all other instances in the current drawing.

To redefine a block in the current drawing

Advanced experience level

- 1 Do one of the following to choose Create Block (- On the ribbon, choose Home > Create Block (in Block) or choose Insert > Create Block (in Block Definition).
- On the menu, choose Draw > Block > Create Block.
- On the Tools toolbar, click the Create Block tool.
- Type *block* and then press Enter.
- 2 In Name, select the name of the block you want to redefine from the list.
- 3 Specify the insertion point for the block.
- 4 Select the entities for the block.
- 5 Make selections about the behavior.
- 6 Click OK.
- 7 When prompted, choose Yes to redefine the block.
The block is redefined, and all instances of the block in the drawing are updated.
- 8 To restore the original entities to the drawing while retaining the new block, type undelete or oops.

You can update all instances of a block inserted from a separate drawing by reinserting the drawing.

Editing blocks in-place

After a block is inserted in a drawing, it can be edited directly in PTCAD, and the source block in the drawing (not on disk) and all references to the block are updated automatically. Editing in-place is an easy way to make changes to a block without having to locate and load it.

To edit blocks in-place

- 1 Do one of the following:
 - On the ribbon, choose Insert > Edit Reference in Place (in Reference).
 - On the menu, choose Tools > Edit Block or X-Ref In-Place > Edit In-Place.
 - Type *refedit* and then press Enter.
- 2 At the prompt, select the block you want to edit.
The Reference Edit dialog box displays.
- 3 In Reference Name, select the block you want to edit.
- 4 Select any of the following options:
 - **Automatically Select All Nested Entities** Select to automatically include all nested entities for editing in-place.
 - **Prompt to Select Nested Entities** Select to enable editing of individual nested entities.
- 5 Select the Settings tab and select from the following options:
 - **Create Unique Layer, Style and Block Names** Select to create unique names for layers, styles, and blocks that you change. A prefix is added to the original name of a changed layer, style, or block. Names of unchanged layers, styles, and blocks do not change.
 - **Display Attribute Definitions for Editing** Select to hide attributes and display attribute definitions while editing. After saving, changed attribute definitions affect new block insertions only. Existing blocks are not affected.
- 6 Click OK.
- 7 Make changes to the contents of the block.
- 8 To add an entity from the drawing to the block, select the entity and do one of the following:
 - Choose Tools > Edit Block or X-Ref In-Place > Add to Working Set.
 - Type *refset*, press Enter, then choose Add.
- 9 To remove an entity from the block, select the entity and do one of the following:
 - Choose Tools > Edit Block or X-Ref In-Place > Remove from Working Set.
 - Type *refset*, press Enter, then choose Remove.


- 10 When you're finished editing the block, do one of the following:
 - Choose Tools > Edit Block or X-Ref In-Place > Close Reference.
 - Type *refclose* and then press Enter.
- 11 Choose Save to save changes or Discard to cancel changes.
If changes are saved, all instances of the block are updated in the current drawing.

Exploding blocks

You can explode an inserted block to its original component entities. When you explode a block, only that single instance of the block is affected. The original block definition remains in the drawing, and you can still insert additional copies of the original block. If you explode a block that contains attributes, the attributes are lost, but the original attribute definitions remain.

Exploding dissociates component entities to their next simplest level of complexity; blocks or polylines in a block become blocks or polylines again


To explode a block

- 1 Do one of the following to choose Explode ():
 - On the ribbon, choose Home > Explode (in Modify) or choose Edit > Explode (in Modify).
 - On the menu, choose Modify > Explode.
 - On the Modify toolbar, click the Explode tool.
 - Type *explode* and then press Enter.
- 2 Select the block.
- 3 Press Enter.

Replacing blocks with other blocks

Use the Replace Block command to replace each occurrence of a block with a different block. Select the block to replace and the replacement block from a list of all blocks available in the current drawing, and then choose whether to purge the unused block definition from the drawing.

To replace a block


- 1 Do one of the following to choose Replace Block (- On the ribbon, choose Express Tools > Replace Block (in Blocks).
- On the menu, choose Express Tools > Blocks > Replace Block.
- Type *blockreplace* and then press Enter.
- 2 In Select Original Block, select the block that you want to replace. Or click Pick to select the block directly in the drawing.
- 3 In Select Replacement Block, select the block that you want to use as a replacement block. Or click Pick to select it directly in the drawing.
- 4 Click OK.
- 5 To remove the unreferenced block definition from the drawing, choose Yes. To keep the block definition in the drawing, choose No.

Converting blocks to external references


There are two ways you can convert and replace blocks with external references in the current drawing:

- Replace all block occurrences with a new external reference that is created from the block.
- Replace all block occurrences with an existing external reference.

To convert and replace a block with a new external reference

- 1 Do one of the following to choose Convert Block to Xref (- On the ribbon, choose Express Tools > Convert Block to Xref (in Blocks).
- On the menu, choose Express Tools > Blocks > Convert Block to Xref.
- Type *blocktoxref* and then press Enter.
- 2 In Block to Convert, select the block that you want to convert, or click Pick to select the block directly in the drawing area.
- 3 Choose Convert Block to Xref.
- 4 Click OK.
- 5 In the Save Block to File dialog box, enter the file name for the external reference, then click Save.

To convert and replace a block with an existing external reference

- 1 Do one of the following to choose Convert Block to Xref (- On the ribbon, choose Express Tools > Convert Block to Xref (in Blocks).
- On the menu, choose Express Tools > Blocks > Convert Block to Xref.
- Type *blocktoxref* and then press Enter.
- 2 In Block to Convert, select the block that you want to convert, or click Pick to select the block directly in the drawing area.
- 3 Choose Replace Block with Existing Xref.
- 4 Click OK.
- 5 In the Select Xref File dialog box, select the drawing file for replacing the selected block, then click Open.
- 6 To remove the unreferenced block definition from the drawing, choose Yes. To keep the block definition in the drawing, choose No.

Working with attributes

An attribute is a particular entity that you can save as part of a block definition. Attributes consist of text-based data. You can use attributes to track such things as part numbers and prices. Attributes have either fixed or variable values. When you insert a block containing attributes, the program adds the fixed values to the drawing along with the block, and you are prompted to supply any variable values.

After you insert blocks containing attributes, you can extract the attribute information to a separate file and then use that information in a spreadsheet or database to produce a parts list or bill of materials. You can also use attribute information to track the number of times a particular block is inserted into a drawing. And if changes are made to block attributes, you can easily update block instances in the drawing with the changes.


Attributes can be visible or hidden. Hidden attributes are neither displayed nor printed, but the information is still stored in the drawing and written to a file when you extract it.

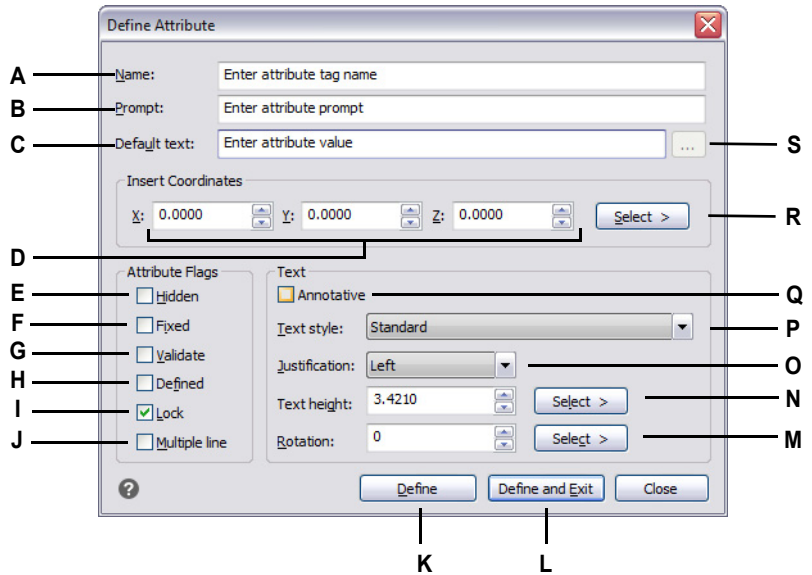
Defining attributes

You add an attribute to a drawing by first defining it and then saving it as part of a block definition. To define an attribute, you specify the characteristics of the attribute, including its name, prompt, and default value; the location and text formatting; and optional modes (hidden, fixed, validate, predefined, and locked).

To define an attribute

Advanced experience level

- 1 Do one of the following to choose Define Attributes (- On the ribbon, choose Insert > Define Attributes (in Block Definition).
- On the menu, choose Draw > Block > Define Attributes.
- On the Tools toolbar, click the Define Attributes tool.
- Type `ddattdef` and then press Enter.
- 2 In the Define Attribute dialog box, type the name, prompt, and default value.
- 3 Under Insert Coordinates, specify the location of the attribute, or click Select to select a point in the drawing.
- 4 Under Attribute Flags, select the optional attribute modes.
- 5 Under Text, specify the text characteristics.
- 6 To add the attribute to the drawing, do one of the following:
 - Click Define to add the attribute and keep the dialog box active so you can define another attribute.
 - Click Define And Exit to add the attribute and end the command.




- A** Type the name you want to assign to the attribute.
- B** Enter the identifying prompt information displayed when you insert a block containing the attribute.
- C** Enter the default or constant value. For variable attributes, the default value is replaced by the actual value when you later insert a block containing the attribute.
- D** Specify the x-, y-, and z-coordinates for the attribute insertion point.
- E** Select to create a hidden attribute.
- F** Select to create a fixed-value attribute.
- G** Select to create an attribute whose value must be validated when you later insert a block containing the attribute.
- H** Select to create an attribute whose value is defined and not requested when you later insert a block containing the attribute, but that you can edit after the block is inserted.
- I** Select to create an attribute whose position is locked.
- J** Select to create an attribute whose default text contains multiple lines of text.
- K** Click to add the attribute and keep the dialog box active so you can define another attribute.
- L** Click to add the attribute and end the command.
- M** Specify the text rotation angle, or click to specify the rotation angle by selecting two points in the drawing.
- N** Specify the text height, or click to specify the height by selecting two points in the drawing.
- O** Choose the text justification.
- P** Choose the text style from those styles already defined in the drawing.
- Q** Select to create an attribute that is annotative by default.
- R** Click to specify the attribute insertion point by selecting a point in the drawing.
- S** Click to enter multiple lines of default text. Available only if Multiple line is selected.

Editing attribute definitions

You can edit an attribute definition before you associate it with a block and before it is saved as part of a block definition.

To edit an attribute definition

Advanced experience level

- 1 Select the attribute definition text to edit.
- 2 Do one of the following to choose Properties ():
 - On the ribbon, choose View > Properties (in Panes).
 - On the menu, choose View > Display > Properties or Modify > Properties.
 - Right-click the attribute definition text, then choose Properties.
 - Type *entprop* and then press Enter.
- 3 Modify the properties, including name, prompt, default value, and other attribute-specific properties.

Attaching attributes to blocks


You can attach attributes to a block after you define it and select it as one of the entities to include. Include the attributes when the program prompts you for the entities to include in the selection set for a block. After the attribute is incorporated into a block, the program prompts you each time you insert the block, so you can specify different values for the attributes each time you insert it into a new drawing.

Editing attributes attached to blocks

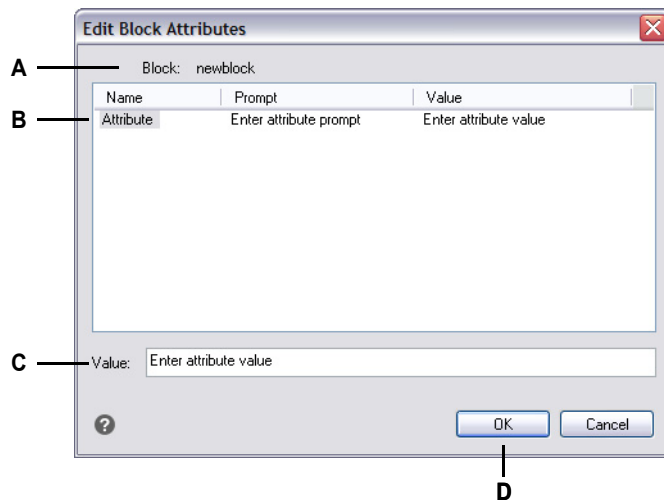
You can edit the attribute values of a block that has been inserted into a drawing.

To edit an attribute attached to a block

Advanced experience level

- 1 Do one of the following to choose Edit Attributes ():
 - On the ribbon, choose Insert > Edit Attributes (in Block Definition).
 - On the menu, choose Modify > Entities > Attributes > Edit Attributes.
 - On the Tools toolbar, click the Edit Attributes tool.
 - Type *ddatte* and then press Enter.
- 2 Select the block to edit.

The Edit Block Attributes dialog box displays all the attributes attached to the block you select.
- 3 Edit the attribute values as necessary.
- 4 Click OK.




A Identifies the name of the block.

B Click to select attribute.

C Type the new value for the selected attribute.

D Click to update the attribute values and exit.

To edit an attribute associated with multiple blocks

- 1 Do one of the following to choose Global Attribute Edit (- On the ribbon, choose Express Tools > Global Edit Attributes (in Blocks).
- On the menu, choose Express Tools > Blocks > Global Edit Attributes.
- Type *gatte* and then press Enter.
- 2 Do one of the following:
 - Select the attribute you want to edit.
 - Select the block you want to edit, then select or type the name of the attribute you want to edit.
 - Choose Block name, then enter the name of the block that is associated with the attribute you want to edit.
- 3 Enter a new attribute value.
- 4 Choose Yes to change the values of all attributes with the specified name, or choose No to select them in the drawing one by one.

Extracting attribute information

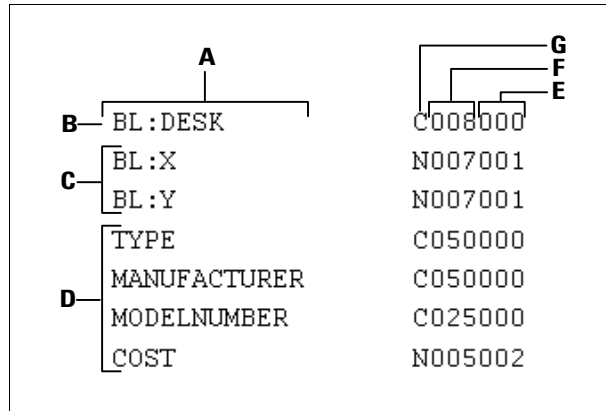
You can extract attribute information from a drawing and save it to a separate text file for use with a database program. You can save the file in any of the following formats:

- **Comma Delimited Format (CDF)** Contains one line for each instance of a block, with individual attribute fields separated by commas. Character string fields are enclosed with single quotation marks. You must specify a template file when extracting to a CDF file.
- **Space Delimited Format (SDF)** Contains one line for each instance of a block. Each attribute field has a fixed length; there are no separators or character string delimiters. You must specify a template file when extracting to a SDF file.
- **Drawing Exchange Format (DXF)** Creates a subset of a standard DXF file (a *.dxx file) containing all the information about each block, including the insertion points, rotation angles, and attribute values. No template file is required.

Before extracting attributes to a CDF or SDF file, you must create a template file. The template file is an ASCII text file that specifies the attribute data fields to be written in the extract file. Each line of the template file specifies one attribute field.

PTCAD recognizes 15 different fields, which contain elements such as the block name, the x-, y-, and z-coordinates of its insertion point, the layer on which it is inserted, and so on. You can include any of these fields. The template file must include at least one attribute name.

Each line in the template file must start with the field name. Block name and insertion-point values must begin with *BL:*. The next nonblank character must be either a *C* (indicating a character string field) or an *N* (indicating a numeric field). This character is then followed by three digits indicating the width of the field (in characters). The final three digits indicate the number of decimal places (for numeric fields). In the case of character fields, the last three digits must be zeros (000). A typical template file is similar to the one shown here:




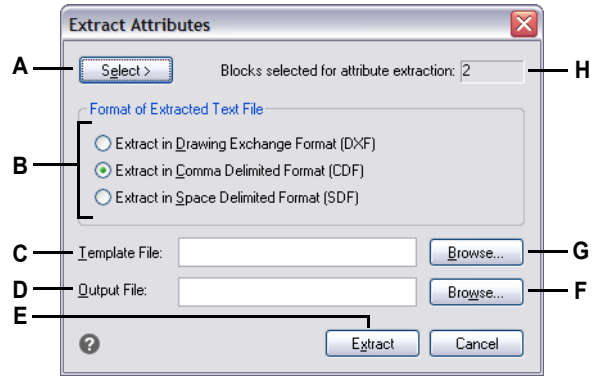
- A** Field name.
- B** Block name. Block names must begin with *BL:*.
- C** Insertion point. Insertion-point values must begin with *BL:*.
- D** Attribute tags.
- E** Number of decimal places for numeric fields or 000 for character fields.
- F** Field width for character or numeric fields.
- G** C for character fields; N for numeric fields.

To create a template file

- 1 Create a template file using any ASCII text editor (such as Microsoft® Notepad or Microsoft® WordPad) or a word-processing program such as Microsoft® Word.
- 2 Include the necessary fields in the template file.
- 3 Save the template file in ASCII text format.

To extract attribute information

- 1 Do one of the following to choose Extract Attributes ():
 - On the ribbon, choose Insert > Extract Attributes (in Block Definition).
 - On the menu, choose Tools > Extract Attributes.
 - On the Tools toolbar, click the Extract Attributes tool.
 - Type *dattext* and then press Enter.
- 2 Click Select, specify the entities from which to extract attributes, and then press Enter.
- 3 Specify the format of the extracted file.
- 4 For CDF and SDF formats, specify the template file.
- 5 Specify the extract output file.
- 6 Click Extract.



- A** Click to select blocks in the drawing containing attributes you want to extract.
- B** Click to specify the format of the extracted file.
- C** Specify the template file for CDF and SDF extracts.
- D** Specify the extract output file.
- E** Click to extract attributes.
- F** Click to specify the output file using a file dialog box.
- G** Click to specify the template file using a file dialog box.
- H** Indicates the number of blocks with attributes selected for extraction.

Synchronizing attributes and blocks

If changes are made to block attributes with the Create Block or Block Editor command, you can automatically apply those changes to block instances using the Synchronize Attributes command.

This command doesn't affect values assigned to attributes.

To synchronize a block with updated attributes

- 1 Do one of the following to choose Synchronize Attributes (

Working with external references

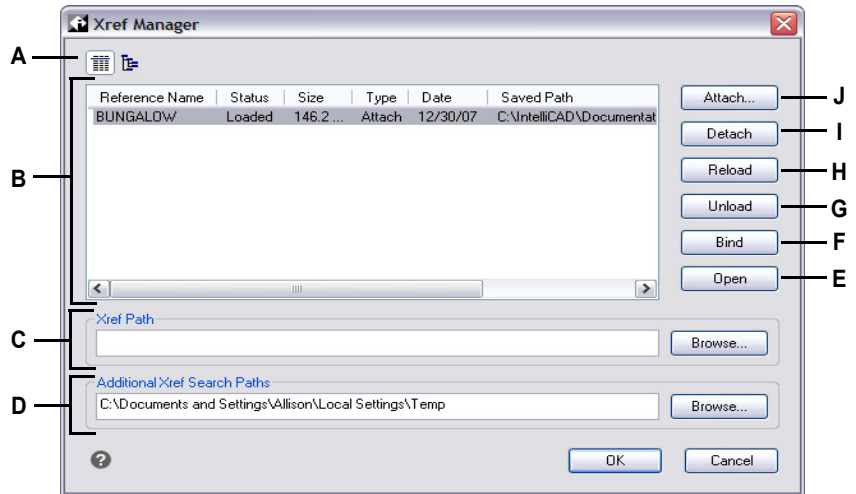
You can link entire drawings to the current drawing as external references. Unlike inserting a drawing as a block, in which you add all the entities from the separate drawing into the current drawing, external references attach a pointer to the external file. The entities in the external reference appear in the current drawing, but the entities themselves are not added to the drawing. Thus, attaching an external reference does not significantly increase the size of the current drawing file.

Understanding external references

External references provide additional capabilities not available when you insert a drawing as a block. When you insert a drawing as a block, the entities are stored in the drawing. Any changes you make to the original drawing are not reflected in the drawing in which you inserted it. When you attach an external reference, however, any changes you make to the original drawing file are reflected in the drawings that reference it. These changes appear automatically each time you open the drawing containing the external reference. If you know that the original drawing was modified, you can reload the external reference anytime you're working on the drawing.

External references are useful for assembling master drawings from component drawings. Use external references to coordinate your work with others in a group. External references help reduce drawing file size and ensure that you are always working with the most recent version of a drawing. However, if you send or receive drawings that contain external references, it is important to include with the master drawing all of the external references attached to it. When you open a drawing that contains external references, the source external reference files must be accessible for the external references to display in the drawing.

The Xref Manager helps you easily attach and work with external references.



- A Click to display a detailed list or hierarchical tree.
- B Select an external reference to modify its attachment.
- C Type or click Browse to specify the external reference location.
- D Type or click Browse to specify other search directories where external references may be located.
- E Click to open the source drawing for the external reference.
- F Click to make the external reference a permanent part of the drawing.
- G Click to remove the external reference, but keep elements and path information for easy reloading.
- H Click to update with changes from the external reference.
- I Click to completely remove the external reference.
- J Click to link a drawing.

Attaching external references

Attaching a separate drawing to the current one creates an external reference. The external reference appears in the drawing as a block definition, but the drawing entities are linked rather than added to the current drawing. If you modify the linked drawing, the current drawing that contains the external reference is updated automatically when you open it, or you can reload the external reference manually so it reflects the latest version of the external reference.


When you attach an external reference, its layers, linetypes, text styles, and other elements are not added to the current drawing. Rather, these elements are also linked from the referenced file.

There are two ways you can attach an external reference:

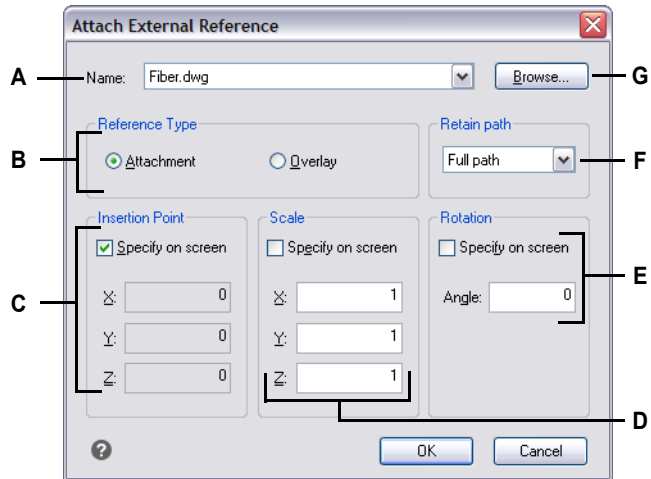
- An attachment is an inserted drawing that contains a link to the original file. Attachments can themselves contain other, nested reference files. When you attach an external reference, any nested references contained in the file also appear in the current drawing.
- An overlay is an inserted drawing that contains a link to the original file. Overlays allow you to lay a drawing on top of another drawing, similar to the way you work manually with transparencies. When a drawing that contains overlaid external references is itself attached or overlaid as an external reference in another drawing, the overlays do not appear as part of the external reference. Use overlaying when you want to see reference geometry in a drawing but you do not need to include that geometry in drawings that will be used by others (nested external references).

You can attach as many copies of an external reference file as you want. Each copy can have a different position, scale, and rotation angle.

To attach an external reference

- 1 Do one of the following to choose Xref Manager ():
 - On the ribbon, choose Insert > Xref Manager (in Reference).
 - On the menu, choose Insert > Xref Manager.
 - On the Tools toolbar, click the Xref Manager tool.
 - Type *xref* and then press Enter.
- 2 Click Attach.
- 3 Specify the drawing file to attach as an external reference, and then click Open.
- 4 In Reference Type, choose how you want to insert the drawing:
 - Attachment – inserts a copy of the drawing and includes any other drawings that are externally referenced within the referenced drawing.
 - Overlay – lays a copy of a drawing over your original drawing; it does not include any nested external references from the externally referenced drawing.
- 5 Make any additional selections.
- 6 Click OK.
- 7 If you marked Specify On-Screen for any items, follow the prompts to attach the external reference.

After an external file is attached, the Manage Xrefs icon displays in the status bar, notifying that the drawing contains a linked external reference file.






- A** Displays the external reference to attach, or select one from the list.
- B** Click Attachment to link a drawing, including any of its own external references. Click Overlay to link a drawing, omitting any of its own nested external references.
- C** Choose to specify the insertion point in the drawing, or enter x-, y-, and z-coordinates.
- D** Choose to specify the scale in the drawing, or enter x-, y-, and z-scale factors.
- E** Choose to specify the rotation angle in the drawing, or enter a rotation angle.
- F** Select to save the folder location of the referenced drawing. If not selected, the referenced drawing must be located in the same folder as the current drawing.
- G** Click to locate and select a different external reference.

Viewing the list of external references

You can view a list of the external references that are linked to the current drawing two different ways using the Xref Manager:

- **List View** displays the external references in a list, which allows you to sort the list of references by name, status, size, type, date, or saved path.
- **Tree View** displays a hierarchical representation of the external references and the relationships between them. The tree view shows the level of nesting relationships of the attached external references.


To view a list of external references

- 1 Do one of the following to choose Xref Manager ():
 - On the ribbon, choose Insert > Xref Manager (in Reference).
 - On the menu, choose Insert > Xref Manager.
 - On the Tools toolbar, click the Xref Manager tool.
 - Type *xref* and then press Enter.
 - On the status bar, click Manage Xrefs, which displays when external references are linked to the drawing.
- 2 Click List View () or Tree View ()

Opening external references

From the Xref Manager you can quickly open the source drawing for any external reference. This is especially helpful if you are working with nested external references, which you cannot bind or detach. From the Xref Manager, open the source drawing, make changes, and then save and close the source drawing. When the Xref Manager displays again, simply reload the external reference.

To open an external reference

- 1 Do one of the following to choose Xref Manager ():
 - On the ribbon, choose Insert > Xref Manager (in Reference).
 - On the menu, choose Insert > Xref Manager.
 - On the Tools toolbar, click the Xref Manager tool.
 - Type *xref* and then press Enter.
 - On the status bar, click Manage Xrefs, which displays when external references are linked to the drawing.
- 2 Select the external reference to open.
- 3 Click Open.

Use a shortcut.


Type `xopen` to open an external reference without using the Xref Manager. To see any changes that you make to the external reference while it is open, reload it.

Removing external references


Removing external references from the current drawing is easy with the Xref Manager. You can unload an external reference, which keeps some information about the external reference in the current drawing for easy reloading later, or you can detach the external reference entirely.

When you unload an external reference, you remove it from the current drawing. However, its elements, such as layers and linetypes, remain in the drawing and it is still listed in the Xref Manager. By detaching an external reference you remove it and all of its elements from the current drawing, and it is no longer listed in the Xref Manager.

To unload an external reference

- 1 Do one of the following to choose Xref Manager (- On the ribbon, choose Insert > Xref Manager (in Reference).
- On the menu, choose Insert > Xref Manager.
- On the Tools toolbar, click the Xref Manager tool.
- Type *xref* and then press Enter.
- On the status bar, click Manage Xrefs, which displays when external references are linked to the drawing.
- 2 Select the external reference to unload.
- 3 Click Unload.

To detach an external reference

- 1 Do one of the following to choose Xref Manager (- On the ribbon, choose Insert > Xref Manager (in Reference).
- On the menu, choose Insert > Xref Manager.
- On the Tools toolbar, click the Xref Manager tool.
- Type *xref* and then press Enter.
- On the status bar, click Manage Xrefs, which displays when external references are linked to the drawing.
- 2 Select the external reference to detach.
- 3 Click Detach.

Nested external references cannot be detached.

Only the external references that are attached directly to the current drawing can be detached.


Reloading external references

When you open or print a drawing, any external references in the drawing are updated automatically.

If a drawing is already open and a referenced drawing is modified, you can update the current drawing manually to display the latest version of the referenced drawing. The program checks for modified external references every five minutes.

You may also want to reload an external reference that has been unloaded temporarily.

To reload an external reference

- 1 Do one of the following to choose Xref Manager (- On the ribbon, choose Insert > Xref Manager (in Reference).
- On the menu, choose Insert > Xref Manager.
- On the Tools toolbar, click the Xref Manager tool.
- Type *xref* and then press Enter.
- On the status bar, click Manage Xrefs, which displays when external references are linked to the drawing.
- 2 Select the external reference to reload.
- 3 Click Reload.

Use a shortcut.


By default, when an external reference is modified outside of PTCAD, a notification appears in a balloon message and in the Event Log. Click Reload in the balloon message to reload the external reference. To turn off these notifications, set the XREFNOTIFY system variable to 0; to turn on notifications, set it to 2. To change how often the program checks for modified external references, set the number of minutes using the XNOTIFYTIME system variable.

Changing the path for external references

If the file associated with an external reference is moved to a different directory or renamed, the program displays a message indicating that it cannot load the external reference. You can re-establish the link to the file by doing any of the following:

- Change the path for the external reference.
- Specify additional directories for PTCAD to search. This is especially helpful if you have several external references that have moved to a new directory.


To change the path for a single external reference

- 1 Do one of the following to choose Xref Manager (- On the ribbon, choose Insert > Xref Manager (in Reference).
- On the menu, choose Insert > Xref Manager.
- On the Tools toolbar, click the Xref Manager tool.
- Type *xref* and then press Enter.
- On the status bar, click Manage Xrefs, which displays when external references are linked to the drawing.
- 2 Select the external reference whose path you want to change.
- 3 In Xref Path, do one of the following:
 - Enter a new filename or location.
 - Click Browse to locate and select the referenced drawing.
 reloads the specified external reference automatically.

External references cannot be recursive.

You cannot recursively reference a drawing from the same original drawing.

To change the search paths for all external references in the drawing

- 1 Do one of the following to choose Xref Manager (- On the ribbon, choose Insert > Xref Manager (in Reference).
- On the menu, choose Insert > Xref Manager.
- On the Tools toolbar, click the Xref Manager tool.
- Type *xref* and then press Enter.
- On the status bar, click Manage Xrefs, which displays when external references are linked to the drawing.
- 2 In Additional Xref Search Paths, do one of the following:
 - Enter a new directory and its path. Separate multiple paths with a semicolon, for example, c:\My Drawings;d:\My Drawings\Backup.
 - Click Browse to locate and select a directory.

PTCAD searches the specified directories; any found external references are reloaded automatically.


Binding external references to drawings

External references are not part of the drawing. Rather, they are links to an externally referenced file. To provide a copy of a drawing containing external references to someone else, you must also provide all the external reference files. In addition, the person receiving the drawings must either re-create the same paths you used when linking the external references or change the paths for the external references.

To provide a copy of a drawing that contains external references, it is often easier to first bind the external references to the drawing. Binding the external references makes them a permanent part of the drawing, which is similar to inserting a separate drawing as a block.

You can bind external references that are attached directly to the current drawing; you cannot bind nested external references.

To bind an existing external reference to a drawing

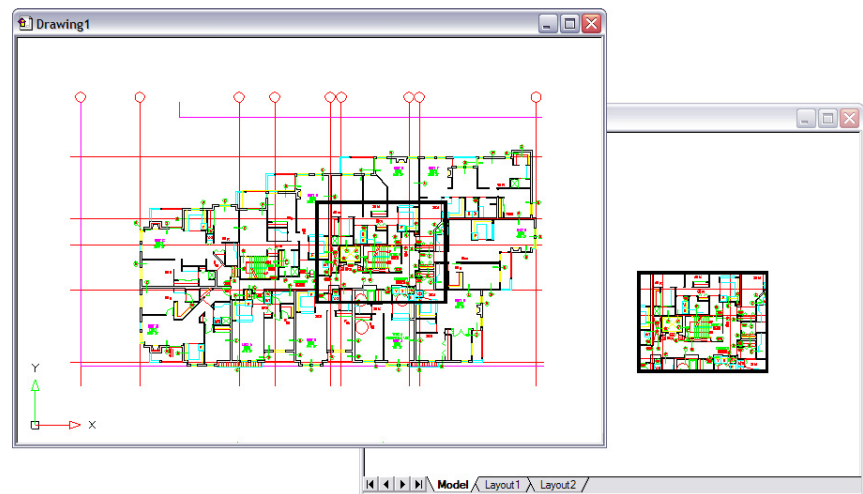
- 1 Do one of the following to choose Xref Manager (- On the ribbon, choose Insert > Xref Manager (in Reference).
- On the menu, choose Insert > Xref Manager.
- On the Tools toolbar, click the Xref Manager tool.
- Type *xref* and then press Enter.
- On the status bar, click Manage Xrefs, which displays when external references are linked to the drawing.
- 2 Select the external reference to bind.
- 3 Click Bind.
- 4 Choose one of the following:
 - **Bind** Binds the external reference and creates a unique name for each named entity, such as a layer or block, that is located in the external reference. For example, a layer named Electric in the external reference will be named Xref\$0\$Electric in the current drawing. If the current drawing already has a layer or block with the same name, the name is changed incrementally, for example, Xref\$1\$Electric.
 - **Insert** Binds the external reference, but does not change the names of any named entities in the external reference. For example, a layer named Electric in the external reference will have the same name, Electric, in the current drawing. If the current drawing has a layer or block with the same name, the named entity in the external reference takes on the properties of the named entity in the current drawing.
- 5 Click OK.

Clipping external references

When you attach a drawing as an external reference, all of the referenced drawing displays in the current drawing. However, after you attach an external reference, you can define a clipping boundary that determines which portion of the referenced drawing is visible or hidden.

You can edit, move, or copy clipped external references the same way you modify unclipped external references. The boundary moves with the reference. If an external reference contains nested clipped external references, they also appear clipped in the drawing.

In addition to clipping external references, you can also partially hide blocks using clipping boundaries.



Example of an external reference clipped using a clipping boundary. The clipping boundary is the rectangle in the top window.

Adding clipping boundaries

When you create a clipping boundary, it affects only the display of the referenced drawing; it does not affect the original referenced drawing or any referenced geometry. The portion of the external reference within the clipping boundary is visible and the remainder of the external reference becomes hidden.

To define a rectangular clipping boundary

- 1 Do one of the following:
 - On the ribbon, choose Insert > Clip Xref (in Reference).
 - On the menu, choose Modify > Xref Clip.
 - Type *xclip* and then press Enter.
- 2 Select the external references to clip. If desired, you can also select blocks.
- 3 Press Enter.
- 4 Press Enter to create a new clipping boundary.
- 5 If prompted, press Enter to delete any existing boundaries.
- 6 Choose Rectangular.
- 7 Define the first corner of the clipping rectangle.
- 8 Define the second corner of the clipping rectangle.
The selected external references are clipped by the rectangle.

Use the shortcut.

You can first select all external references, right-click the selection, and then select Xref Clip from the shortcut menu.

To define a clipping boundary using a polyline

- 1 Draw a polyline where you want to clip external references.
- 2 Do one of the following:
 - On the ribbon, choose Insert > Clip Xref (in Reference).
 - On the menu, choose Modify > Xref Clip.
 - Type *xclip* and then press Enter.
- 3 Select the external references to clip. If desired, you can also select blocks.
- 4 Press Enter.
- 5 Press Enter to create a new clipping boundary.
- 6 If prompted, press Enter to delete any existing boundaries.
- 7 Choose Select Polyline.
- 8 Select the polyline to use as clipping boundary.

Turning clipping boundaries on and off

You can turn xref clipping on or off. When a clipping boundary is turned off, the boundary does not display and the entire external reference is visible, provided that the geometry is on a layer that is on and thawed. When a clipping boundary is turned off, it still exists and can be turned on. However, deleting a clipping boundary is permanent.

To turn clipping boundaries on and off

- 1 Do one of the following:
 - On the ribbon, choose Insert > Clip Xref (in Reference).
 - On the menu, choose Modify > Xref Clip.
 - Type *xclip* and then press Enter.
- 2 Select the desired external references.
- 3 Press Enter.
- 4 To turn off clipping boundaries, choose Off. To turn on existing clipping boundaries, choose On.
- 5 Press Enter.

If you are turning off a clipping boundary, click the clipped portion of the external reference to view the previously hidden portion of the referenced drawing.

Use the XCLIPFRAME system variable.

When the XCLIPFRAME system variable is on (set to 1), you can select and print the clipping boundary frame.

Deleting clipping boundaries

If you no longer need a clipping boundary for an external reference, you can delete it.

To delete a clipping boundary

- 1 Do one of the following:
 - On the ribbon, choose Insert > Clip Xref (in Reference).
 - On the menu, choose Modify > Xref Clip.
 - Type *xclip* and then press Enter.
- 2 Select the desired external references.
- 3 Press Enter.
- 4 Choose Delete, and then press Enter.
- 5 Click the clipped portion of the external reference.

The previously hidden portion of the referenced drawing displays.

Editing external references in-place

After an external reference is inserted in a drawing, it can be edited directly in PTCAD, and the source drawing file is updated automatically. Editing in-place is an easy way to make changes to the source file without having to locate the file and load it.

To edit an external reference in-place

- 1 Do one of the following:
 - On the ribbon, choose Insert > Edit Reference in Place (in Reference).
 - On the menu, choose Tools > Edit Block or choose X-Ref In-Place > Edit In-Place.
 - Type *refedit* and then press Enter.
- 2 At the prompt, select the external reference you want to edit.
The Reference Edit dialog box displays.
- 3 In Reference Name, select the external reference you want to edit.
- 4 Select the Settings tab and select from the following options:
 - **Create Unique Layer, Style and Block Names** Select to create unique names for layers, styles, and blocks that you change. A prefix is added to the original name of a changed layer, style, or block. Names of unchanged layers, styles, and blocks do not change.
 - **Display Attribute Definitions for Editing** Select to hide attributes and display attribute definitions while editing. After saving, changed attribute definitions affect new insertions only.
- 5 Click OK.
- 6 Make changes to the contents of the external reference. Any new entities created during edit in-place are automatically added when the external reference is closed and saved.
- 7 To add an existing entity from the drawing to the external reference, select the entity and do one of the following:
 - Choose Tools > Edit Block or X-Ref In-Place > Add to Working Set.
 - Type *refset*, press Enter, then choose Add.

- 8 To remove an entity from the external reference, select the entity and do one of the following:
 - Choose Tools > Edit Block or X-Ref In-Place > Remove from Working Set.
 - Type *reset*, press Enter, then choose Remove.
- 9 When you're finished editing the external reference, do one of the following:
 - Choose Tools > Edit Block or X-Ref In-Place > Close Reference.
 - Type *refclose* and then press Enter.
- 10 Choose Save to save changes or Discard to cancel changes.
- 11 The external reference is updated and the current drawing displays the changes.

You can also type *xopen* to open an external reference directly.

To see any changes that you make to the external reference while it is open, reload it.

Attaching underlays created in other file formats

When you attach an underlay, a picture representation of the file's contents is inserted into the drawing. An underlay is similar to an image and different from an external reference in that it cannot be linked and automatically updated.

You can attach underlays using files that have the following formats:

- PDF format — Portable document format viewable using Adobe® Acrobat® Reader® and Adobe® Acrobat. The PDF format uses the .pdf file extension.
- Autodesk® DWF™ format — Autodesk Design Web Format™ (used with .dwf files) is used to distribute a drawing for others to view in a Web browser, review, and edit using free Autodesk software and tools. The DWF format uses the .dwf file extension.
- DGN format — Drawing files used with Bentley® Microstation®. The DGN format uses the .dgn file extension.
- PCG format — Point cloud files used by Autodesk® software and tools. The PCG format uses the .pcg file extension.
- RCP/RCS format — Point cloud files. The RCP/RCS format uses the .rcp or .rcs file extension.
- IFC format — BIM files in the Industry Foundation Classes format for building and construction data.
- RVT/RFA format — BIM files used by Autodesk Revit® for building and construction data.

Attaching a PDF underlay

Attaching a .pdf file is similar to attaching an image file.

To attach a PDF underlay

- 1 Use one of the following methods:
 - On the ribbon, choose Insert > PDF Underlay (in Data).
 - On the menu, choose Insert > PDF Underlay.
 - Type *pdfattach* and then press Enter.
- 2 Choose the .pdf file you want to attach.
- 3 Click Open.
- 4 Specify which page of the .pdf file to attach.
- 5 Select an insertion point.
- 6 Enter the scale in which to insert the .pdf file.
- 7 Enter the rotation to use for the insertion.

Attaching a DWF underlay

Attaching a .dwf file is similar to attaching an image file.

To attach a DWF underlay

- 1 Use one of the following methods:
 - On the ribbon, choose Insert > DWF Underlay (in Data).
 - On the menu, choose Insert > DWF Underlay.
 - Type *dwfattach* and then press Enter.
- 2 Choose the .dwf file you want to attach.
- 3 Click Open.
- 4 Select an insertion point.
- 5 Enter the scale in which to insert the .dwf file.
- 6 Enter the rotation to use for the insertion.

Attaching a DGN underlay

Attaching a .dgn file is similar to attaching an image file.

To attach a DGN underlay

- 1 Use one of the following methods:
 - On the ribbon, choose Insert > DGN Underlay (in Data).
 - On the menu, choose Insert > DGN Underlay.
 - Type *dgnattach* and then press Enter.
- 2 Choose the .dgn file you want to attach.
- 3 Click Open.
- 4 Select an insertion point.
- 5 Enter the scale in which to insert the .dgn file.
- 6 Enter the rotation to use for the insertion.

Attaching a point cloud underlay

A point cloud is a set of 3D points that represents the surface of an entity in three dimensions. Point cloud files are typically created by 3D-scanners.

Attaching a point cloud file (.pcg, .rcp, or .rcs file) is similar to attaching an image file.

To attach a point cloud underlay

- 1 Do one of the following to choose Point Cloud Underlay (☁):
 - On the ribbon, choose Insert > Point Cloud Underlay (in Data).
 - On the menu, choose Insert > Point Cloud Underlay.
 - Type *pointcloudattach* and then press Enter.
- 2 Choose the .pcg, .rcp, or .rcs file you want to attach.
- 3 Click Open.
- 4 In the Attach Point Cloud dialog box, select an insertion point.
- 5 Enter the scale in which to insert the point cloud file.
- 6 Enter the rotation to use for the insertion.
- 7 If you want to lock the point cloud so it cannot be moved or rotated after insertion, mark Lock Point Cloud.
- 8 If you want to zoom in to the point cloud automatically after insertion, mark Zoom to Point Cloud.
- 9 Click OK.

Attaching a BIM underlay

A Building Information Modeling (BIM) file is a drawing that typically contains a 3D model of a building that can be used for visualization, design, analysis, and to generate building construction documents. You can attach the following types of BIM files:

- **RVT format** — Drawing files used by Autodesk Revit® for building and construction data. You can attach .rvt files only using a 64-bit version of the program.
- **RFA format** — Family files used by Autodesk Revit® typically for storing building and construction elements. You can attach .rfa files only using a 64-bit version of the program.
- **IFC format** — Files in the IFC (Industry Foundation Classes) format used for building and construction data.

Attaching a BIM file is similar to attaching an image file.


After you attach a BIM underlay, note the following:

- **Layers** Even though BIM files do not contain layers, you can control visibility by enabling/disabling floors or categories using the pane that displays for the BIM file. Use the BIMPROP command to open the corresponding pane (and use BIMPROPCLOSE to close the pane).
- **Entity snapping** BIM underlays support entity snapping if you enable snapping using the UOSNAP system variable (or RVTOSNAP for only .rvt and .rfa files or IFCOSNAP for only .ifc files). You can also use the Options command, Snapping tab.
- **Exploding** The BIM Explode command breaks apart an attached BIM underlay into polyface meshes and polylines that can be edited. BIM Explode is similar to using the Explode command for a BIM underlay except all BIM underlay intelligence is kept intact and each resulting entity is placed on its own layer.

Import an .ifc file to retain its entities.

When importing an .ifc file, all entities are available and construction and architectural entities are converted to AEC entities. For more details, see “Importing an IFC format file” on page 790.

To attach a BIM underlay

- 1 Do one of the following to choose BIM Underlay (- On the ribbon, choose Insert > BIM Underlay (in Data).
- On the menu, choose Insert > BIM Underlay.
- Type *bimattach* and then press Enter.
- 2 Choose the .rvt, rfa, or .ifc file you want to attach.
- 3 Click Open.
- 4 Select an insertion point.
- 5 Enter the scale in which to insert the file, then press Enter.
- 6 Enter the rotation to use for the insertion, then press Enter.

Working with data links

Data links allow you to share data between CAD drawings and spreadsheets. First you create a data link to a spreadsheet (.xls or .xlsx file), and then you select the data link when inserting a table in a drawing.

When content in the linked source .xls/.xlsx file changes, you can update the table in your drawing to include the modified data. And vice versa, you can update the external source .xls/.xlsx files to include the modified data from your drawing.

Using data links is different than importing an .xls/.xlsx file.

Importing an .xls or .xlsx file creates and fills a table with the data contained in the file, but the external source file is not linked to the drawing so future changes in either the drawing or source file cannot be updated automatically. For more details about importing, see “Importing an XLS format file” on page 794.



Creating a data link

Each data link corresponds to a single sheet in an .xls or .xlsx file. You can link to the whole spreadsheet, a named range, or a custom range that you specify.

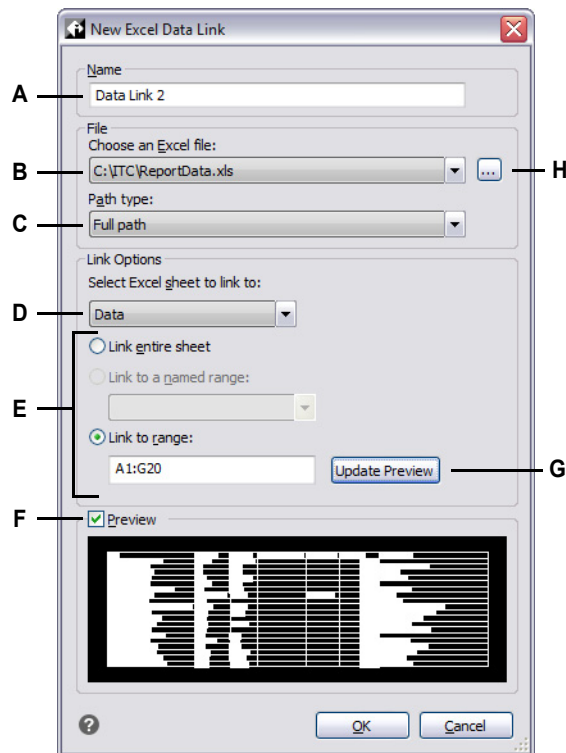
To create a data link, Microsoft® Excel® or another spreadsheet program that works with .xls or .xlsx files must be installed on your computer.

Once the data link is created, you can include the referenced data in your drawing by inserting a table and selecting the data link. The data link can also be created at the same time you draw tables. For more details about tables, see “Drawing an empty table” on page 176.

To create a data link

- 1 Do one of the following to choose Data Link Manager :
 - On the ribbon, choose Insert > Data Link Manager (in Data); Annotate > Data Link Manager (in Tables).
 - On the menu, choose Insert > Data Link > Data Link Manager.
 - Type *datalink* and then press Enter.
- 2 Click New Data Link .
- 3 In Name, enter a name for the data link.
- 4 In Choose an Excel File, click [...] to browse for an .xls, .xlsx file, or choose a file from the list of files that are linked to from the current drawing.
- 5 In Path Type, choose whether you want to use a full path, relative path, or no path to find the specified file.
- 6 In Select Excel Sheet to Link To, choose the sheet from the list of all sheets contained in the specified Excel file.

- 7 Choose one of the following:
- **Link Entire Sheet** Select to include data from the entire specified sheet within the Excel file.
 - **Link to a Named Range** Select to include data from a named range of cells within the Excel file. Available only when the specified file contains named ranges.
 - **Link to Range** Select to include data from a custom range of cells. You can specify an entire column (e.g., A:A), a set of entire columns (e.g., A:D), or a rectangular area (e.g., A1:D10).
- 8 Click OK.
- 9 Click OK again.




- A** Enter a name for the data link.
- B** Select an Excel file to link to, or select Browse for a File to select a different file.
- C** Choose whether to use a full path, relative path, or no path to find the specified file.
- D** Choose the sheet in the Excel file that contains the content you want to link to.
- E** Choose to link to all content in the sheet, a named range (if defined in the source file), or a custom range that you specify.
- F** Turns the preview on or off. For large source files, turning off the preview can improve performance.
- G** Click to update the preview. (Available if Link to Range is selected.)
- H** Click to browse for an .xls or .xlsx file to link to.

Opening the source file of a data link

If you need to open a source file to edit its content and then update its linked data in your drawing, you can easily open the source file by using the Data Link Manager command.

To open the source file of a data link




- 1 Do one of the following to choose Data Link Manager (- On the ribbon, choose Insert > Data Link Manager (in Data); Annotate > Data Link Manager (in Tables).
- On the menu, choose Insert > Data Link > Data Link Manager.
- Type *datalink* and then press Enter.
- 2 Select a data link.
- 3 In Details, confirm that the referenced file is the one you want to open.
- 4 Click Open Source File.

Managing data links



Data links can be renamed and deleted, and their settings can be changed, including the files they link to.

If a source file is moved, deleted, or renamed, the link becomes broken. You can reestablish the link to the same file in a new location or a different file altogether.



To modify a data link or fix a broken one

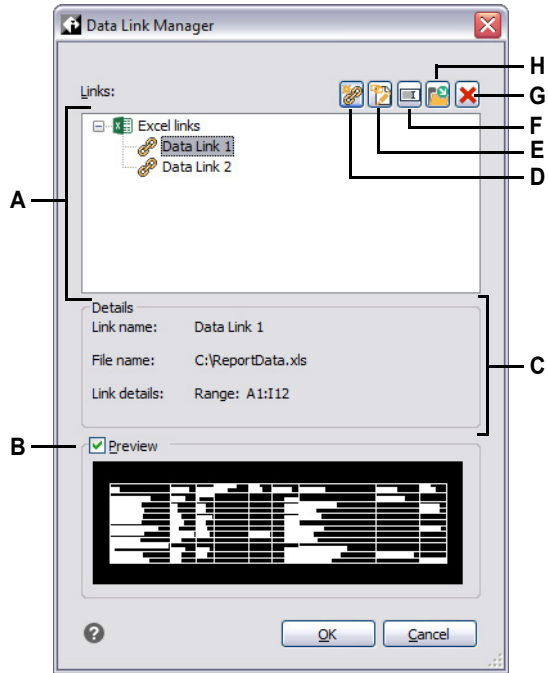
- 1 Do one of the following to choose Data Link Manager (- On the ribbon, choose Insert > Data Link Manager (in Data); Annotate > Data Link Manager (in Tables).
- On the menu, choose Insert > Data Link > Data Link Manager.
- Type *datalink* and then press Enter.
- 2 Select the data link you want to modify.
Broken data links are indicated in the Links list ().
- 3 Click Modify Data Link ().
- 4 If the data link is broken or for any other reason, click [...] to select a different file to link to.
- 5 Select any additional options you want to change. Click [?] for more details.
- 6 Click OK.
- 7 Click OK again.

To rename a data link

- 1 Do one of the following to choose Data Link Manager (- On the ribbon, choose Insert > Data Link Manager (in Data); Annotate > Data Link Manager (in Tables).
- On the menu, choose Insert > Data Link > Data Link Manager.
- Type *datalink* and then press Enter.
- 2 Select the data link you want to rename.
- 3 Click Rename Data Link ().
- 4 Type a new name, and then press Enter.
- 5 Click OK.
- 6 Click OK again.

To delete a data link

- 1 Do one of the following to choose Data Link Manager (- On the ribbon, choose Insert > Data Link Manager (in Data); Annotate > Data Link Manager (in Tables).
- On the menu, choose Insert > Data Link > Data Link Manager.
- Type *datalink* and then press Enter.
- 2 Select the data link you want to delete.
- 3 Click Delete Data Link ().
- 4 Click OK to confirm the deletion, or click No to cancel.
- 5 Click OK.
- 6 Click OK again.




- A** Displays a list of all data links in the drawing. Right-click for more options.
- B** Mark to turn the preview display on or off.
- C** Displays details about the selected data link.
- D** Click to create a data link.
- E** Click to modify the selected data link.
- F** Click to rename the selected data link.
- G** Click to delete the selected data link.
- H** Click to open the source file associated with the selected data link.


Updating data link content

If you make changes to linked data in the drawing or referenced source file, you can automatically synchronize the linked content.

To apply changes made in the source file to linked data in your drawing

- 1 Do one of the following to choose Update from Source (- On the ribbon, choose Insert > Update from Source (in Data); Annotate > Update from Source (in Tables).
- On the menu, choose Insert > Data Link > Update from Source.
- Type `datalinkupdate`, press Enter, and then choose Update Data Link.
- 2 Select the tables you want to update with modified data from external source files, or do one of the following:
 - Choose Data Link to enter the name of a data link to update.
 - Choose All Data Links to update all data links in the drawing.

To update the source file with changes made to linked data in your drawing

- 1 Do one of the following to choose Save to Source (- On the ribbon, choose Insert > Save to Source (in Data); Annotate > Save to Source (in Tables).
- On the menu, choose Insert > Data Link > Save to Source.
- Type `datalinkupdate`, press Enter, and then choose Write Data Link.
- 2 Select the tables that contain modified linked data you want to save to corresponding external source files, then press Enter.

Get notified about modified or missing data links.

By default, the status bar indicates when a table in the drawing is linked to data and a balloon message displays when a data link is modified or missing. Use the `DATALINKNOTIFY` system variable to change this setting.

Working with images

You can modify and view raster images directly inside of PTCAD. You can load, edit, and modify multiple images as overlays or underlays to your PTCAD drawings. The images can be selected for use with PTCAD commands by selecting the image frame, which can be turned on or off for printing or selection purposes.

PTCAD supports numerous image file formats, including BMP, JPG, GIF, EMF, TIF, PNG, WMF, SID, and many more.

Attaching images

When you attach an image to a drawing, the image displays in the drawing but is not saved in the drawing. Similar to an external reference, the image file remains saved in its original location on your computer, network, or other media.

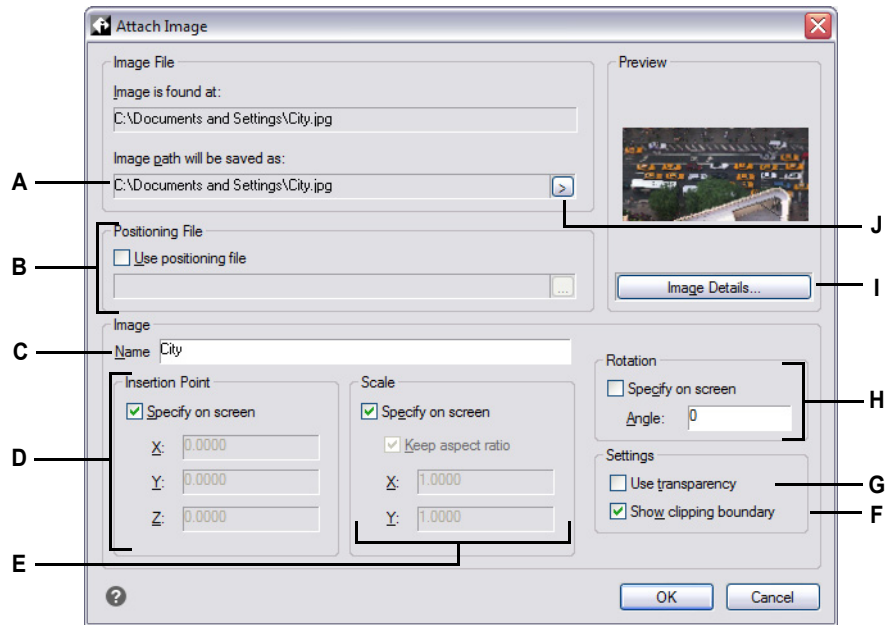
If you send or receive drawings that contain images, it is important to include with the drawing all of the image files attached to it. When you open a drawing that contains images, the source image files must be accessible for the images to display in the drawing.

To attach an image

- 1 Do one of the following to choose Attach Image:
 - On the ribbon, choose Insert > Attach Image (in Data).
 - Choose Insert > Image > Attach Image.
 - On the Image toolbar, click the Attach Image tool.
 - Type *imageattach* and then press Enter.
- 2 Specify a file to attach, and then click Open.
- 3 In Image Path will be Saved As, enter a different image file location, if necessary. You can click [>] to choose how you want to save the image path:
 - Full Path — The image is referenced using its full path, for example, c:\My Pictures\MyImage.jpg. Use this option if the image is saved in a folder unrelated to the current drawing folder.
 - Relative Path — The image is referenced using a path relative to the current drawing folder, for example, ..\My Pictures\MyImage.jpg. Use this option if the image is stored in a subfolder of the current drawing folder.
 - File Name Only — The image is referenced using its file name in the current drawing folder, for example, MyImage.jpg. Use this option if the image is saved in the same folder as the current drawing.
- 4 In the Attach Image dialog box, specify the position, scale, rotation, transparency, and clipping options, and then click OK.

NOTE *Transparency works for images that support alpha transparency, that is, images that have at least one color that can be viewed as a transparent color.*

- 5 In the drawing, specify an insertion point, scale, and rotation if you chose to specify those on the screen.



- A** Enter a different image file location, if necessary.
- B** Choose to place the image automatically in the drawing using specifications located in a positioning file. Enter the TWF file associated with the image or click [...] to select it.
- C** Enter a name for the image.
- D** Choose to specify the insertion point in the drawing upon insertion, or enter coordinates.
- E** Choose to specify the size of the image in the drawing upon insertion, or enter the size values.
- F** Choose whether to turn clipping display on or off for the image.
- G** Choose whether entities located under the image are visible (for images that support alpha transparency).
- H** Choose to specify rotation in the drawing upon insertion, or enter how many degrees to rotate the image to the left.
- I** Click to view image information, including color depth and resolution.
- J** Choose to save the image path as its full path, relative path to the current drawing folder, or file name in the current drawing folder.

You can also attach images using the Image Manager or Explorer

Choose Insert > Image > Image Manager, and then click Attach to specify an image and then attach it, or if you want to quickly add another occurrence of an image already located in the drawing, select the image in the Image Manager and then click Add. Or, choose Tools > Explorer and attach an image as an externally reference file.


Modifying images

You can modify an image by changing its brightness, contrast, fade, size, rotation, or transparency. These changes affect the image in the drawing only — not the original external image file.

In addition to modifying a single image or multiple images that you select, you can also modify all occurrences of an image within a drawing. For example, if your company logo appears in multiple locations throughout a drawing, you can use the Image Manager to specify the changes once and apply them to all occurrences of the logo.

You can use other PTCAD commands for typical modifications, such as Delete, Move, Copy, Rotate, and more.

To modify images

- 1 Do one of the following to choose Image Manager ():
 - On the ribbon, choose Insert > Image Manager (in Data).
 - On the menu, choose Insert > Image > Image Manager.
 - On the Image toolbar, click the Image Manager tool.
 - Type *image* and then press Enter.

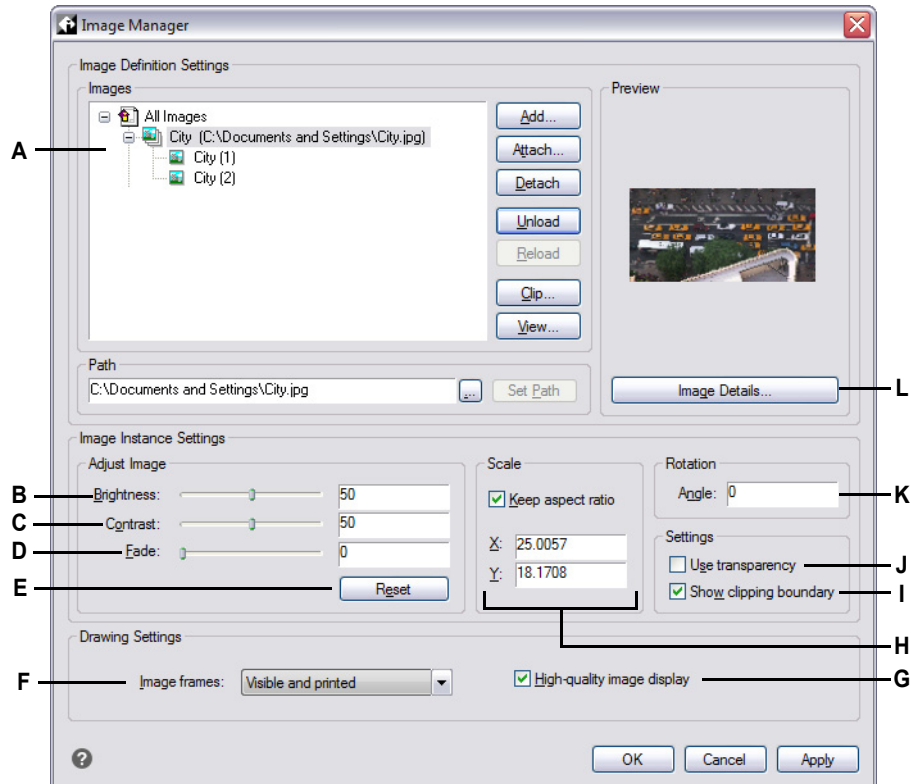
TIP You can also modify images by selecting one or more images in a drawing, and then choosing *Modify > Properties*.

- 2 In the Images list, select the image you want to modify. If there is more than one occurrence of the image in the drawing, do one of the following:
 - To modify all occurrences of the image, select a top-level image in the list.
 - To modify a single occurrence of the image, expand a top-level image in the list, and then select the individual image.
- 3 Adjust the Brightness, Contrast, and Fade by moving the slider to the setting you want or by entering an exact number. The image preview shows how your changes will affect the image.

TIP If you want to restore the image to the default brightness, contrast, and fade settings, click *Reset*.

- 4 Adjust the Size by making changes to the width (X) and height (Y) in drawing units. Mark *Keep Aspect Ratio* if you want the width and height to change together to retain the aspect ratio of the image.
- 5 Adjust the Rotation by entering the number of degrees you want to rotate the image to the left. Zero degrees indicates no rotation.
- 6 Mark *Use Transparency* if you want entities located under the image to be visible (for images that support alpha transparency, that is, images that have at least one color that can be viewed as a transparent color).

- 7 Mark Show Clipping Boundary if you want to show the image clipped, if a clipping boundary is defined for the image. Unmarking this option displays the whole image, even if a clipping boundary is defined.
- 8 Click OK.



- A Click a top-level image to modify all occurrences located in the drawing. Expand a top-level image and click an individual image to modify that occurrence only.
- B Move the slider or enter an exact number for image brightness.
- C Move the slider or enter an exact number for image contrast.
- D Move the slider or enter an exact number for image fade, which fades the image without displaying entities located under the image.
- E Click to return to the default brightness, contrast, and fade settings.
- F Choose whether all image frames in the drawing display and print.
- G Choose whether all images in the drawing display in high quality (requires more system resources).
- H Enter the width (X) and height (Y) in drawing units. Mark Keep Aspect Ratio to change width and height together.
- I Choose whether to turn clipping display on or off for the image.
- J Choose whether entities located under the image are visible (for images that support alpha transparency).
- K Enter how many degrees to rotate the image to the left.
- L Click to view details about the selected image.

Changing the display of images


You can change the following for how all images display in a drawing:

- Image quality — Images can display in high or low resolution.
- Image frames — Images can display and print with or without frames on their edges.

Changing the display quality for all images

High quality displays images in high resolution and requires more system resources. Draft quality displays images in low resolution and consumes fewer system resources. Changing the quality setting affects all images in the drawing.

To change the display quality for all images

- 1 Do one of the following to choose Image Quality (

TIP *This feature is also available in the Image Manager. Type image, then in the Image Manager dialog box, choose your selection for High-quality image display.*

Turning image frames on or off for all images

When image frames are turned on, a frame displays and prints on the edge of all images in the drawing. When image frames are turned off, none of the images display or print with a frame. Each image frame displays with the properties (layer, color, linetype, etc.) that are assigned to the image.

Turning image frames off may be helpful, for example, if the images are part of a background in the drawing.

To turn image frames on or off for all images


- 1 Do one of the following to choose Display Image Frame (- On the ribbon, choose Insert > Display Image Frame (in Data).
- On the menu, choose Insert > Image > Display Image Frame.
- On the Image toolbar, click the Display Image Frame tool.
- Type *imageframemode* and then press Enter.
- 2 Choose On or Off.
- 3 If you chose On, choose whether to print image frames.

Image frames can be turned on an off using various methods.

Type image, then in the Image Manager dialog box, make your selection in Image Frames. Or use the IMAGEFRAME system variable. Additionally, type tframe to turn on or off all image and wipeout frames in the drawing.

Clipping images


You can clip images so that only a portion of the image is visible in a drawing. The visible portion (or the invisible portion for inverted clips) can be in the shape of a rectangle, polygon, or existing polyline.

Image clipping can be turned on and off. If you turn off clipping for an image, the entire image is visible provided that the image is on a layer that is on and thawed. The clipping information is retained however, and you can turn clipping back on at any time.


If you delete clipping from an image, the clipping is removed permanently but the image itself remains in the drawing.

Clipping images in the shape of a polygon or rectangle


To clip an image in the shape of a polygon or rectangle

- 1 Do one of the following to choose Clip Image (- On the ribbon, choose Insert > Clip Image (in Data).
- On the menu, choose Insert > Image > Clip Image.
- On the Image toolbar, click the Clip Image tool.
- Type *imageclip* and then press Enter.
- 2 Select the edge of the image you want to clip.
- 3 Choose New Boundary to create a new clipping boundary.
- 4 By default, only the portion of the image located within the clipping polygon will be visible. If you want to invert the clip, that is, hide the area inside the boundary and make the outer portion visible, choose Invert Clip before the next step.
- 5 Choose one of the following:
 - Polygonal, then select the points for the polygon, and press Enter when the polygon is complete.
 - Rectangular, then select the first and opposite corners of the clipping rectangle.


*Clipping images in the shape of an existing polyline***To clip an image in the shape of an existing polyline**

- 1 Do one of the following to choose Clip Image ():
 - On the ribbon, choose Insert > Clip Image (in Data).
 - On the menu, choose Insert > Image > Clip Image.
 - On the Image toolbar, click the Clip Image tool.
 - Type *imageclip* and then press Enter.
- 2 Select the edge of the image you want to clip.
- 3 Choose New Boundary.
- 4 By default, only the portion of the image located within the clipping polygon will be visible. If you want to invert the clip, that is, hide the area inside the boundary and make the outer portion visible, choose Invert Clip before the next step.
- 5 Choose Select Polyline.
- 6 Select an existing polyline to use as the clipping boundary, then press Enter. If the polyline is not closed, the clipping boundary is formed as if the start and end points connect.

*Turning clipping on or off for images***To turn clipping on or off for an image**

- 1 Do one of the following to choose Clip Image ():
 - On the ribbon, choose Insert > Clip Image (in Data).
 - On the menu, choose Insert > Image > Clip Image.
 - On the Image toolbar, click the Clip Image tool.
 - Type *imageclip* and then press Enter.
- 2 Select the edge of the image for which you want to turn clipping on or off.
- 3 Choose On or Off.


*Removing clipping from images***To remove clipping from an image**

- 1 Do one of the following to choose Clip Image ():
 - On the ribbon, choose Insert > Clip Image (in Data).
 - On the menu, choose Insert > Image > Clip Image.
 - On the Image toolbar, click the Clip Image tool.
 - Type *imageclip* and then press Enter.
- 2 Select the edge of the image for which you want to remove clipping.
- 3 Choose Delete.

Unloading and reloading images

If you find that including an image affects system performance, you can unload it so only the image frame displays to mark its location. If you want an unloaded image to print, reload it before printing. You may also want to reload an image if the original file contains new content.


To unload and reload an image

- 1 Do one of the following to choose Image Manager (- On the ribbon, choose Insert > Image Manager (in Data).
- On the menu, choose Insert > Image > Image Manager.
- On the Image toolbar, click the Image Manager tool.
- Type *image* and then press Enter.
- 2 In the Images list, select the desired image. If there is more than one occurrence of the image in the drawing, do one of the following:
 - To unload or reload all occurrences of the image, select a top-level image in the list.
 - To unload or reload a single occurrence of the image, expand a top-level image in the list, and then select the individual image.
- 3 Do one of the following:
 - To unload the image so only its outer edge displays, click Unload.
 - To reload the image so its contents display and print, click Reload.

Changing the path for images

If the file associated with an image is renamed or moved to a different location, the program displays a message indicating that it cannot load the image. You can re-establish the link to the file by changing the path for the image.


To change the path for an image

- 1 Do one of the following to choose Image Manager ():
 - On the ribbon, choose Insert > Image Manager (in Data).
 - On the menu, choose Insert > Image > Image Manager.
 - On the Image toolbar, click the Image Manager tool.
 - Type *image* and then press Enter.
- 2 In the Images list, select the desired image. If there is more than one occurrence of the image in the drawing, do one of the following:
 - To change the path for all occurrences of the image, select a top-level image in the list.
 - To change the path for a single occurrence of the image, expand a top-level image in the list, and then select the individual image.
- 3 Click the [...] button.
- 4 Select the file with its new name or in its new location, and then click Open.
- 5 Click Set Path.

Deleting images

Once an image is no longer required in the drawing, you can delete it from the drawing. Deleting an image removes it from the drawing, and from the list of images in the Image Manager dialog box.

To delete an image

- 1 Do one of the following to choose Image Manager ():
 - On the ribbon, choose Insert > Image Manager (in Data).
 - On the menu, choose Insert > Image > Image Manager.
 - On the Image toolbar, click the Image Manager tool.
 - Type *image* and then press Enter.
- 2 In the Images list, select the image you want to delete. If there is more than one occurrence of the image in the drawing, do one of the following:
 - To delete all occurrences of the image, select a top-level image in the list.
 - To delete a single occurrence of the image, expand a top-level image in the list, and then select the individual image.
- 3 Click Detach.

Working with geographic locations

Adding a geographic location to a drawing allows you to draw entities that correspond to specific locations on a map, which can be especially helpful when working with drawings that contain survey, civil, and architectural data.

A geographic location is commonly specified by entering latitude, longitude, and elevation values with a coordinate system, but the location can also be specified using your current location or a Keyhole Markup Language map file (.kml or .kmz). Note that an Internet connection is required.


Once a geographic location is specified, an online map displays in the background of the drawing along with a red geographic location marker, which PTCAD uses to compute all relative geographic coordinates in the drawing. It is common to zoom and pan the map next, so you can view a specific area of the map. You may want to create position markers, which indicate and annotate geographic locations on the online map. Or you may want to capture an area as a map image, which creates a static image in the drawing. Then you can turn off the online map, and draw entities at specific geographic coordinates on top of the map image, updating the map image as needed and changing between hybrid, road, and satellite map types.

Adding a geographic location

Geographic locations can be added using latitude and longitude coordinates, your current location, or a Keyhole Markup Language map file (.kml or .kmz). After you specify a geographic location, a map displays with a red geographic location marker, which indicates the coordinates used to insert the online map and used to calculate all relative geographic coordinates.

An Internet connection is required.

To specify a geographic location

- 1 Do one of the following to choose Set Location (- On the ribbon, choose Insert > Set Location (in Geolocation).
- On the menu, choose Insert > Geolocation > Set Location.
- Type *geo* and then press Enter.

Only one geographic location can be specified in a drawing at a time.

*If the Set Location command is unavailable, a location is already set in the drawing. Use the Geographic Map command (type *geomap*) to check whether the display is turned off.*

- 2 If prompted, click Yes to allow the program to use the Internet. Or click No to cancel the command.

- 3 Do one of the following, depending on how you want to specify the location:
 - Use your current location — If necessary, turn on location access for your computer, then click Load Current Location.
 - Load a map file — Click Load Map File, select a .kml or .kmz file, then click Open. The first location mark defined in the selected file is used to place the geographic location marker.
 - Enter location values — Enter the latitude, longitude, and optional elevation.
- 4 In Coordinate System, select a coordinate system or choose Select Coordinate System.
The dialog box that displays may require extra time to open as it scans the entire coordinate database to find coordinate systems appropriate for the specified location.
- 5 Select a coordinate system.
The list displays coordinate systems that are referenced by the latitude and longitude previously determined in the Geographic Location - Set Location dialog box.
- 6 Click OK.
- 7 Click OK again.
- 8 In model space of the drawing, specify the location of the geographic location marker, which corresponds to the location on the Earth surface that you specified in the Geographic Location - Set Location dialog box.
- 9 Specify the angle of deviation from the x-axis that indicates the north direction. Or, to determine the north direction using a line, choose First Point, specify the start point of the line, then specify the end point of the line to set the angle.
An online map and a red geographic marker display in the drawing. You may need to zoom or pan to view the map precisely.


Changing how a geographic location map displays

You can display online maps in these ways:

- Aerial — Displays aerial satellite imagery.
- Road — Displays the road view.
- Hybrid — Displays aerial satellite imagery overlaid with roads.
- Off — Hides the online map in the current viewport. The map display is turned off, but the map and its data remain in the drawing.

An Internet connection is required.

To change the display of a geographic location map

- 1 Do one of the following to choose Geographic Map () , which is available only if an online map was added using the Set Location command and the map service is turned on using the Map Service command:
 - On the ribbon, choose Geolocation > Map Aerial, Map Road, Map Hybrid, or Map Off (in Online Map).
 - On the menu, choose Insert > Geolocation > Map Aerial, Map Road, Map Hybrid, or Map Off.
 - Type *geomap* and then press Enter.
- 2 If you entered the command at the command prompt, choose the map display type, then press Enter.


Removing a geographic location

When you remove a geographic location from a drawing, you remove all associated data, including the coordinate system details, online map, and the red geographic location marker.

If you need to hide the online map temporarily, you can simply turn off its display using the Geographic Map command. For more details, see “Changing how a geographic location map displays” on page 608 in this chapter.

If you need to turn off Internet access temporarily for the online map, use the Map Service command. For more details, see “Turning the map service on and off” on page 612 in this chapter.

To remove a geographic location


- 1 Do one of the following to choose Remove Location () , which is available only if an online map was added using the Set Location command:
 - On the ribbon, choose Geolocation > Remove Location (in Location).
 - On the menu, choose Insert > Geolocation > Remove Location.
 - Type *georemove* and then press Enter.
- 2 Choose Yes to confirm deletion.

Adding markers and annotations to a geographic location map

Position markers indicate and annotate a location on an online map. Each position marker has a point, leader, and multiline text.

An Internet connection is required.

To add a position marker to an online map

- 1 Do one of the following to choose Mark Position () , which is available only if an online map was added using the Set Location command and the map service is turned on using the Map Service command:
 - On the ribbon, choose Geolocation > Mark Position (in Tools).
 - On the menu, choose Insert > Geolocation > Lat-Long, My Location, or Point.
 - Type *geomarkposition* and then press Enter.
- 2 Choose one of the following, depending on how you want to define the position marker:
 - Lat-Long — Places the position marker by specifying its latitude and longitude. Enter the latitude and longitude values at the prompts that display.
 - My Location — Places the position marker at the point that corresponds to your current location. An Internet connection is required and location access must be turned on for your computer.
 - Point — Places the position marker by specifying its location in the drawing. At the prompt that displays, select the location in the drawing.
- 3 To specify an annotation at the position marker location, press Enter, and type multiline text. Or to specify another location for the annotation, click the desired point in the drawing, then type multiline text.
- 4 When finished typing text, click Close Editor to accept your changes and close the text editor.

You can change the size of position markers.


At the command prompt, type `geomarkpositionsize` and enter the desired size.

Working offline with images of geographic map areas

You can create a raster image, or map image, of a geographic map area. This is especially helpful when you need to work offline. Each map image can display with a type — aerial, road, or hybrid — and at a specific resolution.


An Internet connection is not required after a map image is created, but it is required while creating or updating it.

To create a map image


- 1 Do one of the following to choose Capture Area () , which is available only if an online map was added using the Set Location command and the map service is turned on using the Map Service command:
 - On the ribbon, choose Geolocation > Capture Area (in Online Map).
 - On the menu, choose Insert > Geolocation > Capture Area.
 - Type *geomapimage* and then press Enter.

- 2 Specify the first corner of the rectangle to define the map image, or choose Center to use the map's center point.
- 3 Specify the opposite corner of the rectangle to define the map image.


To change the display type of a map image

- 1 Select a map image.
- 2 Do one of the following to choose Map Image Type (- On the ribbon, choose Map Image > Map Aerial, Map Road, Map Hybrid (in Properties).
- Type *geomapimagesetmaptype* and then press Enter.
- 3 If you entered the command at the command prompt, choose the display type for the map image:
 - Aerial — Displays aerial satellite imagery.
 - Road — Displays the road view.
 - Hybrid — Displays aerial satellite imagery overlaid with roads.

To change the resolution of a map image

- 1 Select a map image.
- 2 Do one of the following to choose Map Image Resolution (- On the ribbon, choose Map Image > Very Fine, Fine, Optimal, Coarse (in Properties).
- Type *geomapimagesetresolution* and then press Enter.
- 3 If you entered the command at the command prompt, choose the resolution.

To reload or reset a map image

- 1 Select a map image.
- 2 Do one of the following to choose Reload Image () , which is only available if the map service is turned on using the Map Service command:
 - On the ribbon, choose Map Image > Reload Image (in Update).
 - Type *geomapimageupdate* and then press Enter.
- 3 Choose one of the following:
 - Optimize — Resets the map image to display with an optimal resolution for viewing.
 - Reload — Updates the map image to include data from the latest online map.

Use a shortcut.

Right-click a map image and choose Geolocation to select commands for working with map images.


Turning the map service on and off

If Internet privacy or bandwidth is a concern while working with a drawing, turning off the map service prevents the program from accessing the Internet to display the online map and prevents use of the Mark Position, Capture Area, and Reload Image commands. Then when you're ready, it's easy to turn on the map service again.

You might also need to turn the map service on if you previously chose to not allow the program to access the Internet when starting various geographic location commands.

If you need to hide the online map temporarily, you can simply turn off its display using the Geographic Map command. For more details, see "Changing how a geographic location map displays" on page 608 in this chapter.

To turn the map service on or off

- 1 Do one of the following to choose Map Service () , which is available only if an online map was added using the Set Location command:
 - On the ribbon, choose Geolocation > Map Service (in Location).
 - On the menu, choose Insert > Geolocation > Map Service.
 - Type *geomapservice* and then press Enter.
- 2 Do one of the following:
 - Choose Yes to turn on the map service, and if prompted, click Yes to allow internet use.
 - Choose No to turn off the map service.

Printing drawings

You can print a copy of your drawing exactly as you created it, or you can add formatting and specify print controls to change how your drawing looks when it is printed.

Sometimes you may require multiple printed drawings, each with a different look or layout. For example, you may need one printed drawing for a client presentation, along with several other variations for production contractors. For each type of printed drawing that you require, you can create a layout that defines its characteristics, including scale, area to print, print style tables, and more.

This section explains how to:

- Start printing right away.
- Set up a drawing to print multiple layouts from paper space on a Layout tab.
- Customize how you want your drawing to look when it is printed.
- Define how to print your drawing further using print styles.
- Print or plot your drawing.
- Publish your drawing using sheet sets or sheet lists.


Topics in this chapter

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Getting started printing

When you create a drawing, you do most of your work on the Model tab. At any time you can print your drawing to see how it looks on paper. It's easy to get started printing, and then later create layouts and custom print settings to enhance your printed output.

To start printing

- 1 From the Model tab, do one of the following to choose Print (

There are many print options detailed in this section, such as the scale of the drawing, print area, print style tables, and more.

Type *qprint* and then press Enter to print the current viewport.

The Print dialog box is bypassed and the drawing is sent directly to the selected printer.

Defining layouts for printing

You can print your drawing directly from the Model tab where you created it, or you can create custom layouts for printing on Layout tabs.

When printing from the Model tab, you can print your drawing exactly the way it appears, or you can modify the drawing before printing by adding dimensions, a legend, or a title block.

You typically use the Layout tabs for printing if you require multiple print layouts. You may also want to use a Layout tab for printing even if you want your drawing printed only one way. For example, if you want to include large amounts of text on your printed drawing, you can add the text to a Layout tab so it does not clutter your drawing while you work on the Model tab.

Understanding layouts

When you create a drawing, you do most of your work on the Model tab. Each drawing that you create can contain numerous layouts that simulate the paper on which you will print a copy of the drawing. Each of these layouts is created on a Layout tab.

You can prepare a separate layout for each way you want to print your drawing. The layout allows you to organize different views to control which portion of your drawing prints and at what scale.

Before you print, you can also include additional entities and layout settings that control how your drawing prints. Additional items only appear on the Layout tab, not on the Model tab. For example, a layout can contain dimensions, title blocks, legends, or keynotes that print with your model, but do not clutter the screen when you work with your model on the Model tab.

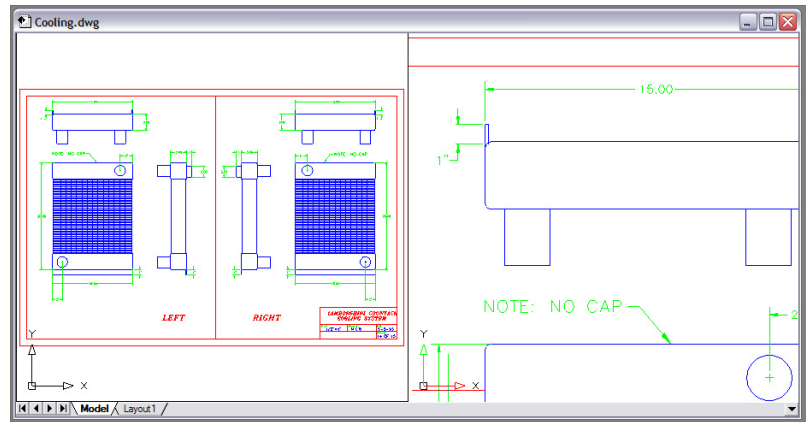
Use these general steps to prepare your drawing for printing multiple layouts:

- 1 On the Model tab, create your drawing.
- 2 Create a new layout. You can use an existing Layout1 or Layout2 tab, or you can create a new Layout tab. For details, see “Creating a new layout” on page 619 in this chapter.
- 3 Create at least one layout viewport on the Layout tab. Use each viewport to help control which portion of the drawing prints and at what scale. For details, see “Working with layout viewports” on page 622 in this chapter.
- 4 Include any additional items that may be required for the specific layout, such as dimensions, a legend, or a title block.
- 5 Specify additional settings for the layout, such as the scale of the drawing, print area, print style tables, and more. For details, see “Customizing and reusing print settings” on page 629 in this chapter.
- 6 Print or plot your drawing. For more details, see “Printing or plotting your drawing” on page 660 in this chapter.

Understanding paper space and model space

When you start a drawing session, your initial working area is called model space. Model space is an area in which you create two-dimensional and three-dimensional entities based on either the World Coordinate System (WCS) or a user coordinate system (UCS). You view and work in model space while using the Model tab.

Your view of this area is a single viewport that fills the screen. You can create additional views on the Model tab, called viewports, which can show the same or different two-dimensional or three-dimensional views, all of which are displayed in a tiled manner. You can work in only one of these viewports at a time on the Model tab, and you can print only the current viewport.



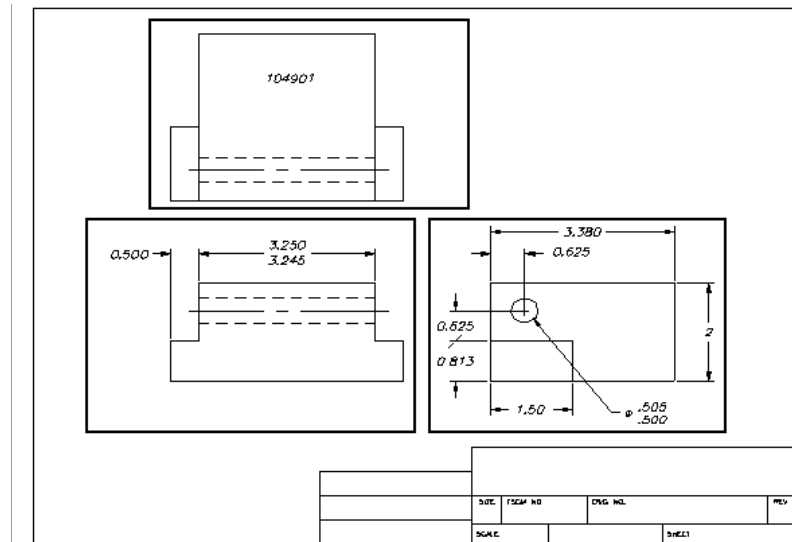
Model space with two viewports.

PTCAD provides an additional work area, called paper space. The contents of paper space represent the paper layout of your drawing. In this work area, you can create and arrange different views of your model similar to the way you arrange detail drawings or orthogonal views of a model on a sheet of paper. You can also add dimensions, keynotes, annotations, borders, title blocks, and other print-related entities in paper space, which reduces clutter when you work with your model in model space.

You view and work in paper space while using a Layout tab. Each view, or layout viewport, that you create in paper space provides a window of your drawing in model space. You can create one layout viewport or several. You can place layout viewports anywhere on the screen; their edges can be touching or not; and you can print them all at the same time.

You do not need to use paper space to print your drawing, but it offers several advantages:

- Print the same drawing with different print settings that you save with each layout, for example, printer configuration files, print style tables, lineweight settings, drawing scale, and more.
- Add print-related entities that are not essential to the model itself, such as keynotes or annotations, to reduce clutter when you work with your model in model space on the Model tab.
- For a single layout, create multiple layout viewports that print the model at different views and scales.



Paper space with layout viewports.

You can copy and move entities between model space and paper space.
 Choose *Modify > Change Space*, then select the entities you want to copy or move from paper space to model space or from model space to paper space.

Viewing drawings in paper space and model space

When you work in paper space on a Layout tab, you can still view your drawing in model space. First you need to create a layout viewport in paper space; this allows you to view your model space entities from paper space.

Within a layout viewport you can modify and snap to model space entities while working in model space and even snap to model space entities from paper space. Snapping to model space entities from paper space allows you to accurately dimension model space entities in paper space. And although it is generally more convenient to modify entities on the Model tab, there are often times when it is convenient to make modifications from a layout viewport on a Layout tab.

Zooming or panning the drawing in model space or paper space affects the entire drawing, unless you use multiple windows or viewports. Additionally, if you are working from paper space, you can lock the layout viewport so the viewport scale and view center do not change while panning and zooming in the layout viewport.

To view a drawing in model space on the Model tab

Do one of the following:

- Click the Model tab.
- Right-click a Layout tab and choose Activate Model Tab.
- Right-click Model or Paper Space in the status bar, and then choose Model.
- While using a layout tab, type *model*.
- Type *ctabnext* or press Ctrl+Page Down to switch to the next tab; type *ctabprev* or press Ctrl+Page Up to switch to the previous tab.

To view a drawing in paper space on a Layout tab

Do one of the following:

- Click one of the Layout tabs.
- Click the navigation arrows at the beginning of the row of Layout tabs to view Layout tabs in the order in which they appear.
- Click (▼) at the end of the row of Layout tabs, then choose a layout from the list.
- Right-click a Model or Layout tab and choose Activate Previous Layout.
- Type *layout* and then press Enter. In the prompt box, choose Set. Type a name for the layout you want to make current, and then press Enter.
- Double-click Model or Paper Space in the status bar. For example, double-click “Model” or “M:Tab Name” in the status bar to switch to paper space.
- While using a Layout tab, type *pspace* and then press Enter.
- While using a Layout tab, double-click outside of the layout viewport.
- Type *ctabnext* or press Ctrl+Page Down to switch to the next tab; type *ctabprev* or press Ctrl+Page Up to switch to the previous tab.

To view a drawing in model space on a Layout tab

Do one of the following:

- Click the desired Layout tab, and then create and view a layout viewport. For more details, see “Working with layout viewports” on page 622 in this chapter.
- Double-click Model or Paper Space in the status bar. For example, double-click “P:Tab Name” in the status bar to switch to model space on the current Layout tab.
- Type *mspace* and then press Enter.
- While using a Layout tab, double-click inside of the layout viewport.

Displaying the Model and Layout tabs

The Model and Layout tabs can be hidden, if desired. You may want to hide the tabs if you only work on the Model tab or if you use the command bar and status bar to switch between tabs.

To turn the Model and Layout tabs display on or off

Do one of the following:

- On the ribbon, choose View > Model and Layout tabs (in Display).
- Choose View > Display > Model and Layout tabs.
- Choose Tools > Options > Display tab, and choose Show Tabs.

Creating a new layout

In PTCAD, you can create multiple layouts for a single drawing. Each layout represents a sheet of paper. For each layout you can specify the print area, print styles, print scale, lineweight scale, pen mappings, and add viewports, dimensions, a title block, and other geometry specific to the layout.

The entities you add to a layout in paper space do not appear in model space.

Each layout requires at least one layout viewport, which is created automatically when you create a new layout. This viewport displays the drawing’s model space entities.

When you create a new drawing, the drawing automatically contains two default layouts: Layout1 and Layout2. You can start by using one of the default layouts, you can create your own, or you can create a new layout from another template (.dwt) file, drawing (.dwg) file, or drawing interchange (.dxf) file. You can also use PTCAD’s Explorer to create and manage layouts.


Each drawing can contain up to 255 layouts.

To create a new layout using the Layout1 or Layout2 tab


- 1 Click the Layout1 or Layout2 tab.
- 2 If necessary, set up at least one layout viewport. For details, see “Working with layout viewports” on page 622 in this chapter.

- 3 If desired, rename the layout. For details, see “To rename a layout” on page 622 in this chapter.

To create a new layout using a new Layout tab

- 1 Do one of the following to choose New Layout ():
 - On the ribbon, choose View > New Layout (in Layouts).
 - Choose Insert > Layout > New Layout.
 - On the Layouts toolbar, click the New Layout tool.
 - Type *layout*, press Enter, and choose New.
 - Right-click a Layout tab and choose New.
- 2 Type a unique name for your layout and then press Enter.
The name can be up to 255 characters in length and can contain letters, numbers, the dollar sign (\$), hyphen (-), and underscore (_), or any combination.
- 3 Set up at least one layout viewport. For details, see “Working with layout viewports” on page 622 in this chapter.

To create a new layout from an existing file

- 1 Do one of the following to choose Layout from Template ():
 - On the ribbon, choose View > Layout from Template (in Layouts).
 - On the menu, choose Insert > Layout > Layout from Template.
 - On the Layouts toolbar, click the Layout from Template tool.
 - Type *layout*, press Enter, and choose Template.
 - Right-click a Layout tab and choose From Template.
- 2 Select the desired template, drawing, or drawing interchange file that contains the layout you want, and then click Open.
- 3 Select the layout(s), and then click OK. You can choose multiple layouts by holding down Ctrl while selecting layout names.

Reusing layouts from other files

Save time by re-using layouts that you have already created. Within the same drawing, you may want to make a copy of a layout that contains most of the settings you want, and then make changes to the new copy. If you created layouts that you want to use again when you create new drawings, you can save the layouts as a drawing template.

To make a copy of a layout

- 1 Do one of the following:
 - Right-click the Layout tab you want to copy, then choose Copy.
 - Type *layout* and then press Enter. Then choose Copy and type the name of the layout you want to copy, and then press Enter.

- 2 Type a name for the new layout, and then click OK or press Enter, depending on how you started the command.

To save a layout as a drawing template


- 1 Do one of the following:
 - Right-click the Layout tab you want to save, then choose Save As.
 - Type *layout* and then press Enter. Then choose Save and type the name of the layout you want to save, and then press Enter.
- 2 Specify the file name and location for the template, and then click Save.

After you save a layout as a template, you can use the template when you create new drawings. You can also import the template's layouts into another drawing.

Exporting layouts to model space of a new drawing

Any layout can be exported to model space of a new drawing. During the export, entities might be modified by trimming, scaling, or exploding in order to visually represent the layout contents in model space.

To export a layout to model space of a new drawing

- 1 Click the Layout tab you want to export.
- 2 Do one of the following to choose Export Layout to Model (- On the ribbon, choose View > Export Layout to Model (in Layouts).
- On the menu, choose Insert > Layout > Export Layout to Model.
- On the Layouts toolbar, click the Export Layout to Model tool.
- Type *exportlayout* and then press Enter.
- Right-click the Layout tab and choose Export Layout to Model.
- 3 Specify the name of the file you want to create, and then click Save.

Managing layouts in a drawing

You can rename layouts, delete layouts, and view a list of all layouts available in a drawing. You can also change the order in which the Layout tabs appear; the Model tab is always stationary.

If you want to rename, delete, or reorder a layout when the Layout tabs are hidden, you can type *layout* to make your changes or choose View > Display > Model and Layout Tabs to display the tabs.

To rename a layout

- 1 Right-click the Layout tab you want to rename.
- 2 Choose Rename.
- 3 Type a new name for the layout.
- 4 Click OK.

The name can be up to 255 characters in length and can contain letters, numbers, the dollar sign (\$), hyphen (-), and underscore (_), or any combination.

To delete a layout

- 1 Right-click the Layout tab to delete.
- 2 Choose Delete.
- 3 Click OK to confirm the deletion.

You cannot delete the Model tab or the last remaining Layout tab.

To delete all geometry from the Model tab or a Layout tab, first select all geometry and then use the Erase command.

To reorder the Layout tabs

- 1 Right-click the Layout tab you want to move.
- 2 Do one of the following:
 - Choose Move Right, and then choose a new location.
 - Choose Move Left, and then choose a new location.

To view a list of all layouts

- 1 Type *layout* and then press Enter.
- 2 In the prompt box, choose ? to list all layouts.
- 3 Type *s* or press Enter to scroll through the layouts.

Working with layout viewports

A layout viewport is a window in a Layout tab (paper space) that displays all or a portion of a drawing's model space entities.

Understanding layout viewports

When you begin working in a drawing on the Model tab, it consists of a single view of your model. You may have created additional views by dividing the drawing space into multiple windows; each window is a separate viewport on the Model tab.

Similarly, when you begin working in a drawing on a Layout tab, it consists of a single view from paperspace of your model. You can also create multiple layout viewports that display unique views of your model. Each layout viewport functions as a window into your model space drawing — with each window looking different from the next. You can customize the view center, scale, layer visibility, and contents of each layout viewport. Each layout viewport is created as a separate entity that you can move, copy, or delete.


Click any layout viewport to make it the current viewport, and then add or modify model space entities in that viewport, even while snapping to model space entities from paper space. Any changes you make in one layout viewport are immediately visible in the other viewports (if the other layout viewports are displaying that portion of the drawing). Zooming or panning in the current viewport affects only that viewport.

This section focusses on working with layout viewports in paper space on a Layout tab. For additional information about viewports in model space, see “Dividing the current window into multiple windows” on page 243.

Creating layout viewports

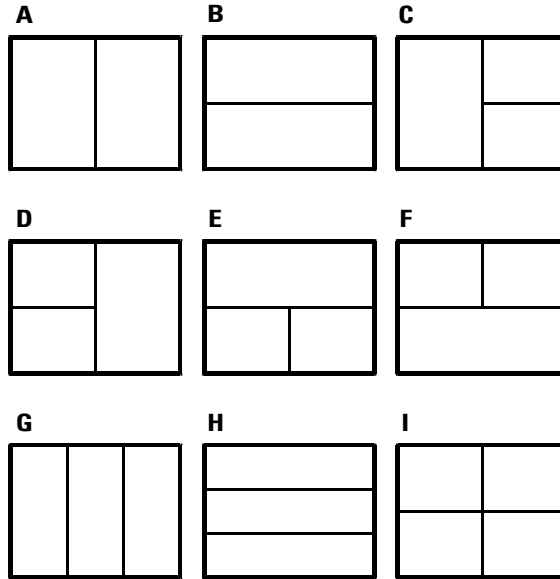
The first time you switch to a Layout tab, your model displays in a default layout viewport. You can create other layout viewports anywhere inside the drawing area. You can control the number of viewports created and the arrangement of the viewports.

To create layout viewports

- 1 Do one of the following to choose Layout Viewports (- On the ribbon, choose View > Layout Viewports (in Layouts or in Viewports).
- On the menu, choose View > Viewports > Layout Viewports.
- On the Viewports toolbar, click the Layout Viewports tool.
- Type *mview* and then press Enter.
- 2 Specify two opposing corners to create a custom rectangular viewport, or in the prompt box, choose one of the following:
 - Fit To View — Creates a layout viewport that fills the screen.
 - Entity — Converts a closed entity to a layout viewport. You can convert a circle, ellipse, closed polyline, spline, or region.
 - Polygonal — Creates a non-rectangular layout viewport.
 - Create 2 Viewports, Create 3 Viewports, Create 4 Viewports — Creates two, three, or four layout viewports using an orientation that you specify. You can select whether to arrange the viewports to fill the current graphic area or a rectangular area that you specify.

The border of a new layout viewport is created on the current layer.

You can make layout viewport borders invisible by creating a new layer before you create layout viewports and then turning off that layer after you create the layout viewports. To select a layout viewport's borders, you must turn that layer back on before you can rearrange or modify the layout viewport.



You can create a single layout viewport, or you can divide the graphic area into two viewports arranged vertically (A) or horizontally (B); three viewports arranged left (C), right (D), above (E), below (F), vertically (G), or horizontally (H); or four viewports (I).


Viewing and scaling layout viewports

If you have created numerous layout viewports, your system performance may be affected. If necessary, you can turn a layout viewport on or off. Turning off a layout viewport does not delete the viewport or its contents; it simply turns off its display.


You can also change how you view items within a layout viewport by specifying a scale factor, which changes how large or small model space entities appear within the layout viewport. Another helpful tool is the Align Space command, which changes the pan and zoom magnification of model space entities in a layout viewport so they align with entities in paper space or in other layout viewports.

While working in a layout viewport, you can use the Maximize Viewport command to enlarge the view to full size and emulate model space, allowing you to easily work on the geometry in that view. When done, use the Minimize Viewport command to switch back to the original scale and center point of the layout viewport.

To turn layout viewports on or off


- 1 Click the desired Layout tab.
- 2 Do one of the following to choose Layout Viewports ():
 - On the ribbon, choose View > Layout Viewports (in Layouts or in Viewports).
 - On the menu, choose View > Viewports > Layout Viewports.
 - On the Viewports toolbar, click the Layout Viewports tool.
 - Type *mview* and then press Enter.
- 3 Choose On or Off.
- 4 Select the edge of the layout viewport to turn on or off, and then press Enter.

To maximize a layout viewport

- 1 On a Layout tab, select a layout viewport. Or, skip this step to maximize the current layout viewport.
- 2 Do one of the following:
 - On the ribbon, choose View > Maximize Viewport (in Viewports).
 - On the menu, choose View > Viewports > Maximize Viewport.
 - On the status bar, click Maximize Viewport (.
 - Type *vpmax* and then press Enter.


The layout viewport is enlarged.

To minimize a layout viewport (if it is maximized)


- 1 Do one of the following:
 - On the ribbon, choose View > Minimize Viewport (in Viewports).
 - On the menu, choose View > Viewports > Minimize Viewport.
 - On the status bar, click Minimize Viewport ().
 - Type *vpmin* and then press Enter.

The layout viewport returns to its original scale and center point.

To change the layout viewport scale

- 1 Do one of the following to choose Properties ():
 - On the ribbon, choose View > Properties (in Panes).
 - On the menu, choose View > Display > Properties or Modify > Properties.
 - On the Modify toolbar, click the Properties tool.
 - Type *entprop* and then press Enter.
- 2 Select the edge of the layout viewport.
- 3 In Custom Scale, enter the scale at which you want to view model space entities from within the layout viewport.
- 4 Click OK.

To report the scale of a layout viewport scale

- 1 Do one of the following to choose List Viewport Scale ():
 - On the ribbon, choose Express Tools > List Viewport Scale (in Layout).
 - On the menu, choose Express Tools > Layout > List Viewport Scale.
 - Type *vpscale* and then press Enter.
- 2 Select the edge of the layout viewport, then press Enter.


The viewport scale (paperspace to model space) displays in the command bar.

To change the scale of model space entities relative to paper space


- 1 Click the Model tab.
- 2 Click a viewport to make it current.
- 3 Choose View > Zoom > Zoom.
- 4 Type the zoom scale factor relative to paper space by appending the suffix *xp* to the scale factor, and then press Enter.

For example, to increase the scale of the entities in the viewport on the Model tab to twice the size of paper space units, type *2xp*. To decrease the scale to half the size of paper space units, type *.5xp*.

To change the zoom and pan of a layout viewport relative to paper space or another layout viewport

- 1 Click a Layout tab.
- 2 Do one of the following to choose Align Space (- On the ribbon, choose Express Tools > Align Space (in Layout).
- On the menu, choose Express Tools > Layout > Align Space.
- Type *alignspace* and then press Enter.
- 3 Specify the first alignment point in model space of a layout viewport.
- 4 Specify the second alignment point in model space of the same layout viewport, or press Enter if you do not need to specify a second point.
- 5 Specify the first alignment point in paper space or another layout viewport.
- 6 If you previously specified a second alignment point in model space, specify the second alignment point in paper space or another layout viewport.
- 7 Select the layout viewport to align by changing its pan and zoom magnification, then press Enter.

To synchronize layout viewports

- 1 Do one of the following to choose Synchronize Viewports (- On the ribbon, choose Express Tools > Synchronize Viewports (in Layout).
- On the menu, choose Express Tools > Layout > Synchronize Viewports.
- Type *vpsync* and then press Enter.
- 2 Select the master layout viewport that has the desired zoom factor.
- 3 Select one or more viewports to align with the master viewport, then press Enter.

Modifying layout viewports


After you create layout viewports, you can modify them as needed. On the Layout tab, you can snap to the viewport borders using entity snaps. You can copy, delete, move, scale, and stretch layout viewports as you would any other drawing entity.

Additionally, you can lock a layout viewport so the viewport scale and view center do not change in model space while panning or zooming in the layout viewport. If you are working on model space entities from a Layout tab, locking the layout viewport prevents you from constantly changing the layout viewport scale and view center.

And assigning a UCS to each viewport allows you to quickly switch between layout viewports and immediately draw in a different UCS. This can greatly increase productivity, especially when creating complex 3D models.

Modifying a layout viewport on a Layout tab does not affect the model space entities within the layout viewport.

To modify layout viewport properties

- 1 Click the desired Layout tab.
- 2 Do one of the following to choose Properties ():
 - On the ribbon, choose View > Properties (in Panes).
 - On the menu, choose View > Display > Properties or Modify > Properties.
 - On the Modify toolbar, click the Properties tool.
 - Type *entprop* and then press Enter.
- 3 Select the edge of the layout viewport you want to modify.
- 4 Adjust the center point, width, or height of the viewport.
- 5 In Custom Scale, enter the scale at which you want to view model space entities from within the layout viewport.
- 6 In Display Locked, choose True to lock the viewport scale and view in model space while panning or zooming in the layout viewport.
- 7 Mark UCS per Viewport if you want to use a unique UCS for each layout viewport.
- 8 Click OK.

You can select only layout viewports for modification.

If you click a viewport on the Model tab, it makes that viewport active, not available for modification.

Clipping layout viewports

You can clip layout viewports so that only a portion of the viewport is visible on a Layout tab. You can clip layout viewports in the shape of a new polygon or an existing circle, ellipse, closed spline, closed polyline, or region.

If you delete clipping from a layout viewport, the clipping is removed permanently but the viewport itself and its contents remain in the drawing.

To clip a layout viewport in the shape of an existing entity

- 1 Click a Layout tab, and select the desired layout viewport.
- 2 Type *vpclip* and then press Enter.
- 3 In the drawing, select an existing circle, ellipse, closed spline, closed polyline, or region to use as a clipping boundary.

To clip a layout viewport in the shape of a new polygon

- 1 Click a Layout tab, and select the desired layout viewport.
- 2 Type *vpclip* and then press Enter.
- 3 Press Enter to create a new clipping boundary.
- 4 Define the first point of the clipping polygon.
- 5 Define additional points.
- 6 Press Enter when done.

To delete a clipping boundary

- 1 Click a Layout tab, and select the desired layout viewport.
- 2 Type *vpclip* and then press Enter.
- 3 Choose Delete, and then press Enter.

Customizing and reusing print settings

Most drawings require adjustments to print settings in order to print the way you want. You can make adjustments to the print settings each time you print, but you can also create page setups, which save the print settings and assign them to different layouts in your drawing. If you have several perspectives of your drawing that require printing, using page setups is the most efficient way to print.

Working with page setups


Page setups store printer information for specific models or layouts, which eliminates the need to completely reconfigure your print settings each time you print a drawing and helps ensure that each perspective of a drawing prints as planned.

Assigning a page setup to a model or layout

Because the main model on the Model tab and the various layouts for printing on the Layout tabs all may require unique print settings, the model and each layout can be assigned a separate page setup. If some layouts use the same print settings, those layouts can be assigned the same page setup.

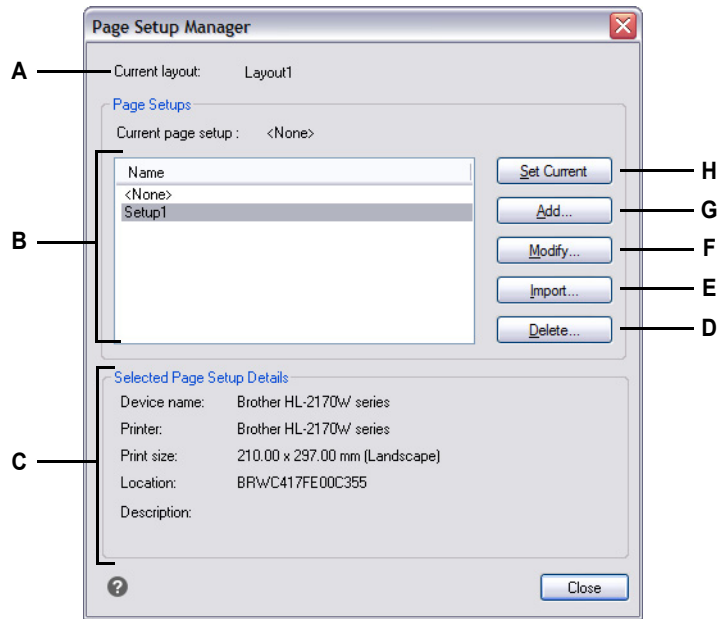
Assigning a page setup to a model or layout doesn't mean it will always print with the specified settings. All of the print settings specified for a page setup can be overridden at print time.

To assign a page setup to a model or layout

- 1 Click the Model tab or Layout tab that you want to assign a page setup.
- 2 Do one of the following to choose Page Setup Manager (- On the ribbon, choose the Application button then choose Page Setup Manager, or choose Output > Page Setup Manager (in Print).
- On the menu, choose File > Page Setup Manager.
- On the Format toolbar, click the Page Setup Manager tool.
- Type *pagesetup* and then press Enter.
- 3 Select the desired page setup.
- 4 Click Set Current.
- 5 Click OK.

You can also choose a page setup at print time.

In the Print dialog box, select a different page setup from the Page Setup list before you click Print.



- A** Displays the name of the model or layout to which the current page setup is assigned.

B Displays a list of all model page setups or layout page setups, depending on whether you were viewing the Model tab or a Layout tab before opening the dialog box.

C Displays details about the currently selected page setup.
- D** Click to delete the currently selected page setup.

E Click to import a page setup from a different drawing.

F Click to modify the settings for the currently selected page setup.

G Click to create a new page setup.

H Click to assign the currently selected page setup to the current layout.


Creating a page setup

There are two types of page setups:

- Model page setup — Contains print settings available for the model on the Model tab.
- Layout page setup — Contains print settings available for one or more layouts on the Layout tabs.

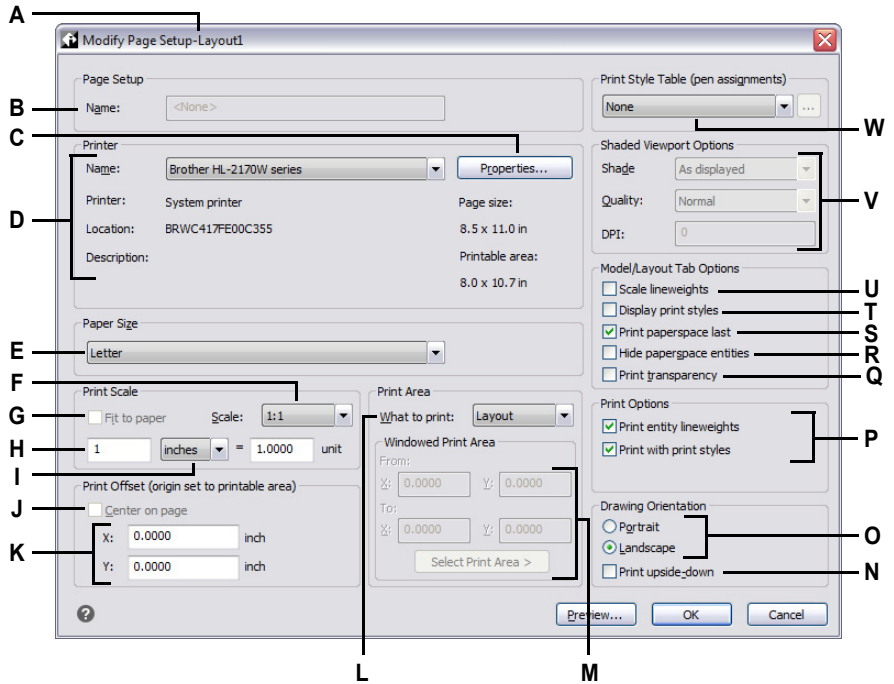
PTCAD comes with two default page setups — one model page setup and one layout page setup. You can create as many additional page setups, of either type, as required for any drawing. Each page setup specifies many aspects of printing, including page size, default printer or plotter, page orientation, print scale, and more.

To create a page setup

- 1 To create a model page setup, click the Model tab. To create a layout page setup, click any Layout tab.
- 2 Do one of the following to choose Page Setup Manager (- On the ribbon, choose the Application button then choose Page Setup Manager, or choose Output > Page Setup Manager (in Print).
- On the menu, choose File > Page Setup Manager.
- On the Format toolbar, click the Page Setup Manager tool.
- Type *pagesetup* and then press Enter.
- 3 Click Add.

You can create a new page setup based on the print settings of an existing page setup. Select an existing page setup in the list, then click Add. The new page setup uses the print settings of the existing page setup as a starting point.

- 4 Enter a name for the page setup, then click OK.
- 5 Select the desired print options.
- 6 Click OK.




- A** Displays "Layout" if creating a layout page setup or "Model" if creating a model page setup.
- B** Enter any changes to page setup name.
- C** Click to specify options for the selected printer or printer configuration (.pc3) file.
- D** Select the printer or printer configuration file and view its details.
- E** Select a paper size supported by the selected printer.
- F** Select a predefined print scale, or choose Custom to specify your own.
- G** Select to fit the specified print area to the current paper size.
- H** Specify the custom scale for the print area by typing the ratio of drawing units to printed inches or printed millimeters.
- I** Click to specify drawing units and paper size in millimeters or inches.
- J** Select to center the print area on the printed page.
- K** Type x- and y-coordinates to specify the origin of the print area.
- L** Click to select the area of the drawing that you want to print.
- M** Type the x- and y-coordinates of the two opposing corners of the rectangular area to print, or click Select Print Area to specify coordinates in the drawing window. (Available only if Window is selected for What to print.)
- N** Select to print the drawing upside down on your printer.
- O** Select portrait (vertical) or landscape (horizontal) orientation.
- P** Select whether to print lineweights and print styles.
- Q** Select to print entities with their assigned transparency.
- R** Select to prevent paperspace entities from printing.
- S** Select to print paperspace entities after printing modelspace entities.
- T** Select to show print styles when viewing the layout.
- U** Select to print entities with their assigned lineweights. If you turn off lineweight printing, entities print with a default outline.
- V** Select options for shaded viewports. Quality and DPI are not currently implemented.
- W** Select a print style table to apply during printing, or select None. Click [...] to modify the selected print style table.

Modifying an existing page setup

You can change any of the print settings associated with a page setup, which eliminates the need to override the settings when it comes time to print the model or each layout that is assigned the page setup.

If you change the settings for a layout print setup, all layouts assigned that print setup will print using the new settings.


To modify an existing page setup

- 1 To modify a model page setup, click the Model tab. To modify a layout page setup, click any Layout tab.
- 2 Do one of the following to choose Page Setup Manager (- On the ribbon, choose the Application button then choose Page Setup Manager, or choose Output > Page Setup Manager (in Print).
- On the menu, choose File > Page Setup Manager.
- On the Format toolbar, click the Page Setup Manager tool.
- Type *pagesetup* and then press Enter.
- 3 Select the page setup you want to change.
- 4 Click Modify.
- 5 Select the desired print options.
- 6 Click OK.

Importing page setups from a different drawing

If you have a different drawing file that contains one or more page setups defined with the print settings you need for your current drawing, you can import the page setups from the other file instead of re-creating them in the current drawing.

To import a page setup from a different drawing

- 1 Do one of the following to choose Page Setup Manager (- On the ribbon, choose the Application button then choose Page Setup Manager, or choose Output > Page Setup Manager (in Print).
- On the menu, choose File > Page Setup Manager.
- On the Format toolbar, click the Page Setup Manager tool.
- Type *pagesetup* and then press Enter.
- 2 Click Import.

Use a shortcut.

Type *psetupin*.


- 3 Select the drawing that contains the page setups you want to import, then click Open.
- 4 Select the model and layout page setups to import.
- 5 Click OK.

All page setups from the specified drawing file are imported, even though they might not all display in Page Setup Manager. Imported model page setups display if you're working on a Model tab, and layout page setups display if you're working on a Layout tab.

Deleting a page setup

If you delete a page setup that is assigned to the model or a layout, that model or layout will no longer be assigned a page setup.

To delete a page setup

- 1 To delete a model page setup, click the Model tab. To delete a layout page setup, click any Layout tab.
- 2 Do one of the following to choose Page Setup Manager (- On the ribbon, choose the Application button then choose Page Setup Manager, or choose Output > Page Setup Manager (in Print).
- On the menu, choose File > Page Setup Manager.
- On the Format toolbar, click the Page Setup Manager tool.
- Type *pagesetup* and then press Enter.
- 3 Select the page setup you want to delete.
- 4 Click Delete.
- 5 Click Yes to confirm the deletion.

Setting the paper size and orientation

You can specify a paper size and paper orientation for all drawings. You can also adjust the orientation by printing a drawing upside down on the paper. Each layout in your drawing can specify whether to print upside down.

To select the paper size and orientation


- 1 Click the Layout tab or Model tab for which you want to set paper size and orientation.
- 2 Do one of the following to choose Page Setup Manager (

Selecting a printer or plotter

You can specify a printer or plotter to be used when printing any drawing. You can print your drawing on any printer or plotter that is compatible with Windows, including raster printers.

Instead of a physical printer, you can choose to print directly to a file, including .pdf, dwf, .png, and .jpg. Printing to a file is not the same as exporting. If you need to export a drawing to one of the various file formats available, see “Exporting drawings” on page 799.

To select a printer or plotter

- 1 Click the Layout tab or Model tab for which you want to select a printer.
- 2 Do one of the following to choose Page Setup Manager ():
 - On the ribbon, choose the Application button then choose Page Setup Manager, or choose Output > Page Setup Manager (in Print).
 - On the menu, choose File > Page Setup Manager.
 - On the Format toolbar, click the Page Setup Manager tool.
 - Type *pagesetup* and then press Enter.
- 3 Select the desired page setup, then click Modify.
- 4 From the Printer Name list, select a printer or plotter.
- 5 If desired, click Properties to choose options for the currently selected printer.
- 6 Click OK.
- 7 Click OK.

The printer or plotter you select for a page setup is also used when you publish sheet lists. For details, see “Publishing a sheet list to the printer named in page setup” on page 677 in this chapter.

Setting the scale and view


You can print or plot the entire drawing or a selected portion of a drawing. You can choose to print what is visible on the screen, or you can specify to print an area of the drawing.

You can control the position of the drawing on the paper by specifying the origin of the print area, the location of the lower left corner of the print area, in relation to the lower left corner of the paper. The origin is normally set to 0,0, which places the lower left corner of the print area as close to the lower left corner of the paper as the printer or plotter will allow. You can specify a different origin, however, by specifying different coordinates.


When you create a drawing, you generally draw entities full-size. When you print the drawing, you can specify the scale of the resulting print or let the program adjust the size of the drawing to fit the paper. To print the drawing at a specific scale, specify the scale as a ratio of drawing units to printed units.

If you are printing from a Layout tab, the scale and view options you specify can be different for each layout that you create.


To automatically scale the drawing for printing

- 1 Click the Layout tab or Model tab for which you want to set to scale automatically.
- 2 Do one of the following to choose Page Setup Manager (- On the ribbon, choose the Application button then choose Page Setup Manager, or choose Output > Page Setup Manager (in Print).
- On the menu, choose File > Page Setup Manager.
- On the Format toolbar, click the Page Setup Manager tool.
- Type *pagesetup* and then press Enter.
- 3 Select the desired page setup, then click Modify.
- 4 In Print Scale, select Fit to Paper to scale the drawing to fit on one printed page.
- 5 Click OK.
- 6 Click OK.

To specify the scale factor yourself


- 1 Click the Layout tab or Model tab for which you want to specify the scale factor.
- 2 Do one of the following to choose Page Setup Manager (- On the ribbon, choose the Application button then choose Page Setup Manager, or choose Output > Page Setup Manager (in Print).
- On the menu, choose File > Page Setup Manager.
- On the Format toolbar, click the Page Setup Manager tool.
- Type *pagesetup* and then press Enter.
- 3 Select the desired page setup, then click Modify.
- 4 In Print Scale, do one of the following:
 - Select a pre-defined scale in the Scale list. For example, choose 1:2 if you want 1 printed unit (inch or millimeter) to equal 2 drawing units. The list of available scales is set up using the Scales List command. For more details, see “Customizing the scales list” on page 64.
 - Type the ratio of printed units of measure (inches or millimeters) to drawing units.
- 5 To specify the printed units of measure, choose Inches or Millimeters.
- 6 Click OK.
- 7 Click OK.

To specify a portion of the drawing to print

- 1 Click the Layout tab or Model tab for which you want to specify the area to print.
- 2 Do one of the following to choose Page Setup Manager (- On the ribbon, choose the Application button then choose Page Setup Manager, or choose Output > Page Setup Manager (in Print).
- On the menu, choose File > Page Setup Manager.
- On the Format toolbar, click the Page Setup Manager tool.
- Type *pagesetup* and then press Enter.
- 3 Select the desired page setup, then click Modify.
- 4 In What to Print, click one of the following:
 - Display — Prints the view on the screen.
 - Extents — Prints the area that contains entities in the drawing.
 - Limits — Prints to the limits defined for the drawing. (Available for model page setups only.)
 - Layout — Prints to the edge of the layout. (Available for layout page setups only.)
 - View — Prints the selected saved view. (Available for drawings that have saved views.)
 - Window — Prints the portion of the drawing contained in the specified window, maintaining the aspect ratio of the windowed area to the drawing.

If you clicked Window, you must specify the window. Under Windowed Print Area, enter the diagonal x- and y-coordinates of the window, or select the area on the screen.
- 5 Click OK.
- 6 Click OK.


To specify the print area origin

- 1 Click the Layout tab or Model tab for which you want to setting paper size and orientation.
- 2 Do one of the following to choose Page Setup Manager (- On the ribbon, choose the Application button then choose Page Setup Manager, or choose Output > Page Setup Manager (in Print).
- On the menu, choose File > Page Setup Manager.
- On the Format toolbar, click the Page Setup Manager tool.
- Type *pagesetup* and then press Enter.
- 3 Select the desired page setup, then click Modify.
- 4 Under Print Offset, do one of the following:
 - To center the specified print area on the printed page, select Center on Page.
 - To specify an origin for the print area, type the x- and y-coordinates.
- 5 Click OK.
- 6 Click OK.

Specifying print options specifically for layouts

Each layout in your drawing can specify certain print settings that apply only to layouts: lineweight scaling, print style display, and paperspace print options.

To set print options for only layouts


- 1 Click a Layout tab.
- 2 Do one of the following to choose Page Setup Manager (- On the ribbon, choose the Application button then choose Page Setup Manager, or choose Output > Page Setup Manager (in Print).
- On the menu, choose File > Page Setup Manager.
- On the Format toolbar, click the Page Setup Manager tool.
- Type *pagesetup* and then press Enter.
- 3 Select the desired page setup, then click Modify.
- 4 In the Model/Layout Tab Options area, choose the desired print settings:
 - Scale lineweights — Select to print lineweights in proportion to the specified Print Scale settings; if not selected, lineweights print at their assigned size. Note that print styles can also affect how lineweights print.
 - Display print styles — Select to show print styles when viewing the layout.

- Print paperspace last — Select to print paperspace entities after printing model-space entities. By default, paperspace entities print first.
 - Hide paperspace entities — Select to prevent paperspace entities from printing.
- 5 Click OK.
 - 6 Click OK.

Specifying shaded viewport print options

Each model page setup can specify how to print shaded viewports: as displayed, wire-frame, hidden, or rendered. Note that Quality and DPI are not currently implemented.


To set print options for shaded viewports

- 1 Click the Layout tab or Model tab for which you want to set shaded viewport settings.
- 2 Do one of the following to choose Page Setup Manager (- On the ribbon, choose the Application button then choose Page Setup Manager, or choose Output > Page Setup Manager (in Print).
- On the menu, choose File > Page Setup Manager.
- On the Format toolbar, click the Page Setup Manager tool.
- Type *pagesetup* and then press Enter.
- 3 Select the desired page setup, then click Modify.
- 4 In the Shaded Viewport Options area, choose the desired settings:
 - Shade — Select how to print shaded viewports
 - Quality — Select the resolution to use for the printed viewport. (Not currently implemented.)
 - DPI — Enter the custom dots per inch to use for printing the viewport shading. Available only if Quality is set to Custom. (Not currently implemented.)
- 5 Click OK.
- 6 Click OK.

Specifying pen and line printing options

Each layout in your drawing can specify certain print settings that apply only to layouts: lineweight scaling, print style display, and paperspace print options.

To set pen and line printing options

- 1 Click the Layout tab or Model tab for which you want to set pen and line printing options.
- 2 Do one of the following to choose Page Setup Manager ():
 - On the ribbon, choose the Application button then choose Page Setup Manager, or choose Output > Page Setup Manager (in Print).
 - On the menu, choose File > Page Setup Manager.
 - On the Format toolbar, click the Page Setup Manager tool.
 - Type *pagesetup* and then press Enter.
- 3 Select the desired page setup, then click Modify.
- 4 Choose the desired settings:
 - Print style table — Select a print style table to apply during printing, or select None. If you select a print style table, you can click [...] to modify its settings.
 - Print transparency — Select to print entities with their assigned transparency. Note that turning on this option can slow printing. Also note that the PLOTTRANSPARENCYOVERRIDE system variable can override transparency print settings for all layout viewports at once.

A print style table and print transparency cannot be used at the same time.

 - Print entity lineweights — Select to print entities with their assigned lineweights. If you turn off lineweight printing, entities print with a default outline.
 - Print with print styles — Select to print according to the print style settings in the currently selected print style table.
- 5 Click OK.
- 6 Click OK.

Using printer configuration files

Printer configuration files store the printer information you use for specific drawings or layouts, which eliminates the need to completely reconfigure your print settings each time you print a drawing. Printer configuration files also allow you to share and reuse print settings between different drawings and layouts.


PTCAD supports the printer configuration files (PCP and PC3 files) used by AutoCAD. This feature makes it possible to use existing PCP files saved in AutoCAD, as well as to save your PTCAD print configuration settings to the PC3 format.

You can convert an AutoCAD PC2 or PCP file using the AutoCAD PCINWIZARD command.

Selecting a printer configuration file for printing

You can specify that a printer configuration file be used when printing any drawing.

To select a printer configuration file for printing

- 1 Click the Layout tab or Model tab for which you want to print.
- 2 Do one of the following to choose Print (- On the ribbon, choose the Application button then choose Print, or choose Output > Print (in Print).
- Choose File > Print.
- On the Standard toolbar, click the Print tool.
- Type *print* and then press Enter.
- 3 From the Printer Name list, select a .pc3 or .pcp file.
- 4 Click OK to print.

Modifying and saving printer settings in a .pc3 file


Each .pc3 file stores the following printer settings, depending on the destination printer or file:

- Standard paper sizes and margins
- Graphics settings for vectors, rasters, and text
- Custom properties for colors, scaling, layers, fonts, and more

Paper sizes and margin settings are customizable and saved in .pmp files, not .pc3 files.

When you customize paper sizes and margins, a .pmp file is saved and attached to the .pc3 file automatically. You can also attach an existing .pmp file. For more details, see “Working with paper sizes and .pmp files” on page 645.

To modify and save printer settings in a .pc3 file

- 1 Do one of the following to choose Print (- On the ribbon, choose the Application button then choose Print, or choose Output > Print (in Print).
- Choose File > Print.
- On the Standard toolbar, click the Print tool.
- Type *print* and then press Enter.
- 2 From the Printer Name list, select a .pc3 file.
- 3 Click Properties.
- 4 In the Printer Configuration Editor dialog box, click the Settings tab.
- 5 Expand Media in the list to view and select the default paper size for the printer. Select Source and Size, then in Media Source and Size, select a new default paper size for the printer configuration.
- 6 Expand Graphics in the list and choose from the following options:
 - **Vector Graphics** — Displays settings for printing vector graphics. In Color Depth, select whether to use color or monochrome output and then the color or grayscale bit depth.
 - **Raster Graphics** — Displays settings for printing raster graphics. When printer resources are limited, you can move the sliders to balance printing speed and image quality for raster images, OLE objects, and general trade-offs.
 - **TrueType Text** — Select whether to print text that is assigned TrueType fonts as text (faster but may print with a different font) or graphics (slower but ensures text is printed as displayed).
 - **Merge Control** — To retain the colors of lines that overlap, choose Lines Overwrite. The visible color is of the line that was most recently added to the drawing. To merge colors together, choose Lines Merge.

- 7 Select Custom Properties in the list to specify additional settings for printing, which vary according to the selected printer:
 - **Standard printers and drivers** — The standard dialog box opens for your printer.
 - **DWF .pc3 files** — The Custom Properties dialog box opens. Click [?] for more details about each available option.
 - **PDF .pc3 files** — The Custom Properties dialog box opens. Click [?] for more details about each available option.
 - **JPG and PNG .pc3 files** — In Custom Properties, select the background color and mark whether to rotate raster scan lines by 90 degrees.
- 8 Click Save As.
- 9 Specify a name and location for the .pc3 file, which can be new or the existing .pc3 file, then click Save.
- 10 Click OK.
- 11 Click OK to print, or click Cancel.

You can also modify and save printer configuration files using the Options command. Choose *Options*, click the *Printing* tab, click *Add or Configure Printers*, then make your selections. For more details, see “Setting up printer configuration files” on page 859.


Working with paper sizes and .pmp files

Each .pc3 file stores a list of standard paper sizes and margins, which determine the printable area. If your print jobs often require different settings, you can create your own paper sizes with margins.

User-defined paper sizes and margins are stored in a PMP (Plotter Model Parameter) format file. You can attach a .pmp file to any .pc3 file, allowing you to reuse the paper size definitions that you create for multiple .pc3 files and on multiple computers. Each .pmp file includes the following:

- Modifications you make to standard paper sizes
- Custom paper sizes
- Calibration settings, if your .pmp file was created with another CAD program

To modify the margins of a standard paper size

- 1 Do one of the following to choose Print (- On the ribbon, choose the Application button then choose Print, or choose Output > Print (in Print).
- Choose File > Print.
- On the Standard toolbar, click the Print tool.
- Type *print* and then press Enter.
- 2 From the Printer Name list, select a .pc3 file.
- 3 Click Properties.
- 4 In the Printer Configuration Editor dialog box, click the Settings tab.
- 5 Expand User-defined Paper Sizes in the list, then choose Standard Paper Sizes (Printable Area).
A list displays the standard paper sizes defined for the .pc3 file.
- 6 Select the standard paper size you want to modify with new margins.
- 7 Click Edit.
- 8 Enter new values for the top, bottom, left, and right margins, which determine the printable area of the paper, then click Next.
- 9 Enter a name for the .pmp file where the custom settings will be saved, then click Next.
If a .pmp file is already attached to the .pc3 file in the Printer Configuration Editor dialog box, you are not prompted for a .pmp file name; the modified settings are stored automatically in the attached .pmp file.
- 10 Click Finish.
If a .pmp file was not previously attached to the .pc3 file, a .pmp file is created and attached automatically.

To create a custom paper size

- 1 In the Printer Configuration Editor dialog box, click the Settings tab.
- 2 Expand User-defined Paper Sizes in the list, then choose Custom Paper Sizes.
- 3 Click Add.
- 4 Choose whether to create an entirely new paper size or to start with an existing paper size that you select, then click Next.
- 5 Enter the width and height of the paper, in millimeters or inches, then click Next.
- 6 Enter values for the top, bottom, left, and right margins, which determines the printable area of the paper, then click Next.
- 7 Enter a name for the paper size that will display in the list of custom paper sizes, then click Next.

- 8 Enter a .pmp file name where the paper size is to be saved, then click Next.
If a .pmp file is already attached to the .pc3 file in the Printer Configuration Editor dialog box, you are not prompted for a .pmp file name; the new paper size is stored automatically in the attached .pmp file.
- 9 Click Finish.
If a .pmp file was not previously attached to the .pc3 file, a .pmp file is created and attached automatically.

You can modify and delete custom paper sizes.

To modify a custom paper size, select it in the Manage Custom Paper Sizes list, then click Edit and follow the prompts. To delete a custom paper size, select it in the list and click Delete.

To attach a .pmp file to a .pc3 file

- 1 In the Printer Configuration Editor dialog box, click the Settings tab.
- 2 Expand User-defined Paper Sizes in the list, then choose PMP File Name.
- 3 Click Attach PMP.
- 4 Select a .pmp file.
- 5 Click Open.
The name and path of the attached .pmp file displays for the PMP File Name in the list.

To save and attach user-defined paper sizes in a new .pmp file

- 1 In the Printer Configuration Editor dialog box, click the Settings tab.
- 2 Expand User-defined Paper Sizes in the list, then choose PMP File Name.
- 3 Click Save PMP.
- 4 Specify a name for the .pmp file.
- 5 Click Save.

The new .pmp file is automatically attached to the .pc3 file.

To detach a .pmp file from a .pc3 file

- 1 In the Printer Configuration Editor dialog box, click the Settings tab.
- 2 Expand User-defined Paper Sizes in the list, then choose PMP File Name.
- 3 Click Detach PMP.

Modifications to standard paper sizes and all custom paper sizes are removed from the Printer Configuration Editor dialog box.

Using plotter drivers

To print your drawing, the program sends the output to any printer driver in up to 256 colors, but with no width specified. Initially, the printed output will have a uniform fine width that is the finest line that the plotter device can produce.

The driver then passes colored vectors to the printer, which creates color output on color printers and grayscale output on laser printers. (Color output that converts to grayscale on a laser printer usually is unacceptable by CAD users for final printing.)

With print style tables, you can map all colors to black and set all lineweights to a width you choose. You must use a value appropriate to your printing capabilities. With these features, you can meet most non-presentation print needs.

Using print styles

PTCAD uses print styles to change the appearance of your printed drawing without modifying the actual entities in your drawing. Assigning print styles allows you to customize the color, pen width, linetype, and lineweight that are used to print your drawing.

Print styles help you control what your drawing looks like when it is printed. Rather than describe what an entity looks like on your screen, print styles describe what an entity will look like when you print it. For example, you can map all yellow entities in your drawing to print in blue without modifying the actual entities. You could also map all yellow entities to print with whatever lineweight, linetype, or pen width that you specify.

Because print styles are saved in print style tables, which are files located on your computer, disk, or server, you can reuse them to help eliminate the need to reconfigure your print settings each time you print a drawing. For example, you may have multiple clients who have their own printing preferences. You can save print styles in a named file for each of your clients. You can even share the file with co-workers, or store the files on a network to ensure that everyone in your office uses the same standards.

Understanding print style tables

A print style table is a collection of print styles that allows you to change the appearance of your printed drawing without modifying the actual entities in your drawing. Each print style table is saved in a file that can be located on your computer, disk, or server.

A drawing can use one type of print style table at a time. There are two types of print style tables:

- **Color-dependent print style tables (CTB)** contain a collection of print styles based on each of the 255 index colors available in a drawing. True colors and color books are not applicable to color-dependent print style tables.
- **Named print style tables (STB)** contain a collection of print styles that you define. They can vary regardless of color.

With color-dependent print style tables, you cannot assign print styles to individual entities or layers. To use these print styles, you assign a specific color to an entity or layer. When you specify a color-dependent print style table at printing time, the entity colors and layer colors map to color-based print styles in the print style table that you specify.

With named print style tables, you can assign named print styles to individual entities and layers. Entities and layers assigned print styles are printed according to the print style table that you specify at printing time. If you specify a print style for a specific entity, that print style overrides any print style assigned to the layer on which the entity resides.

Sometimes a named print style assigned to an entity or layer is not located in the print style table that is assigned to a layout or drawing. This can happen if the print style has been deleted from the named print style table or if you assign a different named print style table to the drawing that does not contain the named print style. In this case, the entity is printed using its default properties, which is similar to assigning the Normal print style to an entity or layer. If you plan on interchanging named print style tables within the same drawing, it's a good idea to coordinate the tables to use the same print style names.

If a new drawing is based on a template, the new drawing uses the same type of print style table as the template. If a new drawing is created without a template, the type of print style table is specified in the New Drawing Wizard; by default, the new drawing uses the print style table type specified in Tools > Options on the Printing tab. Every drawing is designed to use print style tables, but you decide whether to implement them.

Comparison of print style table types

	Color-dependent print style table (CTB)	Named print style table (STB)
Description	Contains pre-defined print styles according to color; there is one print style for each of the 255 index colors available in the drawing. Entities with the same color are printed the same way.	Contains unique print styles that you create. Entities with the same color can have different print settings.
Example	All blue entities print with a .5 millimeter lineweight.	One entity prints with a .7 millimeter lineweight; a second blue prints with a .5 millimeter lineweight.
Number of print styles	255 (fixed).	At least one (varies).
Print style names	Print style names range from "Color_1" to "Color_255". You cannot rename print styles.	You define new print style names. You can rename all print styles except the Normal print style.
Add, delete, and modify print styles	You can modify the existing print styles, but you cannot add or delete print styles.	You can add and delete print styles. You can modify all print styles except the Normal print style, which uses the default characteristics of the entity.
Create additional tables	Yes.	Yes.
Assignment	Current print style: New entities always assigned BYCOLOR.	Current print style: Assigned to new entities.
	Print styles: Assigned to colors in the print style table.	Print styles: Assigned to entities and layers.
	Print style tables: Assigned to a Layout tab or the Model tab.	Print style tables: Assigned to a Layout tab or the Model tab.
Legacy file support	You can import existing printer configuration files (PCP) files into the print style table. CTB files are similar to PCP files primarily used in previous versions of PTCAD.	Not applicable.

Implementing print style tables

Every drawing is designed to use print style tables, but you decide whether to implement them. Even if you use one of the default print style tables available with PTCAD, using print style tables requires planning ahead of time to ensure that your drawing prints as planned.

For example, a single drawing of a floor plan might require the printing of the following drawing sheets:

- **Main Floor Plan** Walls print with thick, black lines.
- **Electrical Plan** Walls print with normal gray lines, indicating that they are not the focus.
- **HVAC Plan** Walls print with normal gray lines, indicating that they are not the focus.
- **Roof Plan** Walls print with thin, gray lines and a hidden linetype, indicating that they are hidden under the roof in a plan view.

In this example, you can create four named print style tables, each containing a print style named “WallPstyle”. Each print style table contains its own settings for “WallPstyle” to control how the walls print. Assign WallPstyle to either the wall entities or to a wall layer. Then, assign a different named print style table each time you print, or create four layouts and assign a different print style table to each layout.

The following table describes, in order, the steps to get you started using both color-dependent and named print style tables.

Getting started using print style tables

Color-dependent	Named	Task	Command	Where to get details
X	X	Create a new drawing. Select a drawing template that uses the desired print style table type or choose it in the New Drawing Wizard.	File > New	Creating a new drawing, page 46
X	X	(Optional) Create a new print style table.	File > Print Styles Manager	Creating new print style tables, page 653
X		Assign colors to entities and layers that correspond with print style table settings.	Modify > Properties Tools > Explorer	Setting the current entity color, page 52; Modifying the properties of entities, page 406; Setting the layer color, page 294
	X	Set the current print style assigned to new entities.	Tools > Drawing Settings > Entity Creation tab; status bar; <i>printstyle</i>	Setting the current print style, page 57
	X	Assign print styles to entities.	Modify > Properties; Entity Properties toolbar; <i>printstyle</i>	Modifying the properties of entities, page 406
	X	Use the Explorer to assign print styles to layers.	Tools > Explorer	Setting the layer print style, page 299
X	X	Assign a print style table to the Model tab, a layout, or to all layouts in the drawing.	File > Print > Advanced tab	Assigning print style tables, page 652
X	X	(Optional) Make changes to the assigned print style table.	File > Print Styles Manager File > Print > Advanced tab	Modifying print style tables, page 654
X	X	Print the drawing.	File > Print	Printing or plotting your drawing, page 660

The following table describes how to further customize how print styles work within your drawings.

Further customizing print style tables

Task	Command	Where to get details
Copy, rename, or delete print style tables.	File > Print Styles Manager	Copying, renaming, or deleting print style tables, page 657
Change a drawing to use another type of print style table, for example, change a drawing that uses color-dependent print style tables to one that uses named print style tables.	<i>convertpstyles</i>	Changing the print style table type of a drawing, page 657
Convert a color-dependent print style table to a named print style table.	<i>convertctb</i>	Converting print style tables, page 658
Change the default location where print style tables are stored.	Tools > Options > Paths/Files tab	Changing the options on the Paths/Files tab, page 829
Customize how print styles work with new drawings that you create and older drawings that you open.	Tools > Options > Printing tab	Changing the options on the Printing tab, page 855


Assigning print style tables

Select a print style table before printing if you want to change how your drawing appears when you print it. Print style tables can modify how colors, pen widths, line-types, and lineweights look when they are printed.

You can assign print style tables globally for all layouts (including the Model tab), or individually for the Model tab or a Layout tab. Assigning a print style table to an individual layout allows you to further customize the layouts you use to print a drawing.

However, assigning different named print style tables to various layouts may result in mismatched print style names; a named print style assigned to an entity or layer may not be located in the assigned print style table at print time. In this case, entities are printed using their default properties, which is similar to assigning the Normal print style to an entity or layer.

To assign a print style table at the same time you print

- 1 If necessary, click the desired Layout tab, or click the Model tab.
- 2 Do one of the following to choose Print (- On the ribbon, choose the Application button then choose Print, or choose Output > Print (in Print).
- On the menu, choose File > Print.
- Type *print* and then press Enter.

You can assign a print style table to a page setup.

Use the Page Setup Manager to assign a print style table to a page setup that you use for printing. For more details, see [Specifying pen and line printing options](#).

- 3 Under Print Style Table (Pen Assignments), select a print style table that you created or one of the following:
 - **None** — Applies no print style table. Entities print according to their own properties.
 - **Icad** — Uses the default print style table and its color assignments.
 - **Monochrome** — Prints all colors as black.
- 4 Select Save Changes to Layout, and then click OK.


When a drawing is first created it is assigned to use either color-dependent or named print style tables.

For details on converting a drawing to use a different type of print style table, see “[Understanding print style tables](#)” on page 649 in this chapter.

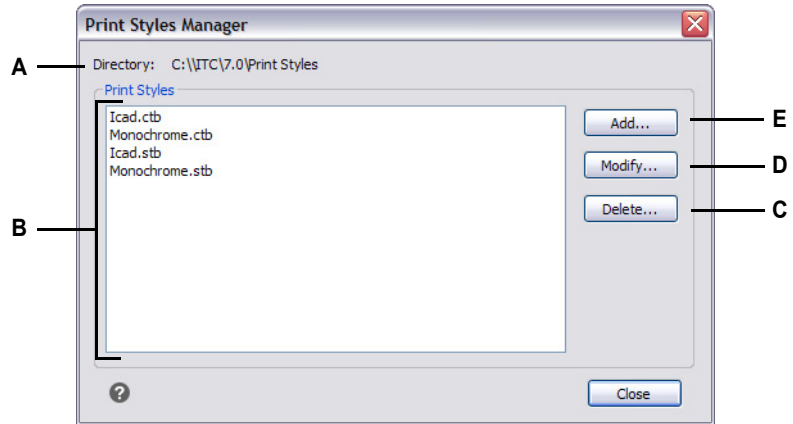
Creating new print style tables

PTCAD offers several print style tables to help you get started. If you want to customize your print output further, you can create your own print style table. You can create a new print style table entirely from scratch, based on PTCAD registry settings, or by importing a printer configuration file (PCP file).

To create new print style tables

- 1 Do one of the following to choose Print Styles Manager (- On the ribbon, choose the Application button then choose Print > Print Styles Manager, or choose Output > Print Styles Manager (in Print).
- On the menu, choose File > Print Styles Manager or choose Format > Print Styles Manager.
- On the Format toolbar, click the Print Styles Manager tool.
- Type *stylesmanager* and then press Enter.
- 2 Click Add.
- 3 Complete the setup wizard.

On the last wizard page, you can click Print Style Table Editor to set up the print styles for the table. For more details about the Print Style Table Editor options, see the next section.



- A** Displays the location where print style tables are located.
- B** Displays a list of all print style tables available in the named folder.
- C** Click to delete the currently selected print style table.
- D** Click to modify the settings for the currently selected print style table.
- E** Click to create a new print style table.

Modifying print style tables

When your drawing was created, it was set up to use color-dependent or named print style tables:

- **Color-dependent print style tables (.ctb files)** — You can modify individual print styles within the table, but you cannot add, rename, or delete print styles. Color-dependent print style tables always have 255 print styles, each named for a specific color. Your changes affect all entities and layers assigned that color.
- **Named print style tables (.stb files)** — You can add, modify, rename, and delete individual print styles within the table. However, you cannot modify, rename, or delete the Normal print style. Your changes affect all entities and layers that are assigned that print style name.

Use a system variable to determine the type of print style table your drawing uses.

If you can't remember what type of print style table is assigned to your drawing, use the `PSTYLEMODE` system variable to determine the print style table type.


Each print style within a print style table specifies a color, pen numbers, linetype, and lineweight. PTCAD recognizes additional characteristics for compatibility with AutoCAD only, including: dither, grayscale, screening, adaptive, line end style, line join style, and fill style.

When specifying print style characteristics, be sure to consider the limitations of your output device.

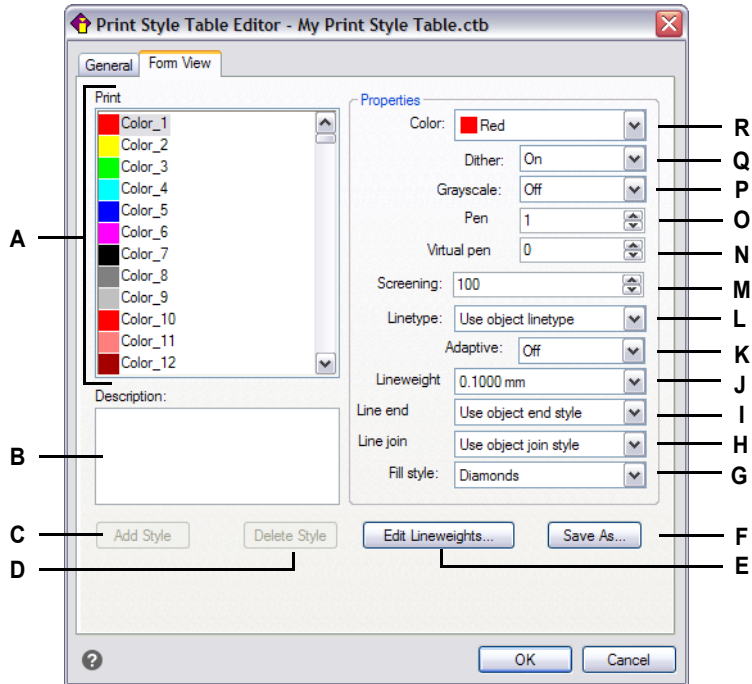
It is recommended that you only modify print style tables that you have created.

If you modify a default print style table that came with PTCAD, you overwrite the original information which is then lost.

To modify print style tables

- 1 Do one of the following to choose Print Styles Manager (- On the ribbon, choose the Application button then choose Print > Print Styles Manager, or choose Output > Print Styles Manager (in Print).
- On the menu, choose File > Print Styles Manager or choose Format > Print Styles Manager.
- On the Format toolbar, click the Print Styles Manager tool.
- Type *stylesmanager* and then press Enter.
- 2 Select the print style table you want to modify.
- 3 Click Modify.
- 4 Click the General tab, and then do any of the following:
 - Enter a new table description.
 - Select Apply Global Scale Factor to Non-ISO Linetypes to apply the scale factor to non-ISO linetypes used for any print style in the current print style table. This also applies to fill patterns, which are not used in PTCAD, but are recognized for compatibility with AutoCAD.
 - Enter a scale factor to apply to non-ISO linetypes used for any print style in the current print style table.
- 5 Click the Form View tab, and then do any of the following:
 - Make format changes to a print style by selecting it in the Print Styles list, then make color, pen map, linetype, or lineweight changes for the print style. Your changes are saved automatically for the selected print style.
 - Add a new print style by clicking Add Style. Enter a new name, and then click OK. Select the options for the print style. (Available for named print styles only.)
 - Rename a print style by selecting it in the Print Styles list. Single-click the print style again, and then enter a new name. (Available for named print styles only.)
 - Delete a print style by selecting it in the Print Styles list. Click Delete Style. (Available for named print styles only.)

6 Click OK.




- A Select a print style to modify it.
- B Enter a description for the selected print style.
- C Click to create a new print style. (Named print style tables only.)
- D Click to delete the selected print style. (Named print style tables only.)
- E Click to modify the list of available lineweights for the current print style table.
- F Click to save the print style table with a new name or in a new location.
- G Choose a fill style for the selected print style.
- H Choose a line join style for the selected print style.
- I Choose a line end style for the selected print style.
- J Choose a lineweight for the selected print style.
- K Choose whether to adjust the linetype scale automatically to complete the linetype pattern when necessary.
- L Choose a linetype for the selected print style.
- M Select the level of color intensity for the selected print style: 100 is full intensity, 0 is white. (Dither must be on.)
- N Type or scroll to the width of the virtual pen for the selected print style (for printers that don't have physical pens, such as laser or inkjet printers).
- O Type or scroll to the width of the physical pen for the selected print style.
- P Choose whether to print the selected print style in grayscale.
- Q Choose whether to turn on dithering for the selected print style.
- R Choose a color for the selected print style.

Copying, renaming, or deleting print style tables

Copy, rename, or delete a print style table just as you would any other file on your computer. Regardless of which print style table type your drawing uses, you can use the Print Style Manager to manage both color-dependent and named print style tables.

To delete print style tables

- 1 Do one of the following to choose Print Styles Manager ():
 - On the ribbon, choose the Application button then choose Print > Print Styles Manager, or choose Output > Print Styles Manager (in Print).
 - On the menu, choose File > Print Styles Manager or choose Format > Print Styles Manager.
 - On the Format toolbar, click the Print Styles Manager tool.
 - Type *stylesmanager* and then press Enter.
- 2 Select a print style table, then click Delete.
Color-dependent print style tables are .ctb files and named print style tables are .stb files.

To copy, rename, or delete print style tables

- 1 On your computer, open the folder that stores print style tables, for example, `\CompanyName\ProductVersion\Print Styles`.
- 2 Copy, rename, or delete the print style table just as you would any other file on your computer.

Changing the print style table type of a drawing

When your drawing was created, it was set up to use color-dependent print style tables (.ctb files) or named print style tables (.stb files). A drawing can use one type of print style table at a time. If necessary, after a drawing is created you can convert the drawing to use the other type of print style table.

If you can't remember what type of print style table is assigned to your drawing, use the PSTYLEMODE system variable to determine the print style table type.

To change a drawing to use named print style tables

- 1 If you want to reuse any of the existing print style information, convert your color-dependent print style tables to named print style tables. For details, see “To convert a color-dependent print style table to a named print style table” on page 659 in this chapter.
- 2 Open the drawing that uses color-dependent print style tables (.ctb files).
- 3 Type *convertstyles*, and then press Enter.

- 4 If you have already converted your individual color-dependent print style tables to named tables, click OK in the prompt that displays.
If you have not converted the tables, click Cancel. First use *convertctb* to convert your color-dependent print style tables to named tables. If you do not, all of the print style information you specified in your drawing will be lost.
- 5 Select a named print style table (.stb file) that you want to use with the drawing.
- 6 Click Open.

Errors will occur if you have not converted print style tables.

If you have not converted a color-dependent print style table to a named print style table, you will be warned that the table you have selected does not contain color mapping and the drawing cannot be converted.

To change a drawing to use color-dependent print style tables

- 1 Open the drawing that uses named print style tables (.stb files).
- 2 Type *convertstyles*, and then press Enter.
- 3 If you are sure you want to convert the drawing and lose all print style assignments, click OK in the prompt that displays.
If you do not want to lose the print style assignments, click Cancel.

Converting a drawing to use color-dependent print style tables will remove all of the named print style information from entities and layers.

However, the named print style tables are not deleted from your computer.

Converting print style tables

You can convert a color-dependent print style table to a named print style table. You cannot convert a named print style table to a color-dependent print style table because color-dependent tables contain only print styles that are named after the 255 colors to which they map.

Converting a color-dependent print style table to a named print style table can be helpful in the following situations:

- You don't want to create a named print style table from scratch.
- You want to create a named print style table that has the same settings as a color-dependent print style table, but with some new print styles or other custom settings.
- You want to convert a drawing to use named print style tables and you want to reuse most of print styles already defined in a color-dependent print style table.

To convert a color-dependent print style table to a named print style table


- 1 Type *convertctb*, and then press Enter.
- 2 Select the color-dependent print style table (.ctb file) that you want to convert.
- 3 Click Open.
- 4 Enter a name for the new named print style table (.stb file).
- 5 Click Save.

The print styles in the new table are named Style 1, Style 2, and so on. If you want to use different print style names, rename the print styles before you assign them to entities and layers in your drawing. If you rename the print styles after assigning them, they will not match when you print your drawing. For information about renaming print styles, see “Modifying print style tables” on page 654 in this chapter.

Turning print style tables on or off

When you turn off print style tables, entities print according to their own properties. However, all of the print style information is saved so you can easily turn on print styles again. Actual print style table files are not deleted, and for drawings that use named print style tables, entities and layers retain their assigned print styles.

To turn off print style tables

- 1 Click the Layout tab or Model tab for which you want to turn on or off print style tables.
- 2 Do one of the following to choose Page Setup Manager (- On the ribbon, choose the Application button then choose Page Setup Manager, or choose Output > Page Setup Manager (in Print).
- On the menu, choose File > Page Setup Manager.
- On the Format toolbar, click the Page Setup Manager tool.
- Type *pagesetup* and then press Enter.
- 3 Select the desired page setup, then click Modify.
- 4 In Print style table, select a print style table to turn on print style tables, or select None to turn off print style tables.
- 5 Click OK.
- 6 Click OK.

Printing or plotting your drawing

After you have configured your drawing and any layouts for printing, you are ready to print. If desired, you can preview your page before printing.

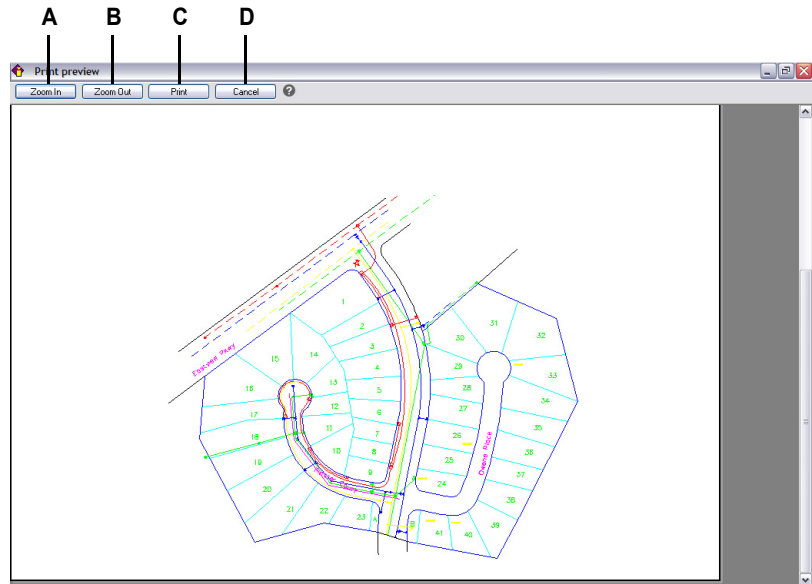
Previewing a drawing before printing

Viewing a drawing before printing gives you a preview of what your drawing will look like when it is printed. This helps you see if there are any changes you want to make before actually printing the drawing.

If you are using print style tables, the preview shows how your drawing will print with the assigned print styles. For example, the preview may display different colors or lineweights than those used in the drawing because of assigned print styles.

To preview a drawing before printing

- 1 If necessary, click the desired Layout tab or the Model tab.
- 2 Do one of the following to choose Print Preview (



- A** Click to zoom in.
- B** Click to zoom out. If you have zoomed in several times, click multiple times to restore the entire preview image.
- C** Click to print the drawing.
- D** Click to close the preview and return to the drawing.


Printing a drawing

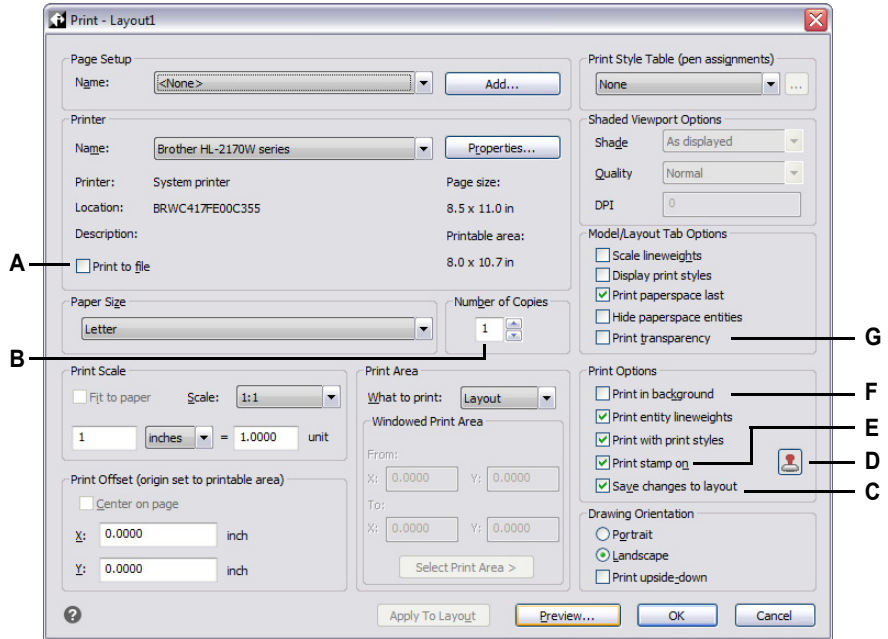
The Print dialog box has print settings described in the previous sections for page setups.

You cannot print a rendered image directly to a printer.

To print a rendered image, you must first save the drawing to a different format and then print it from another graphics program.

To print a drawing

- 1 If necessary, click the desired Layout tab or the Model tab.
- 2 Do one of the following to choose Print (- On the ribbon, choose the Application button then choose Print, or choose Output > Print (in Print).
- On the menu, choose File > Print.
- On the Standard toolbar, click the Print tool.
- Type *print* and then press Enter.
- 3 In Page Setup Name, select the page setup to apply for printing. The print options in the Print dialog box change to reflect the settings of the selected page setup. For details about importing page setups, see “Importing page setups from a different drawing” on page 634 in this chapter.
- 4 Make any necessary adjustments, including for the following options that are available only at print time (not when setting up a page setup):
 - Print to file — Select to print to a file instead of a printer.
 - Number of copies — Enter the number of copies to print.
 - Print transparency — Select to print entities with their assigned transparency. Note that turning on this option can slow printing. Also note that the PLOTTRANSPARENCYOVERRIDE system variable can override transparency print settings for all layout viewports at once.
 - Print in background — Print in the background of other tasks being performed by the computer.
 - Print stamp on — Select to print with a header and footer. When enabled, you can click [...] to modify the text of the print stamp.
 - Save changes to layout — Select to save the print settings for the model or layout.
- 5 Click Print.



- A** Select to print to a file instead of a printer.
- B** Enter the number of copies to print.
- C** Select to save the print settings for the model or layout.
- D** Click to modify the text of the print stamp (available if print stamps are turned on).
- E** Select to print with a header and footer.
- F** Select to print in the background of other tasks being performed by the computer.
- G** Select to print entities with their assigned transparency.


Saving print settings for a model or layout

All print settings can be saved with a model or layout, which can be particularly helpful if you don't use page setups. The next time you print the model or layout, the saved print settings automatically load in the Print dialog box.

Saved print settings are also used when publishing.

If you're publishing drawings using sheets that don't have a page setup assigned, the print settings saved for the model or layout are used by default.

To save print settings with a model or layout

- 1 Click the desired Layout tab or the Model tab for which you want to save print settings.
- 2 Do one of the following to choose Print (- On the ribbon, choose the Application button then choose Print, or choose Output > Print (in Print).
- On the menu, choose File > Print.
- On the Standard toolbar, click the Print tool.
- Type *print* and then press Enter.
- 3 Make your print selections.
- 4 Click Apply to Layout.
- 5 Do one of the following:
 - Click OK to print and close the dialog box.
 - Click Cancel to close the dialog box without printing.

Publishing sheet sets

With sheets sets, you can collect and organize sheets (layouts) from various drawings in one location, which makes it easy to open, view, print, and distribute numerous layouts that are related to a single project. You can publish sheet sets to a printer or plotter and to the following file formats: PDF, DWF, and DWFx.

Sheet sets are saved in .dst files and are managed and published using the Sheet Set Manager pane. If you move a .dst file to another computer, its referenced .dwg files should also be moved.


Some projects benefit from using sheet set information in fields that are inserted in the drawing. For example, you can include the current sheet name and description in fields, which update automatically when using the Update Field command. For more details, see “Working with fields” on page 486.

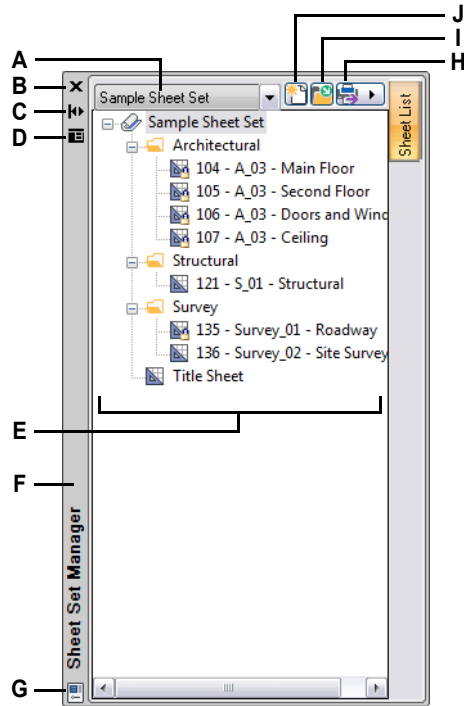
Sheet sets are typically used for large multi-drawing projects, but another way to publish drawings is using sheet lists which are saved in .dsd file and published using the Publish command. For more details, see “Publishing sheet lists” on page 674 in this chapter.

Displaying the Sheet Set Manager pane

The Sheet Set Manager pane is used to collect, organize, and publish collections of sheets (layouts) that are located in multiple drawings.

To display or hide the Sheet Set Manager pane

- 1 Do one of the following to choose Sheet Set Manager :
 - On the ribbon, choose the Application button, then choose Sheet Set Manager; choose Output > Sheet Set Manager (in Print); or choose View > Sheet Set Manager (in Panes).
 - On the menu, choose File > Sheet Set Manager or View > Display > Sheet Set Manager.
 - Type *sheetset* and then press Enter.




- A** Select a sheet set from the list of opened sheet sets.
- B** Click to close the pane.
- C** Click to automatically hide the pane.
- D** Click to display a menu of options: close, dockable, fix to left, fix to right, autohide, and transparency.
- E** Displays the sheets and subsets of the sheet set. Drag-and-drop sheets and subsets to reorder them.
- F** Click and drag to move the pane to a new location. Right-click for more options.
- G** Double-click to dock the pane. Right-click for more options.
- H** Click to publish the displayed sheet set.
- I** Click to open a sheet set.
- J** Click to create a sheet set.


Creating a sheet set

You can create a sheet set (.dst file) by selecting drawings that contain the sheets (layouts) that you want to include. If you have an existing sheet set that contains the general structure you need, you can use the existing sheet set as a template for the new one.

To create a new sheet set using layouts from existing drawings

- 1 Do one of the following:
 - In the Sheet Set Manager pane, click New ().
 - Type *newsheetset*, then press Enter.
- 2 Select Existing Drawings, then click Next.
- 3 Enter the name, optional description, and folder for the sheet set, then click Next.
- 4 Click Import Options and do the following:
 - To automatically add the drawing file name as a prefix to sheet titles, mark Prefix Sheet Titles with File Name.
 - To create and organize subsets for the sheet set based on the folder hierarchy on your computer, mark Create Subsets Based on Folder Structure. If marked, choose whether to omit the top folder when creating the subsets in the resulting sheet set structure.
 - Click OK.
- 5 Click Browse, select the folder that contains drawings with layouts you want to use for sheets, then click OK.
- 6 Uncheck any drawings or layouts you don't want added to the sheet set, then click Next.
- 7 Review the details, then click Finish.


To create a new sheet set based on a template

- 1 Do one of the following:
 - In the Sheet Set Manager pane, click New ().
 - Type *newsheetset*, then press Enter.
- 2 Select An Example Sheet Set, then click Next.
- 3 Specify the path and name of the sheet set you want to use as a template or click Browse to locate it, then click Next.
- 4 Enter the name, optional description, and folder for the sheet set, then click Next.
- 5 Review the details, then click Finish.

Opening a sheet set

When you open a sheet set (.dst file), its sheets and subsets display in the Sheet Set Manager pane.

To open a sheet set

- 1 Do one of the following:
 - In the Sheet Set Manager pane, click Open ().
 - Type *opensheetset*, then press Enter.
- 2 Choose the folder containing the desired sheet set (.dst) file.
- 3 Do one of the following:
 - Choose the file you want to open, then click Open.
 - Double-click the file you want to open.

Close a sheet set.

To close a sheet set but keep the Sheet Set Manager pane open, right-click the name of the sheet set and choose Close Sheet Set.

Modifying sheet set properties

You can change a sheet set's name, description, file path, and more.

To change the properties of a sheet set

- 1 In the Sheet Set Manager pane, right-click the sheet set, then choose Properties.
- 2 Make your selections. Click [?] for more details about each option.
- 3 Click OK.

Working with sheets

Each sheet in a sheet set corresponds to a single layout in a drawing. You can open a sheet's corresponding layout directly from the Sheet Set Manager pane, add sheets, reorganize sheets, and change the properties of sheets.

Opening a sheet

Open any sheet in a sheet set by simply double-clicking it in the Sheet Set Manager pane. As long as the referenced drawing is found and the layout exists, PTCAD opens the drawing and activates the layout.

To open a sheet in the program

- 1 In the Sheet Set Manager pane, do one of the following:
 - Double-click a sheet.
 - Right-click a sheet and choose Open. If Open is unavailable, the referenced drawing is not found.

The sheet opens in the program.

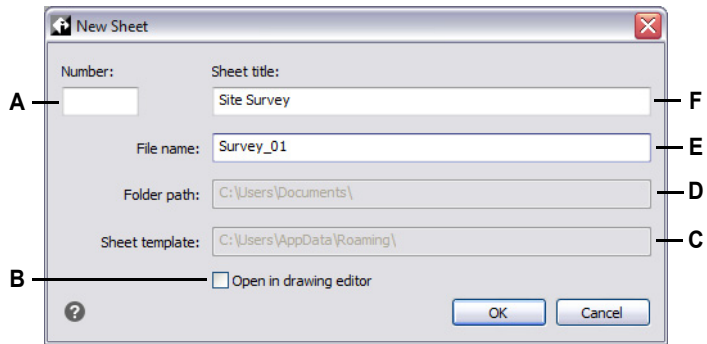
- 2 If the sheet cannot be found, choose one of the following options:
 - **Locate drawing** Allows you to navigate to the folder where the referenced drawing is located and choose the layout.
 - **Remove the sheet** Deletes the sheet from the sheet set.
 - **Associate a different layout** Allows you to navigate to the folder where the referenced drawing is located and choose a different layout.

Adding sheets to a sheet set

From the Sheet Set Manager pane with a sheet set open, you can add existing sheets (layouts) from a drawing or create and add new ones.

To create a new sheet and add it to a sheet set

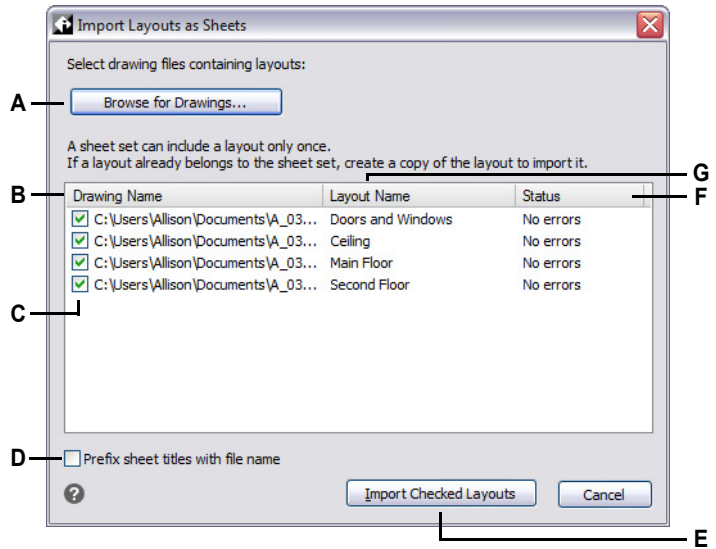
- 1 In the Sheet Set Manager pane, right-click the item located above where the new sheet is to be added, then choose New Sheet.
- 2 Make your selections.
- 3 Click OK.



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| <p>A Indicates the sheet number. It is recommended that you don't change the sheet number until after you've create the sheet.</p> <p>B Mark to open the new sheet in the drawing window after you click OK.</p> <p>C Displays the default template that will be used for the new sheet.</p> | <p>D Displays the default folder path where the new drawing will be stored.</p> <p>E Enter the name of the new .dwg file to create for this sheet.</p> <p>F Enter the name of the new layout you want to create for this sheet.</p> |
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To add a sheet by selecting one or more existing layouts

- 1 In the Sheet Set Manager pane, right-click the item located above where the new sheet is to be added, then choose Import Layout as Sheet.
- 2 Click Browse and select the folder that contains drawings you want to browse for layouts, then click OK.
- 3 Mark each layout you want to add to the sheet set.
- 4 Click Import Checked.



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| <p>A Click to select drawings that contain layouts you want to use for sheets.</p> <p>B Displays all layouts in the selected drawings that are not already in the sheet set.</p> <p>C Mark each layout you want to import as a sheet.</p> | <p>D Click to add the drawing file name as a prefix to sheet titles.</p> <p>E Click to import all layouts that are marked.</p> <p>F Displays the status of the layout.</p> <p>G Displays the name of the layout.</p> |
|--|--|

Modifying sheets

Sheets can be renamed, renumbered, reordered, and deleted. Sheets also have properties which can be changed.

To rename or renumber a sheet

- 1 In the Sheet Set Manager pane, right-click the desired sheet, then choose Rename and Renumber.
- 2 In Title, enter a new sheet name.
- 3 In Number, enter a new sheet number.
- 4 Click OK.

To reorder sheets in a sheet set

- In the Sheet Set Manager pane, select and drag a sheet to its new location in the list.

To delete a sheet from a sheet set

- In the Sheet Set Manager pane, right-click the desired sheet, then choose Remove Sheet.

To change the properties of a sheet

- 1 In the Sheet Set Manager pane, right-click the desired sheet, then choose Properties.
- 2 Make your selections. Click [?] for more details about each option.
- 3 Click OK.

Working with subsets

Sheet sets can contain various subsets, which help organize individual sheets into categories, similar to a folder name. For example, a sheet set might have subsets named Architectural, Civil, Electrical, and Structural. Subsets can also be nested, similar to folders.

To create a subset

- 1 In the Sheet Set Manager pane, right-click the sheet set or subset, then choose New Subset.
- 2 In Name, enter the name of the subset.
- 3 In Description, enter an optional description for the subset.
- 4 In Publish Sheets in Subset, choose one of the following:
 - **Do Not Publish Sheets** Prevents sheets in this subset from being published or sent via eTransmit.
 - **Publish by Sheet “Include for Publish” Setting** Includes sheets in this subset for publishing or eTransmit according to each sheet's “Include for Print/Publish” setting.
- 5 In Prompt for Template, choose whether to prompt for a layout to use as a template when adding new sheets to the subset.
- 6 Click OK.
- 7 Drag-and-drop existing sheets to the subset, or right-click the Subset and choose New Sheet to add new sheets.

To move a subset and its contents

- In the Sheet Set Manager pane, select and drag a subset to its new location in the list.

To rename a subset or change its properties


- 1 In the Sheet Set Manager pane, right-click the desired subset, then choose Properties.
- 2 In Name, enter a new name for the subset.
- 3 Make additional selections. Click [?] for more details about each option.
- 4 Click OK.

Publishing a sheet set

You can publish sheet sets to a printer or plotter and to the following file formats: PDF, DWF, and DWFx.

Sheet sets are saved in .dst files and are managed and published using the Sheet Set Manager pane. Another way to publish drawings is using sheet lists which are saved in .dsd file and published using the Publish command. For more details, see “Publishing sheet lists” on page 674 in this chapter.

To publish a sheet set or part of it

- 1 In the Sheet Set Manager pane, do one of the following depending on what you want to publish:
 - To publish the entire sheet set, click Publish () in the Sheet Set Manager pane or right-click the sheet set name in the list.
 - To publish only the content of a subset, right-click the subset name in the list.
 - To publish only a single sheet, right-click the sheet name in the list.
- 2 If you need to specify options for publishing, choose Publish, then one of the following:
 - **Sheet Set PDF Publish Options** Allows you to specify options for publishing to a .pdf file. Click [?] to see details about each option.
 - **Sheet Set DWF Publish Options** Allows you to specify options for publishing to a .dwf or .dwfx file. Click [?] to see details about each option.
- 3 If you specified publish options, right-click the item again.
- 4 Choose Publish, then choose one of the following:
 - **Publish to PDF** Publishes to a .pdf file. In the dialog box that displays, specify the location and file name of the .pdf file, then click Save.
 - **Publish to DWF** Publishes to a .dwf file. In the dialog box that displays, specify the location and file name of the .dwf file, then click Save.
 - **Publish to DWFX** Publishes to a .dwfx file. In the dialog box that displays, specify the location and file name of the .dwfx file, then click Save.
 - **Publish to Printer** Publishes to a printer. In the dialog box that displays, specify the desired settings. Click [?] to see details about each option. When finished, click Print.

To publish a sheet set via eTransmit

- 1 In the Sheet Set Manager pane, right-click one of the following:
 - The sheet set name if you want to publish the entire sheet set.
 - A subset name if you want to publish only the content of the subset.
 - A sheet name if you want to publish only the single sheet.
- 2 Choose eTransmit.
- 3 Specify the desired options, then click OK. Click [?] to see details about each option or see “Packaging drawings and files for use on another computer” on page 809.

Publishing sheet lists

A sheet list is a collection of sheets that you want to publish using the Publish command. Each sheet in the list references a model or layout tab from any drawing.

First you create a sheet list by specifying the drawing and which of its model and layout tabs that you want to publish. For each sheet, you can assign a page setup and layer state. Save the sheet list, then publish it. You can publish sheet lists to a printer or plotter and to the following file formats: PDF, DWF, and DWFx.



Sheet lists are saved as Drawing Set Description files (.dsd files). If you move a .dsd file to another computer, its referenced .dwg files should also be moved.





Another way to publish drawings is using sheet sets which are saved in .dst files and published from the Sheet Set Manager pane. For more details, see “Publishing sheet sets” on page 665 in this chapter.

Creating a sheet list


A sheet list is saved in a .dsd file and contains references to the model and layout tabs of the drawings you want to publish. Each sheet can be assigned a page setup and layer state, and can be placed in the order you want it to appear among other sheets when published.


To create a sheet list to publish

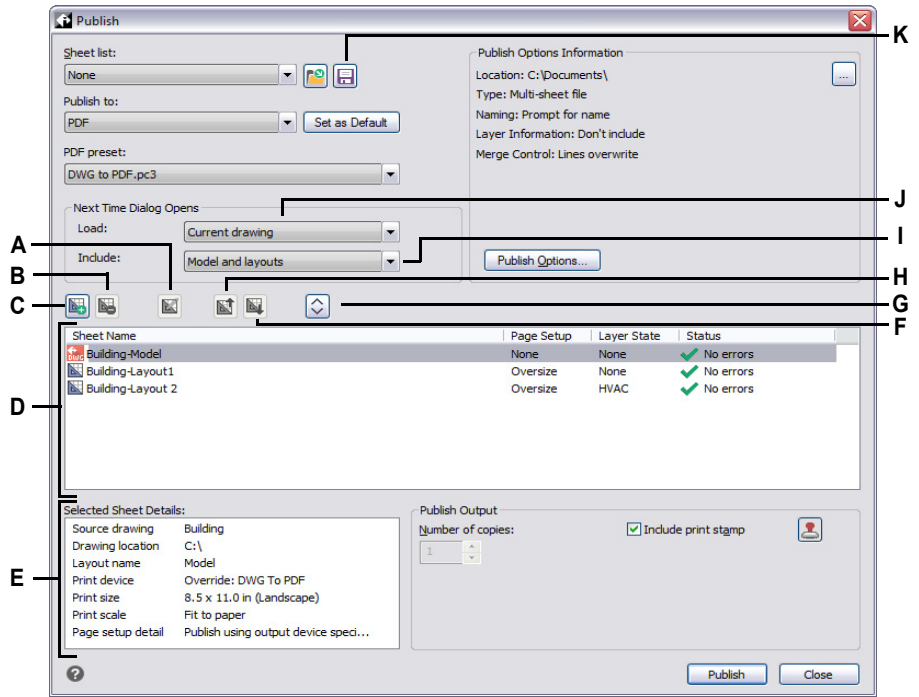
- 1 Do one of the following to choose Publish (- On the ribbon, choose the Application button, then choose Publish > Publish, or choose Output > Publish.
- On the menu, choose File > Publish.
- On the Standard toolbar, click the Publish tool.
- Type *publish* and then press Enter.
- 2 Depending on the dialog box settings, the sheet list that displays first in the dialog box might have too many or too few sheets for your needs. To quickly clear or add sheets, choose one of the following options, then click Close and open the dialog box again:
 - **Load** Select which drawing files to include when adding sheets automatically the next time the dialog box opens: all open drawings, the current drawing only, or none (empty sheet list).
 - **Include** Select which tabs to add as sheets automatically the next time the dialog box opens: model tabs only, layout tabs only, or both model and layout tabs.
- 3 Click Add () to add sheets to the list. You can also press Insert when a sheet is selected or drag drawing files from the Windows Explorer window and drop them in the list.

- 4 Choose which drawings you want to add new sheets from:
 - **Drawings already in the publish list** Select to display layouts or models that are missing from drawings that are already listed in the Publish dialog box. The list is empty if all layouts and models are already included in the sheet list.
 - **Other drawings** Select to display models and layouts from other drawings, then click Browse to select one or more drawing files.
- 5 Mark the layouts and models that you want to include in the sheet list.
- 6 Click OK.
- 7 To remove unwanted sheets from the sheet list, select a sheet, then click Remove () or press Delete.
- 8 To move sheets up or down in the list, select the sheet and click Move Up () or click Move Down (). You can also click Reverse () to reverse the order of all sheets. Sheets are published in the order in which they display.
- 9 To assign a different page setup to sheets, in the Page Setup column, select a sheet (or Ctrl+click to select multiple) and select a page setup from the list.

The page setup you select for a sheet is assigned to all other selected sheets if they have the same type (for a model or layout) and are located in the same drawing.

For details about importing page setups, see “Importing page setups from a different drawing” on page 634 in this chapter. For details about using Page Setup Manager, see “Working with page setups” on page 630 in this chapter.
- 10 To edit page setup settings, do the following:
 - Select a sheet for a model or layout in the drawing that contains the page setup you want to edit. The page setups that display in the Page Setup Manager dialog box depend on whether a model or layout sheet is currently selected.
 - Click Edit Page Setup (). You can also double-click a sheet.
 - Select the page setup to change, then click Modify.
 - Make your selections, then click OK.
 - Click Close.
- 11 To publish a sheet using a specific layer state, in the Layer State column, select a sheet (or Ctrl+click to select multiple) and select a layer state from the list. The list displays all layer states created for a drawing. Choose Layer States Manager in the list to modify and create layer states.

The layer state you select for a sheet is assigned to all other selected sheets if they are in the same drawing.
- 12 Click Save ()
- 13 In the Save DSD File dialog box, enter a name for the sheet list, then click Save.





- A** Click to edit page setup settings in the Page Setup Manager. The page setups that display for editing depend on whether the currently selected sheet is for a model or layout.
- B** Click to delete the currently selected sheet from the sheet list.
- C** Click to add sheets from drawings that you select.
- D** Displays the sheets to publish with each sheet's settings.
- E** Displays details about the currently selected sheet.
- F** Click to move the currently selected sheet down one position.
- G** Click to reverse the order of sheets.
- H** Click to move the currently selected sheet up one position.
- I** Choose whether to add sheets for the model tab, layout tabs, or both when automatically loading sheets when the dialog opens.
- J** Choose which drawings (if any) to load sheets from automatically when the dialog displays.
- K** Click to save the list of sheets in a .dsd file.

Modifying an existing sheet list

Open, or load, an existing sheet list (.dsd file) to modify it. During loading, drawings that are specified in the sheet list must be accessible in the referenced folder location.


To modify a sheet list


- 1 Do one of the following to choose Publish ().
- 3 Locate and select the .dsd file to modify.
- 4 Click Open.
- 5 Make any necessary changes. The options are the same as creating a new sheet. For details, see “Creating a sheet list” on page 674 in this chapter.
- 6 Click Save (.

Publishing a sheet list to the printer named in page setup


Publishing a sheet list prints all referenced sheets at the same time. The sheets are printed to the printer named in the assigned page setup for each sheet. If the Page Setup column indicates None for a sheet, the sheet is published using the print options specified for the layout.

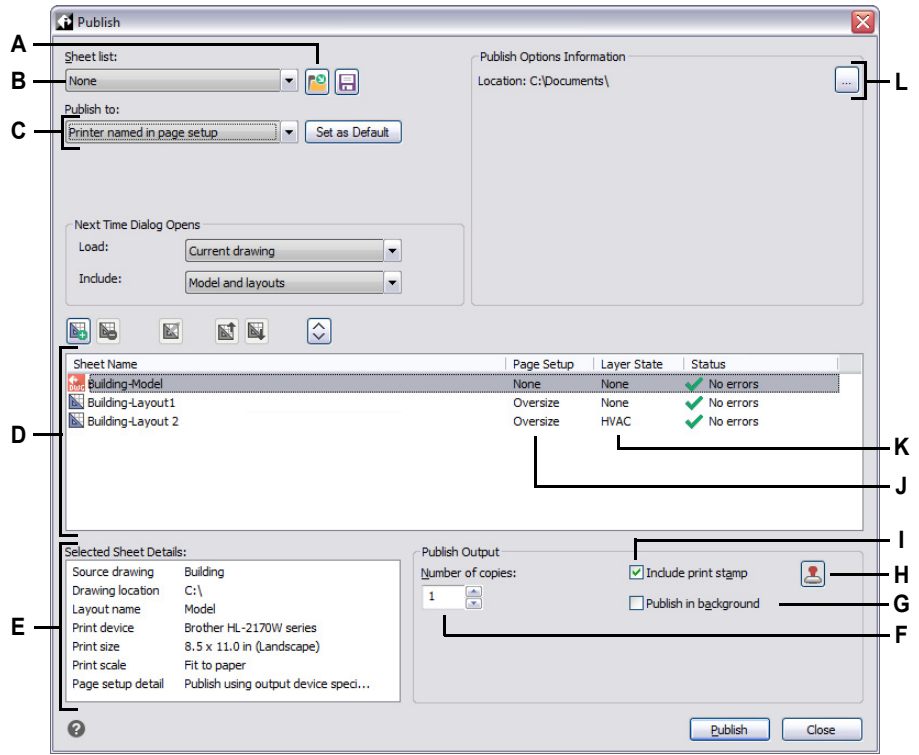
To publish a sheet list to the printer named in page setup

- 1 Do one of the following to choose Publish ():
 - On the ribbon, choose the Application button, then choose Publish > Publish, or choose Output > Publish.
 - On the menu, choose File > Publish.
 - On the Standard toolbar, click the Publish tool.
 - Type *publish* and then press Enter.

- 2 Open the desired sheet list:
 - Click Open ().
 - Locate and select the .dsd file to publish.
 - Click Open.

For details about creating a sheet list, see “Creating a sheet list” on page 674 in this chapter.

- 3 In Publish To, select Printer Named in Page Setup.
- 4 If you regularly publish to the page setup’s printer instead of other formats, optionally click Set as Default.
- 5 Verify the status of each sheet in the list:
 - No Errors — The sheet is ready for publishing.
 - Layout Not Initialized — The sheet’s layout is not assigned a valid printer. If the sheet is assigned a page setup, choose File > Page Setup and specify a valid printer for the page setup. If no page setup is assigned, open the source drawing file, click the referenced Model or Layout tab, choose File > Print, specify a printer, then click Apply to Layout.
- 6 Specify any of the following options:
 - Number of copies — Enter the number of copies to print.
 - Include print stamp — Select to print with a header and footer. Click () to modify the text of the header and footer.
 - Publish in background — Select to print in the background of other tasks being performed by the computer.
- 7 Click Publish.





- A Click to open a sheet list.
- B Select a sheet list to publish.
- C Select to print to the page setup named for each sheet. Click Set as Default to make this the default setting each time you open the dialog box.
- D Displays the sheets to publish with each sheet's settings.
- E Displays details about the selected sheet.
- F Enter the number of copies to print.
- G Mark to publish in the background of other tasks being performed by the computer.
- H Click to modify the text of the print stamp (available if print stamps are included).
- I Mark to publish with a header and footer.
- J Click to change the page setup for the currently selected sheets.
- K Click to change the layer state for the currently selected sheets.
- L Displays the location of the currently selected printer settings. Click [...] to specify a different folder.

Publishing a sheet list to PDF

Publishing a sheet list to PDF prints all referenced sheets at the same time to a .pdf file. PDF format files allow you to distribute your drawing to others for viewing in Adobe® Acrobat® Reader®, which is free software that users can download. PDF format files can also be viewed, reviewed, and edited in Adobe Acrobat.

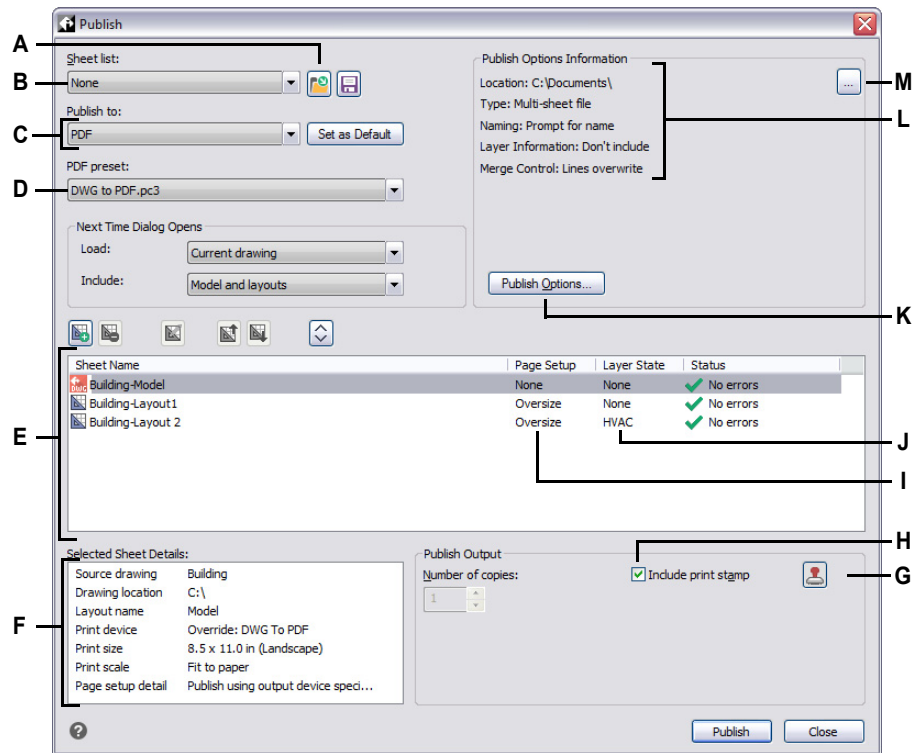
If the Page Setup column indicates None for a sheet, the sheet is published using the print options specified for the layout.

To publish a sheet list to a .pdf file

- 1 Do one of the following to choose Publish (- On the ribbon, choose the Application button, then choose Publish > Publish, or choose Output > Publish.
- On the menu, choose File > Publish.
- On the Standard toolbar, click the Publish tool.
- Type *publish* and then press Enter.
- 2 Open the desired sheet list:
 - Click Open ().
 - Locate and select the .dsd file to publish.
 - Click Open.

For details about creating a sheet list, see “Creating a sheet list” on page 674 in this chapter.
- 3 In Publish To, select PDF.
- 4 If you regularly publish to PDF instead of other formats, optionally click Set as Default.
- 5 In PDF Preset, choose the printer configuration (*.pc3) file that contains the pre-defined settings for creating the .pdf file.
- 6 Click Publish Options and do any of the following:
 - Select options for the published .pdf file.
 - Click Advanced Options to select additional options, then click OK when done.
 - Click [?] to see details about each option.
- 7 Click OK when done setting PDF options.
- 8 Verify the status of each sheet in the list:
 - No Errors — The sheet is ready for publishing.
 - Layout Not Initialized — The sheet’s layout is not assigned a valid printer. If the sheet is assigned a page setup, choose File > Page Setup and specify a valid printer for the page setup. If no page setup is assigned, open the source drawing file, click the referenced Model or Layout tab, choose File > Print, specify a printer, then click Apply to Layout.

- 9 Select Include Print Stamp to print with a header and footer, then click (👤) to modify the text of the header and footer.
- 10 Click Publish.





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| <p>A Click to open a sheet list.</p> <p>B Select a sheet list to publish.</p> <p>C Select to publish to PDF. Click Set as Default to make this the default setting each time you open the dialog box.</p> <p>D Choose the printer configuration (*.pc3) file that contains the predefined PDF options for creating the .pdf file. Available only when publishing to PDF.</p> <p>E Displays the sheets to publish with each sheet's settings.</p> <p>F Displays details about the currently selected sheet.</p> | <p>G Click to modify the text of the print stamp (available if print stamps are included).</p> <p>H Mark to publish with a header and footer.</p> <p>I Click to change the page setup for the currently selected sheets.</p> <p>J Click to change the layer state for the currently selected sheets.</p> <p>K Click to set options for publishing to PDF.</p> <p>L Displays details about the currently selected PDF publish options.</p> <p>M Click to specify a different output folder.</p> |
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Publishing a sheet list to DWF or DWFx

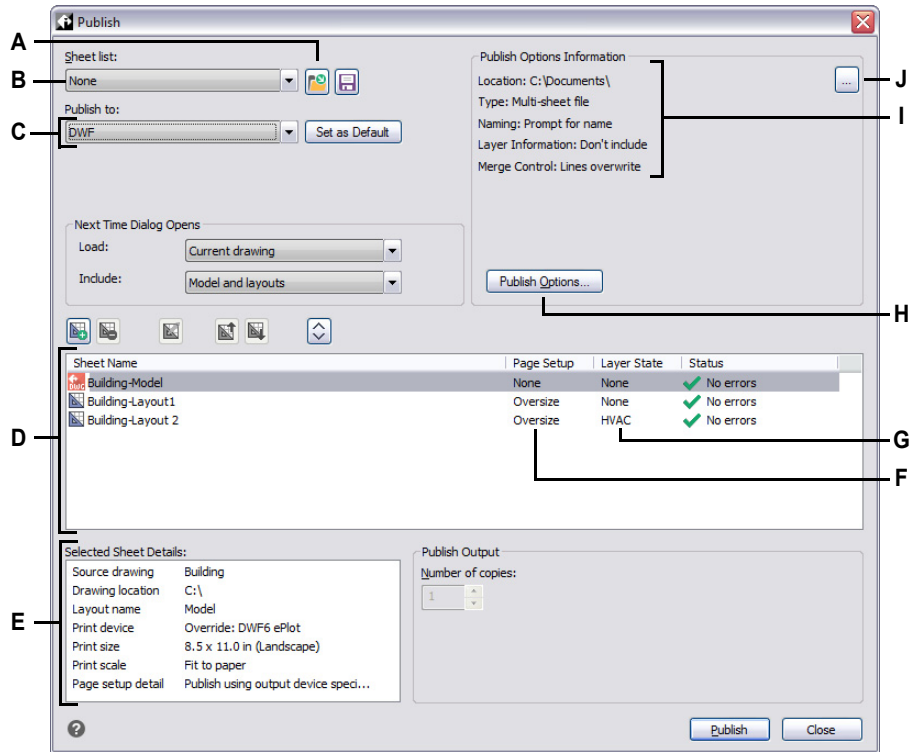
Publishing a sheet list to DWF or DWFx prints all referenced sheets at the same time to a .dwf or .dwfx file, depending on the format you choose. DWF and DWFx format files allow you to distribute your drawing to others for online viewing, reviewing, and editing.

If the Page Setup column indicates None for a sheet, the sheet is published using the print options specified for the layout.

To publish a sheet list to a .dwf or .dwfx

- 1 Do one of the following to choose Publish (- On the ribbon, choose the Application button, then choose Publish > Publish, or choose Output > Publish.
- On the menu, choose File > Publish.
- On the Standard toolbar, click the Publish tool.
- Type *publish* and then press Enter.
- 2 Open the desired sheet list:
 - Click Open ().
 - Locate and select the .dsd file to publish.
 - Click Open.

For details about creating a sheet list, see “Creating a sheet list” on page 674 in this chapter.
- 3 In Publish To, select DWF or DWFx.
- 4 If you regularly publish to DWF or DWFx instead of other formats, optionally click Set as Default.
- 5 Click Publish Options and select options for the published .dwf or .dwfx file. Click [?] to see details about each option.
- 6 Click OK when done setting publish options.
- 7 Verify the status of each sheet in the list:
 - No Errors — The sheet is ready for publishing.
 - Layout Not Initialized — The sheet’s layout is not assigned a valid printer. If the sheet is assigned a page setup, choose File > Page Setup and specify a valid printer for the page setup. If no page setup is assigned, open the source drawing file, click the referenced Model or Layout tab, choose File > Print, specify a printer, then click Apply to Layout.
- 8 Click Publish.




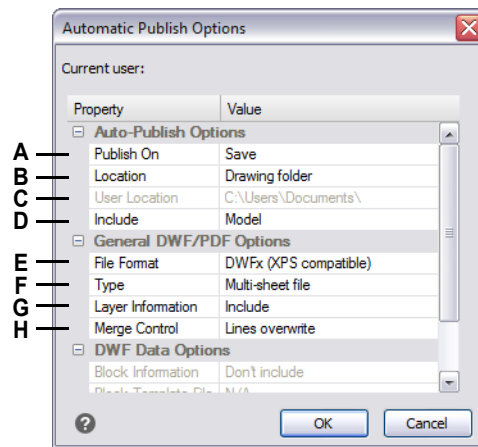
- A** Click to open a sheet list.
- B** Select a sheet list to publish.
- C** Select to publish to DWF or DWFx. Click Set as Default to make this the default setting each time you open the dialog box.
- D** Displays the sheets to publish with each sheet's settings.
- E** Displays details about the currently selected sheet.
- F** Click to change the page setup for the currently selected sheets.
- G** Click to change the layer state for the currently selected sheets.
- H** Click to set options for publishing to DWF/DWFx.
- I** Displays details about the currently selected DWF/DWFx publish options.
- J** Click to specify a different output folder.

Publishing a sheet list automatically

Drawings can be automatically published whenever they are saved or closed, with or without prompting. During automatic publishing, .dwf, dwfx, or .pdf files are saved according to predefined options that you specify. By default, automatic publishing is turned off, so don't forget to turn it on after you set it up.


To set up auto publishing

- 1 Do one of the following to choose Auto Publish ():
 - On the ribbon, choose the Application button, then choose Publish > Auto Publish.
 - On the menu, choose File > Auto Publish.
 - Type *autopublish* and then press Enter.
- 2 Choose Settings.
- 3 Make your selections, then click OK.
- 4 To publish, for example a test publish with your new settings, press Enter. To skip publishing, press Esc.



- | | |
|--|--|
| <p>A Select when to auto publish: when the file is saved or closed, and with or without a prompt.</p> <p>B Select whether to save published files to the same folder as the drawing, a subfolder of the drawing folder, or a different folder.</p> <p>C Select the folder where you want to save published files. (Available only if User Location is selected in Location.)</p> <p>D Select whether the model, layouts or both the model and layouts are published.</p> | <p>E Select whether to publish the drawing to a .dwf, .dwfx or .pdf file.</p> <p>F Select whether to publish all sheets to a single file or each sheet to its own file.</p> <p>G Select whether to include layer information.</p> <p>H Select whether overlapping lines include only the last line or are merged together.</p> |
|--|--|

To turn automatic publishing on or off

- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Printer Options (in Print), or choose Tools > Options (in Manage) then click the Printing tab.
 - On the menu, choose Tools > Options then click the Printing tab or choose File > Printer Options.
 - Type *options*, press Enter, then click the Printing tab.
- 2 Mark or unmark Automatic Publish on Save or Close.
- 3 Click OK.

Drawing in three dimensions

Paper drawings typically represent two-dimensional views of three-dimensional objects. With PTCAD, you can create three-dimensional models of three-dimensional objects.

This section explains how to:

- View entities in three dimensions.
- Create three-dimensional entities.
- Convert three-dimensional entities to surfaces or meshes.
- Edit entities in three-dimensional space.
- Edit three-dimensional solids.
- Display hidden-line, shaded, and rendered views of three-dimensional entities.

The tools and commands for many of the functions described in this section appear on the Draw 3D toolbar and the Insert menu, respectively, when you set the program to the Advanced experience level.

Topics in this chapter

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<i>Creating three-dimensional entities</i>	706
<i>Editing in three dimensions</i>	747
<i>Editing three-dimensional solids</i>	756
<i>Hiding, shading, and rendering</i>	767

Viewing entities in three dimensions

You can view an PTCAD drawing from any position in three-dimensional space. From any selected viewing position, you can add new entities and modify existing entities. You can also generate hidden-line and shaded views from any viewing position.

You view three-dimensional drawings by setting the viewing direction. The viewing direction establishes the viewing position, the Cartesian coordinate corresponding to the viewpoint looking back at the origin point, the 0,0,0 coordinate. When you view a drawing from the default viewpoint (0,0,1), you see a plan view of the drawing.

Changing the viewing direction can be affected by smooth view transitions.

Type vtoptions to specify whether smooth view transitions are used when changing the viewing direction (rotation), and to specify speed and performance settings.

Switching to a preset viewing direction

Using preset views, you can quickly change the viewing direction to look at the drawing from a different vantage point or to work on a three-dimensional model from a different orientation.


To switch to a preset viewing direction

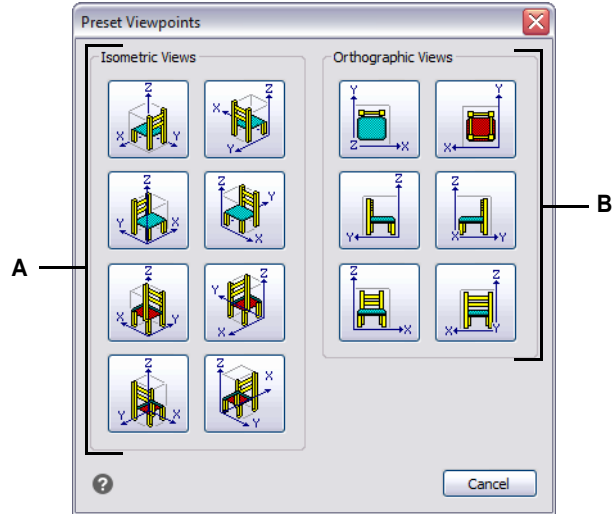
- 1 Do one of the following:
 - On the ribbon, go to the Views area of the View tab.
 - On the menu, choose View > 3D Views.
 - Go to the View toolbar.
 - Type *view*.
 - Click the view control in the upper left corner of the drawing window.
- 2 Select or type the name of one of the following:
 - Top, Front, Right Side, Left Side, Back, Bottom
 - Above, Left Front; Above, Right Front; Above, Right Rear; Above, Left Rear
 - Below, Left Front; Below, Right Front; Below, Right Rear; Below, Left Rear

The view cube also has preset views.

Click a preset view on the view cube to switch the viewing direction. For more details, see “Using the view cube to switch the view” on page 694 in this chapter.

To switch to a preset viewing direction using the Preset Viewpoints command

- 1 Do one of the following to choose Preset Viewpoints ():
 - On the ribbon, choose View > Preset Viewpoints (in Views).
 - On the menu, choose View > Preset Viewpoints.
 - On the View toolbar, click the Preset Viewpoints tool.
 - Type *setvpoint* and then press Enter.
- 2 Click the preset view you want to use.



A Click to select an isometric viewpoint.

B Click to select an orthographic viewpoint.

Creating and switching to named views


As you work on a drawing, you may find that you frequently switch among different portions of it. For example, if you are drawing the floor plan of a house, you may zoom in to particular rooms of the house and then zoom out to display the entire house. Although you can repeat the Pan and Zoom commands to do this, it is much easier to save various views of the drawing. You can then quickly switch among these views.

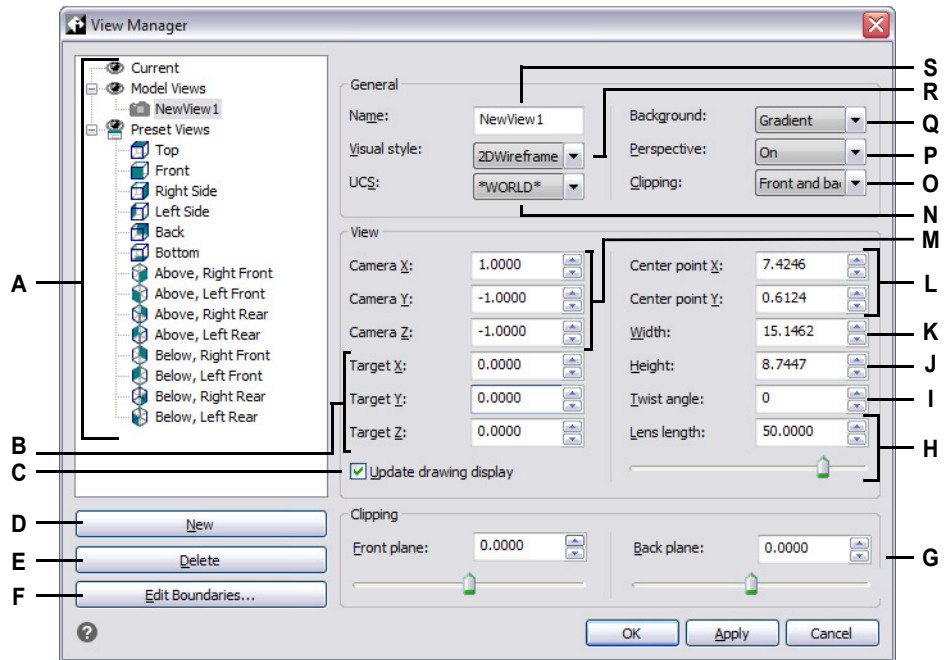
These custom views that you create, called *named views*, can control more settings than just view direction and location — they can also include the visual style, UCS, background, perspective and lens length, clipping, etc.

Creating a named view

Named views can be saved and displayed in the current window.


To create a named view for the current view in the drawing

- 1 Do one of the following to choose View Manager ():
 - On the ribbon, choose View > View Manager (in Views).
 - Choose View > View Manager.
 - On the View toolbar, click the View Manager tool.
 - In the View element of the Explorer, click the View Manager tool.
 - Type *view* and then press Enter.
- 2 Click New.
- 3 In Name, type a name for the view.
- 4 Choose the desired options. If you want to see your changes directly in the drawing as you specify them, mark Update drawing display.
- 5 Click OK when done.




- A** Lists the current view, model views (named views and cameras), and preset views.
- B** Enter the x-, y- and z- coordinates of the selected view's target location, to which the view is oriented.
- C** Mark to update the drawing display automatically as you change settings.
- D** Click to create a new view.
- E** Click to delete the selected view.
- F** Click to select the boundaries of the view directly in the drawing.
- G** Enter the front and back distances of the selected view's clipping planes, or move the slider. Available only when clipping is turned on to the appropriate setting.
- H** Enter the lens length (millimeters) of the selected view, or move the slider. Available only when Perspective is turned on.
- I** Enter the twist angle for the selected view.
- J** Enter the height of the selected view.
- K** Enter the width of the selected view.
- L** Enter the x- and y-coordinates of the selected view's center point.
- M** Enter the x-, y- and z-coordinates of the selected view's camera location, from which the view is oriented.
- N** Select the user coordinate system for the selected view.
- O** Select where to locate boundary clipping of the selected view.
- P** Select Off (orthographic) or On (perspective) for the selected view. (Then specify the lens length.)
- Q** Select a background for the selected view.
- R** Select a visual style for the selected view.
- S** Enter a new name for the selected view.

To create a named view for a portion of the current view in the drawing

- 1 Do one of the following to choose View Manager ():
 - On the ribbon, choose View > View Manager (in Views).
 - Choose View > View Manager.
 - On the View toolbar, click the View Manager tool.
 - In the View element of the Explorer, click the View Manager tool.
 - Type *view* and then press Enter.
- 2 Click New.
- 3 In Name, type a name for the view.
- 4 Make any additional changes to the view. And if you want to see your changes directly in the drawing as you specify them, mark Update drawing display.
- 5 Click OK when done.

To create a named view from an existing view

- 1 Do one of the following to choose View Manager ():
 - On the ribbon, choose View > View Manager (in Views).
 - Choose View > View Manager.
 - On the View toolbar, click the View Manager tool.
 - In the View element of the Explorer, click the View Manager tool.
 - Type *view* and then press Enter.
- 2 In the list of views, select the view that has the settings you want to use as a base for the new view.
- 3 Click New.
- 4 In Name, type a name for the new view.
- 5 Make any additional changes to the view. And if you want to see your changes directly in the drawing as you specify them, mark Update drawing display.
- 6 Click OK when done.


View Manager is not a transparent command.

For example, you cannot use View Manager during the Preset Viewpoints or Define View commands, or when zooming or panning the drawing.

Setting the background for a named view

For each named view that you create, you can assign a solid-color, gradient, or image background. Whenever you switch to that view, the background displays if the view is also assigned any visual style other than 2D Wireframe.

To set the background for a named view

- 1 Do one of the following to choose View Manager ():
 - On the ribbon, choose View > View Manager (in Views).
 - On the menu, choose View > View Manager.
 - On the View toolbar, click the View Manager tool.
 - Type *view* and then press Enter.
 - In the View element of the Explorer, click the View Manager tool.
- 2 Select the model view for which you want to set a background.
- 3 In Visual style, select Conceptual, Hidden, or Realistic.
- 4 In Background, choose one of the following:
 - Solid
 - Gradient
 - Image
- 5 In the Background dialog box, make your selections. Click [?] to see details about each option.
- 6 Click OK.


Use the Background command to set a background for the current view.

To specify a solid-color, gradient, or image background for the current view, type background and make your selections.

Switching to a named view

After you save one or more named views, you can restore any of those views in the current window using either the View command or PTCAD's Explorer.

To switch to a named view

- 1 Do one of the following to choose View Manager ():
 - On the ribbon, choose View > View Manager (in Views).
 - On the menu, choose View > View Manager.
 - On the View toolbar, click the View Manager tool.
 - Type *view* and then press Enter.
 - Click the view control in the upper left corner of the drawing window.
- 2 Select the view you want to switch to.
- 3 Click OK.


You can also switch between named views using the Explorer.

Choose Tools > Explore Views (in Explorer), then double-click the desired named view.

Modifying named views

Once you create a named view, you can modify any of its settings or simply fine-tune it as you drawing changes. Only named views that you create can be changed; preset views cannot be modified.

To change the settings of a named view

- 1 Do one of the following to choose View Manager ():
 - On the ribbon, choose View > View Manager (in Views).
 - Choose View > View Manager.
 - On the View toolbar, click the View Manager tool.
 - In the View element of the Explorer, click the View Manager tool.
 - Type *view* and then press Enter.
- 2 To see your changes directly in the drawing as you make them, mark Update drawing display.
- 3 Change the desired settings.
- 4 Click OK when done.

Using the view cube to switch the view

The view cube is a navigation tool that indicates the current viewing direction and allows you to switch between preset views and custom viewing directions. You can also use the view cube to change the coordinate system and switch between orthographic and perspective projections.

The view cube displays in the current viewport when working in 2D model space or a 3D visual style. It becomes active when you move the mouse over it, and it can display with transparency or not at all when inactive.

Turning the display of the view cube on or off

The view cube is a convenient way to view a drawing from various directions. When you aren't using it, you can turn the display off.

The view cube can be transparent when inactive.

Instead of turning the view cube display off, you might want to make it transparent, all or partially, when it is inactive. For details, see “Changing view cube settings” on page 696 in this chapter.

To display or hide the view cube

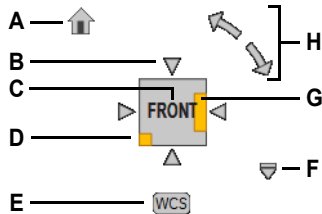
1 Do one of the following to choose View Cube:

- On the ribbon, choose View > View Cube (in Interface).
- On the menu, choose View > Display > View Cube.
- Type *navvcube*, press Enter, then choose On or Off.

Changing the view using the view cube

Use the view cube to view a drawing using the following:

- Home view
- Orthographic and isometric preset views
- Custom views
- Orthographic and perspective projections
- Coordinate systems



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| A Click to switch to the home view saved with the drawing. | E Click to choose the coordinate system. |
| B Click an arrow to change to an orthogonal preset view, orienting from that direction. | F Click to switch to the home view, set the home view, switch between Parallel and Perspective projections, and access view cube settings. |
| C Click a face to switch to the named orthographic preset view. | G Click an edge to switch to a custom view, orienting the view from that direction. |
| D Click a corner to switch to an isometric preset view. | H Click to rotate the view 90 degrees, or click and drag to rotate the view in real-time. |

To change the view direction using the view cube

- 1 Do one of the following:
 - Click Activate Home View to return to the home view that is saved with the drawing. To set the current view as the home view, right-click the view cube, and then choose Set Current View as Home.
 - Click a face of the cube to switch to the Top, Front, Left Side, Right Side, Back, and Bottom preset orthographic views.
 - Click a corner of the cube to switch to the Above, Left Front; Above, Right Front; Above, Right Rear; Above, Left Rear; Below, Left Front; Below, Right Front; Below, Right Rear; and Below, Left Rear preset isometric views.
 - Click an edge of the cube to switch to a custom view, orienting the view from that direction.
 - Click an arrow to switch to the orthographic view, orienting the view from that direction. The arrows display if you already are viewing the drawing with an orthographic view (for example, after clicking a face of the cube).
 - Click the rotation arrows (available when viewing orthographically) to rotate the view 90 degrees, or click and drag the view cube or rotation arrows to rotate the view in real-time.

To switch between parallel (orthographic) and perspective projections using the view cube

- 1 Do one of the following:
 - Click the drop-down arrow in the lower right of the view cube, and then choose Parallel or Perspective.
 - Right-click the view cube, and then choose Parallel or Perspective.

The perspective projection changes all preset views to perspective views.

When using the perspective projection, all views become perspective views and are no longer orthographic or isometric views.

To change the coordinate system using the view cube

- 1 Click the Activate Coordinate System menu.
- 2 Choose WCS or the name of the user coordinate system. You can also choose New UCS to create a new user coordinate system.

Changing view cube settings

You can change the location and appearance of the view cube, such as which corner of the viewport it is located and how transparent it is when inactive. Additionally, you can specify various options for using it. For example, you may want to prevent the model from turning upside down when rotating the view.



To change view cube settings

- 1 Do one of the following to choose View Cube Settings:
 - Click the drop-down arrow in the lower right of the view cube, and then choose View Cube Settings.
 - Right-click the view cube and choose View Cube Settings.
 - Type *navvcube*, press Enter, then choose Settings.
- 2 To change the appearance and location of the view cube, select from the following options:
 - **On-screen Position** Select the corner where to display the view cube in the current viewport.
 - **Size** Select the size of the view cube. If Automatic, the size adjusts automatically depending on the drawing window size and number of viewports.
 - **Inactive Opacity** Move the slider to set the opacity percentage for the view cube when it is inactive. The lower the number, the more transparent the view cube is.
 - **Show UCS Menu** Mark to display the active coordinate system drop-down menu below the view cube.
- 3 Select any of the following options, depending on your preferences for using the view cube:
 - **Snap to Closest View When Dragging** Mark to adjust the current view to the closest preset view when dragging the view cube to rotate the view.
 - **Zoom to Extents after View Change** Mark to automatically zoom the model to fit the current viewport after a view change.
 - **Use View Transitions When Switching Views** Mark to turn on smooth view transitions when switching between views using the view cube.
 - **Orient View Cube to Current UCS** Mark to orient the view cube to reflect the current user coordinate system. When unmarked, it is oriented to reflect the WCS.
 - **Keep Scene Upright** Mark to prevent turning the viewpoint of the model upside down.
- 4 Click OK.


Walking and flying through a drawing

Drawings can be viewed as though you are walking or flying through them

To walk through a drawing

- 1 Do one of the following to choose 3D Walk () or 3D Fly ():
 - On the ribbon, choose Visualize > 3D Walk (in Animations) or 3D Fly (in Animations).
 - On the menu, choose View > Walk and Fly > 3D Walk or 3D Fly.
 - On the Walk and Fly toolbar, click the 3D Walk or 3D Fly tool.
 - Type *3dwalk* or *3Dfly* and then press Enter.
- 2 Click and drag in the drawing to walk or fly through the model, or use the arrow keys on the keyboard. When finished, release the mouse button.
- 3 Repeat to walk or fly again.
- 4 When finished, press Enter, right-click the drawing, or press Esc.

To customize walk and fly settings

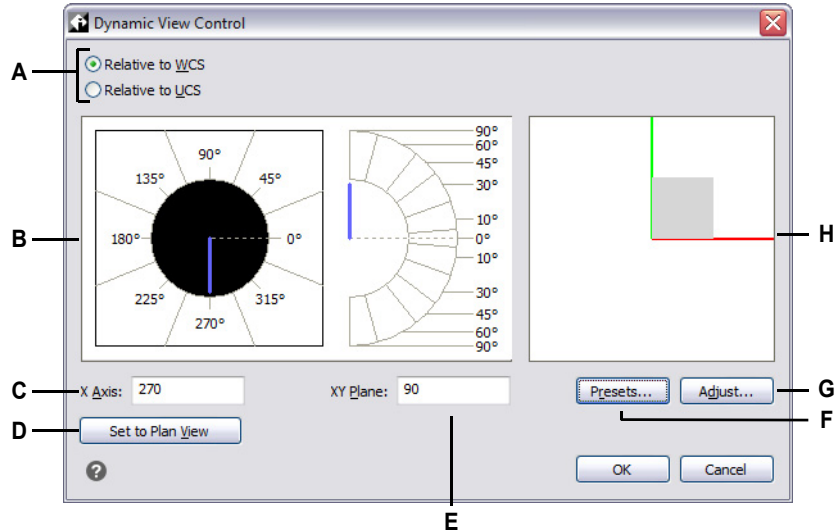
- 1 Do one of the following to choose Walk and Fly Settings ():
 - On the ribbon, choose Visualize > Walk and Fly Settings (in Animations).
 - On the menu, choose View > Walk and Fly > Walk and Fly Settings.
 - On the Walk and Fly toolbar, click the Walk and Fly Settings tool.
 - Type *3dwalkflysettings* and then press Enter.
- 2 In Step Size, enter the value for the step size in drawing units, then enter the speed in steps per second.
- 3 Mark Prevent Clipping if you want to disable clipping.
- 4 In Lens Length, enter the value for the lens length you want to use. This value applies only when using the 3D Walk and 3D Fly commands; it does not change the default lens length that is determined by the LENSLENGTH system variable.
- 5 In Walk Elevation, specify the elevation to apply to walk mode during the 3D Walk command.
- 6 Click OK.

Setting a viewpoint

You can dynamically rotate the viewpoint within the xy plane and relative to the xy plane, and you can pan and zoom the drawing. As you change the viewpoint settings, the drawing display automatically updates.

To dynamically set a view direction

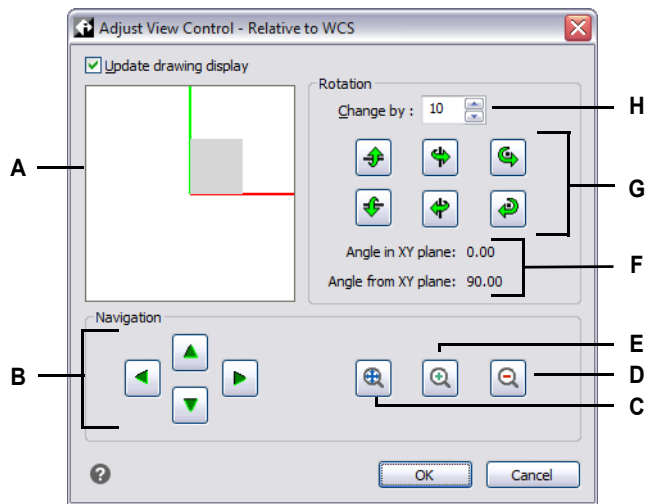
- Do one of the following to choose Dynamic View Control (👁️):
 - On the ribbon, choose View > Dynamic View Control (in Views).
 - On the menu, choose View > Dynamic View Control.
 - On the View toolbar, click the Dynamic View Control tool.
 - Type `viewctl` and then press Enter.
- Make your selections to change the viewpoint.
- To complete the command, click OK.



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| <p>A Select whether the viewing direction is relative to the World Coordinate System or User Coordinate System.</p> <p>B Displays the current viewing direction settings. Click to select a new viewing direction.</p> <p>C Enter the angle from the x-axis.</p> <p>D Click to set the viewing direction to the plan view.</p> | <p>E Enter the angle from the xy-plane.</p> <p>F Click to open the Preset Viewpoints dialog box and set the viewing direction using predefined settings.</p> <p>G Click to open the View Control dialog box and set the viewing direction interactively with the drawing.</p> <p>H Indicates the current viewpoint.</p> |
|--|---|

To set a view direction interactively with the drawing

- 1 Do one of the following to choose Dynamic View Control (🌐):
 - On the ribbon, choose View > Dynamic View Control (in Views).
 - On the menu, choose View > Dynamic View Control.
 - On the View toolbar, click the Dynamic View Control tool.
 - Type *viewctl* and then press Enter.
- 2 Click Adjust.
- 3 Make your selections to change the 3D viewing direction within the drawing.
- 4 Click OK.
- 5 To complete the command, click OK.



- | | |
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| <p>A Indicates the current viewpoint.</p> <p>B Click to pan the drawing.</p> <p>C Click to zoom extents.</p> <p>D Click to zoom out.</p> | <p>E Click to zoom in.</p> <p>F Shows the current viewpoint orientation.</p> <p>G Click to rotate the view about a predefined axis.</p> <p>H Type or select the rotation angle increment.</p> |
|--|---|

Using a section plane to view a drawing

Section planes are used to visualize cross-sections of two- and three-dimensional entities and point clouds. For example, you can move a horizontal section plane that cuts through a multi-level building to interactively view inside the building under the roof, then under the top floor, etc.

A single drawing can contain multiple section planes, allowing you to create cross-sections of different model areas.

Section planes only apply when the graphics device is set to OpenGL ES.


Section planes are used for visualization, not documentation, of a model.

Sections and section lines (if your program version supports them) can be used to document a model. For more details about sections, see “Sectioning and slicing solids” on page 757 in this chapter. For details about section lines, see “Drawing a section line” on page 173.


Drawing section planes

You can draw section planes aligned orthographically with the current UCS or by specifying points.

To draw an orthographic section plane

- 1 Do one of the following to choose Section Plane (- On the ribbon, choose Visualize > Section Plane (in Live Sections); AEC > Section Plane (in Sections, if your program version supports AEC entities).
- On the menu, choose View > Live Sections > Section Plane; AEC > Section Planes > Section Plane (if your program version supports AEC entities).
- Type *sectionplane* and then press Enter.
- 2 Choose Orthographic.
- 3 Choose the orthographic orientation (relative to the UCS) that you want the section plane to align with.

To draw a section plane by specifying points

- 1 Do one of the following to choose Section Plane (- On the ribbon, choose Visualize > Section Plane (in Live Sections); AEC > Section Plane (in Sections, if your program version supports AEC entities).

- On the menu, choose View > Live Sections > Section Plane; AEC > Section Planes > Section Plane (if your program version supports AEC entities).
 - Type *sectionplane* and then press Enter.
- 2 Specify a first point.
 - 3 Specify a second point.
 - 4 Do one of the following:
 - To draw a section plane that is automatically aligned with the current view direction, press Enter.
 - To draw a section plane with a different orientation, pick a third point to define its vertical direction.

You can specify the elevation and exact location of a selected section plane.

Type entprop to open the Properties pane. In Vertex, specify which vertex to change, then enter the x-, y-, and z-coordinates of the vertex.

Viewing a model using section planes

Use the center grip of a section plane to reposition the section plane and change the view of intersecting entities. Also if needed, you can turn live sectioning on or off for any section plane. This allows to you change whether the cross sections of intersecting entities are displayed.

If the visualization of an entity is unchanged by a section plane, check that the entity is on a sectionable layer. For more details, see “Controlling whether a layer is sectionable” on page 294.

Section planes render with the current visual style.

Using shaded visual styles makes it easier to see and select a section plane.


To view a model interactively using a section plane

- 1 Select the section plane.
- 2 Click and drag the double-arrow grip in the center of the section plane to move it and interactively visualize different areas of the model.
- 3 Click and drag other grips to resize the section plane, if necessary.

Section planes are infinite.

Although you can use grips to change the size of the section plane, the section plane is infinite. Resizing the plane helps with selecting and visualizing the plane entity.


To turn live sectioning on or off for a section plane

- 1 Do one of the following to choose Live Section On/Off (- On the ribbon, choose Visualize > Live Section On/Off (in Live Sections); AEC > Live Section On/Off (in Sections, if your program version supports AEC entities).
- On the menu, choose View > Live Sections > Live Section On/Off; AEC > Section Planes > Live Section On/Off (if your program version supports AEC entities).
- Type *livesection* and then press Enter.
- 2 Select the section plane for which you want to turn live sectioning on or off.

Changing display options for the intersection area of section planes

The appearance of the intersection area where the section plane intersects entities is customizable. The display settings you choose apply to all section planes in the drawing.

To change display options for the intersection area of section planes


- 1 Do one of the following to choose Section Plane Settings (- On the ribbon, choose Visualize > Section Plane Settings (in Live Sections); AEC > Section Plane Settings (in Sections, if your program version supports AEC entities).
- On the menu, choose View > Live Sections > Section Plane Settings; AEC > Section Planes > Section Plane Settings (if your program version supports AEC entities).
- Type *sectionplanesettings* and then press Enter.
- Choose Settings when using the Section Plane command.
- 2 To display the intersection area with a color or pattern that is different than the rest of the section plane, do the following:
 - Mark Display Section Fill.
 - In Fill Color, select the color used to fill the intersection area.
- 3 To display the intersection area with a pattern and fill, do the following:
 - Mark Display Section Pattern.
 - In Pattern Type, select the pattern.
 - In Pattern Color, select the color for the pattern.
- 4 Click OK.

Using a camera to view the drawing


Cameras define settings for 3D views of drawings according to a location and target. You can also specify the camera height, clipping boundaries for the view, and other options.

To modify a camera, select it in the drawing and move its grips or change its properties in the Properties pane. Cameras are named views that can also be changed using the View Manager. For more details about the Properties pane, see “Modifying the properties of entities” on page 406. For more details about View Manager, see “Working with named views” on page 333.

To create a camera

- 1 Do one of the following to choose Create Camera ():
 - On the ribbon, choose View > Create Camera (in Views).
 - On the menu, choose View > Create Camera.
 - On the View toolbar, click the Create Camera tool.
 - Type *camera* and then press Enter.
- 2 Specify the camera location.
- 3 Specify the target point of the view.
- 4 Choose one of the following to specify more options:
 - **Name** Enter a new name for the camera, then press Enter.
 - **Location** Specify a new camera location, then press Enter.
 - **Height** Specify the height of the camera, then press Enter.
 - **Target** Specify a new target point for the camera view, then press Enter. The target location is the center of the view.
 - **Lens** Enter the length of the lens. The greater the value, the more narrow the view. The value must be 600 or less millimeters.
 - **Clipping** There are two clipping planes that help define the view. Anything between the camera and the front clipping plane is hidden, and anything between the back clipping plane and the target is hidden. Choose Yes to specify a front (or back) clipping plane, enter a distance, and then press Enter. Choose No for either option to disable the clipping plane.
 - **View** Choose Yes to switch the current view to the camera settings when the camera is created. Choose No to keep the current view the same when the camera is created.
- 5 When done, press Enter.


To switch to a camera view

- 1 Do one of the following to choose View Manager ():
 - On the ribbon, choose View > View Manager (in Views).
 - On the menu, choose View > View Manager.
 - On the View toolbar, click the View Manager tool.
 - Type *view* and then press Enter.
- 2 Select a camera view.
- 3 Click OK.

Displaying a plan view of the current drawing

You can set the current viewing direction to the plan view of the current user coordinate system (UCS), a previously saved UCS, or the World Coordinate System (WCS).

To display a plan view of the current drawing

- 1 Do one of the following to choose Plan View ():
 - On the ribbon, choose View > Plan View (in Views).
 - On the menu, choose View > Plan View.
 - On the View toolbar, click the Plan View tool.
 - Type *plan* and then press Enter.
- 2 In the prompt box, choose one of the following:
 - Current displays the plan view of the current UCS.
 - UCS displays the plan view of a saved UCS. The program prompts you for the name of the UCS.
 - World displays the plan view of the WCS.

Creating three-dimensional entities

PTCAD supports the following types of three-dimensional models:

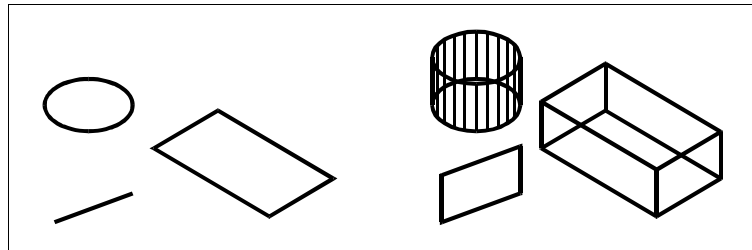
- Wire-frame models, which consist of lines and curves that define the edges of a three-dimensional entity. You can create a wire-frame model by drawing lines, arcs, polylines, and other two-dimensional entities anywhere in three-dimensional space. Wire-frame models have no surfaces; they always appear as outlines. Because you must individually draw and position each entity that makes up a wire-frame model, creating one can be exacting and time-consuming.
- Surface models, which consist of both edges and the surfaces between those edges. You can create a surface model by applying elevation and thickness to two-dimensional planar entities or by using specific three-dimensional entity-creation commands. Surface models consist of individual planes forming a faceted, polygonal mesh.
- Facet model meshes, which are smart surface model entities that can be combined into composite meshes and rendered more like solids.
- 3D solids, which are three-dimensional ACIS entities that consist of faces and edges. 3D solids appear to have volume and are easier to work with than wire-frame and surface models. PTCAD supports viewing and limited editing of 3D solids, including moving, rotating and scaling. Additionally, some versions of PTCAD allow you to create and more completely edit 3D solids.

Applying elevation and thickness

By default, the program creates new two-dimensional entities with a zero elevation and thickness. The easiest way to create a three-dimensional entity is to change the elevation or thickness property of an existing two-dimensional entity.

The elevation of an entity is its z-coordinate position in relation to the xy plane in which the entity is drawn. An elevation of 0 indicates that the entity is drawn on the xy plane of the current UCS. Positive elevations are above this plane; negative elevations are below it.

The thickness of an entity is the distance it is extruded above or below its elevation. A positive thickness extrudes the entity upward in the positive z direction of the entity; a negative thickness extrudes it downward in the negative z direction. The thickness is applied uniformly to the entire entity. You can extrude any two-dimensional entity into a three-dimensional entity by changing the thickness of the entity to a nonzero value. For example, a circle becomes a cylinder, a line becomes a three-dimensional plane, and a rectangle becomes a box.



Two-dimensional entities.

Two-dimensional entities with thickness added.

You can create three-dimensional entities using any of the following methods:


- Draw two-dimensional entities in three-dimensional space.
- Convert two-dimensional planar entities into three-dimensional entities by applying elevation and thickness.
- Convert two-dimensional planar entities into three-dimensional entities by revolving or extruding.
- Create three-dimensional entities such as boxes, cylinders, cones, domes, spheres, and wedges.

Three-dimensional solids are drawn as true solids with versions of PTCAD that support three-dimensional ACIS solids.


Three-dimensional solids that you can create include: box, cone, cylinder, dish, dome, pyramid, sphere, torus, and wedge.

You can change the default elevation and thickness values to create new entities with an elevation and thickness already applied.


To set the current elevation

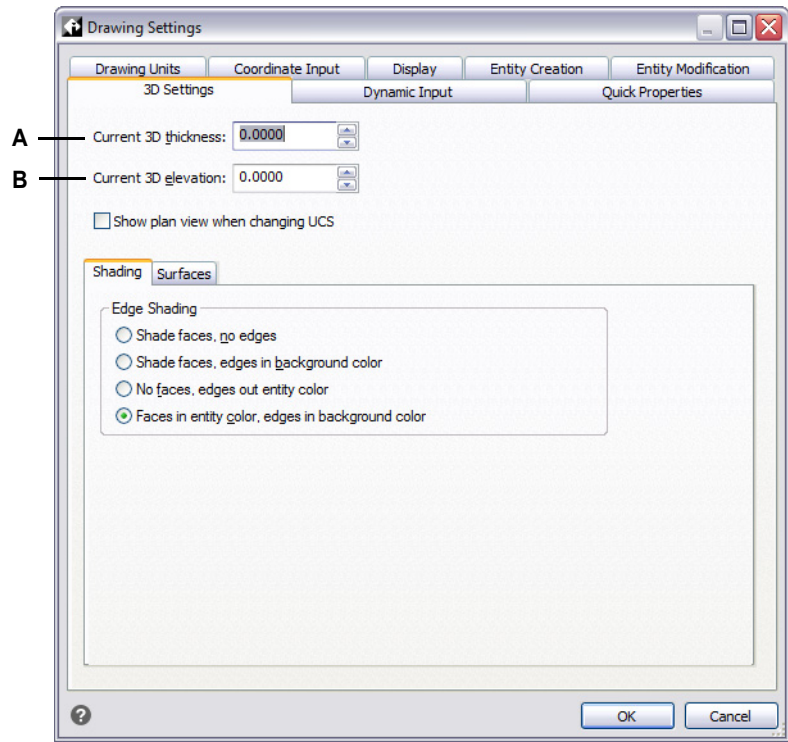
- 1 Do one of the following to choose Elevation ():
 - On the ribbon, choose Draw > Elevation (in Settings).
 - On the menu, choose Format > Elevation.
 - On the Format toolbar, click the Elevation tool.
 - Type *elev* and then press Enter.
- 2 Specify the New Current Value For Elevation, and then press Enter.

To set the current thickness

- 1 Do one of the following to choose Thickness (- On the ribbon, choose Draw > Thickness (in Settings).
- On the menu, choose Format > Thickness.
- On the Format toolbar, click the Thickness tool.
- Type *thickness* and then press Enter.
- 2 Specify the New Current Value For Thickness, and then press Enter.


To set the current elevation and thickness using a dialog box

- 1 Do one of the following to choose Drawing Settings (- On the ribbon, choose Application button > Drawing Utilities; Home > Drawing Settings (in Utilities); or Tools > Drawing Settings (in Manage).
- On the menu, choose Tools > Drawing Settings.
- On the Tools toolbar, click the Drawing Settings tool.
- Type *dsettings* and then press Enter.
- 2 Click the 3D Settings tab.
- 3 Click the Surfaces tab.
- 4 To change the current thickness, in the Current 3D Thickness box, type a new thickness value or click the arrows to select a new thickness.
- 5 To change the current elevation, in the Current 3D Elevation box, type a new elevation value or click the arrows to select a new elevation.
- 6 Click OK.



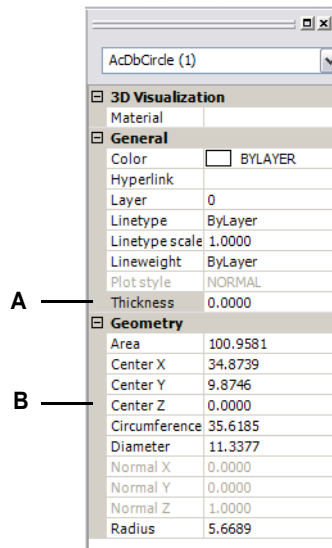
- A** Type or select the current three-dimensional thickness. **B** Type or select the current three-dimensional elevation.

To change the thickness and elevation of an existing entity

- 1 Do one of the following to choose Properties ():
 - On the ribbon, choose View > Properties (in Panes).
 - On the menu, choose View > Display > Properties or Modify > Properties.
 - On the Modify toolbar, click the Properties tool.
 - Type *entprop* and then press Enter.
- 2 Select the entity, and then press Enter.
PTCAD displays the Properties palette (its exact appearance depends on the type of entity you select).
- 3 To change the thickness, in the Thickness box, type a new thickness value or click the arrows to select the new thickness.
- 4 To change the elevation, in the Z coordinate box (or some entities have an Elevation box), type a new elevation value or click the arrows to select the new elevation.
- 5 Click OK.

When you change the thickness of an entity, you do not change the entity type.

*If you want to extrude an entity and convert it to a three-dimensional solid, use the **Extrude** command. If you want to thicken a surface or mesh entity and convert it to a three-dimensional solid, use the **Thicken** command.*



A Type or select the new thickness.


B Type or select the new elevation.

Creating three-dimensional faces

You can create a three-dimensional face, which consists of a section of a plane in three-dimensional space. You define a three-dimensional face by specifying the x,y,z coordinates of three or more corners. After you specify the fourth point, the program continues to prompt you for additional faces by alternating prompts for the third point and fourth point to allow you to build a complex three-dimensional entity. Each three- or four-sided plane is created as a separate three-dimensional face entity.

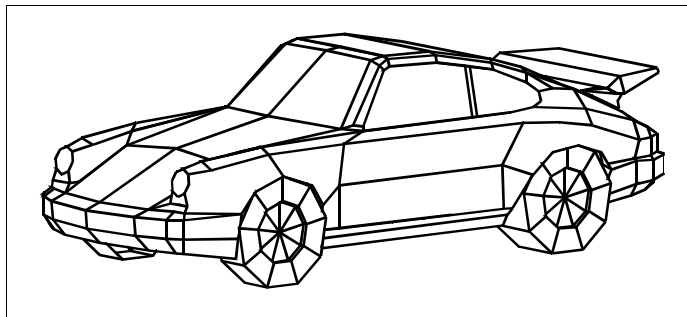
To create a three-dimensional face

Advanced experience level

- 1 Do one of the following to choose Face ():
 - On the ribbon, choose Draw 3D > 3D Face (in Draw 3D Meshes).
 - On the menu, choose Draw > 3D Meshes > Face.
 - On the Draw 3D Meshes toolbar, click the Face tool.
 - Type *face* and then press Enter.
- 2 Specify the first point of the three-dimensional face.
- 3 Specify the second, third, and fourth points.
- 4 Specify the third and fourth points for additional faces.
- 5 To complete the command, press Enter.

Any or all edges of a three-dimensional face can be invisible to allow you to more accurately model entities with holes in them.

As the program prompts you for the corner points, in the prompt box, choose Invisible Edge to make the next edge invisible. Or after the 3D face is drawn, use the Edge command. For details about the Edge command,




An example of a three-dimensional model created using three-dimensional faces.

Creating rectangular meshes

You can create a three-dimensional rectangular mesh consisting of four-sided polygons. You determine the size of the mesh by specifying the number of vertices along the primary (M-direction) and secondary (N-direction) mesh axes and then specifying the coordinates for each vertex.

To create a rectangular mesh

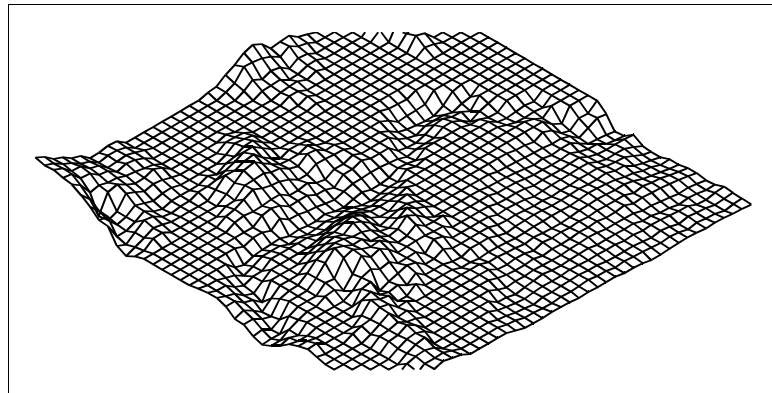
Advanced experience level

- 1 Do one of the following to choose Mesh ():
 - On the ribbon, choose Draw 3D > Meshes (in Draw 3D Meshes).
 - On the menu, choose Draw > 3D Meshes > Mesh.
 - On the Draw 3D Meshes toolbar, click the Mesh tool.
 - Type *mesh* and then press Enter.
- 2 Specify the number of vertices along the primary mesh axis.
- 3 Specify the number of vertices along the secondary mesh axis.
- 4 Specify the coordinates for each vertex.

Specifying the coordinates for the last vertex completes the mesh and ends the command.

Although creating rectangular meshes manually can be exacting, they are useful for representing complex surfaces such as three-dimensional terrain models.

The Mesh tool is most useful when combined with scripts or LISP programs that mathematically calculate the coordinates of the vertices.




An example of a three-dimensional terrain model created using rectangular meshes.

Creating polyface meshes

You can create a polygon mesh consisting of faces connecting three or more vertices. You first determine the coordinates of each vertex and then define each face by entering the vertex numbers for all the vertices of that face. As you create each face, you can control the visibility and color of each edge and assign each edge to specific layers.

To create a polyface mesh

Advanced experience level

- 1 Do one of the following to choose Polyface Mesh ():
 - On the ribbon, choose Draw 3D > Polyface Mesh (in Draw 3D Meshes).
 - On the menu, choose Draw > 3D Meshes > Polyface Mesh.
 - On the Draw 3D Meshes toolbar, click the Polyface Mesh tool.
 - Type *pface* and then press Enter.
- 2 Specify the coordinates of each vertex.

After each vertex that you specify, the next vertex number is displayed, and you are prompted for the coordinates of the vertex. Specify the coordinates, and then press Enter. Continue to specify the coordinates for each numbered vertex.
- 3 To finish specifying vertex coordinates, press Enter.
- 4 Specify the vertex numbers that define the first face.

You specify the face by entering the vertex numbers that were defined when you specified coordinates in step 2. Each face can be composed of three or more numbered vertices.
- 5 To finish defining the first face, press Enter.
- 6 Specify the next face by entering its vertex numbers.
- 7 To complete the command, press Enter.

Edges can be made invisible.


Type the vertex number as a negative value.

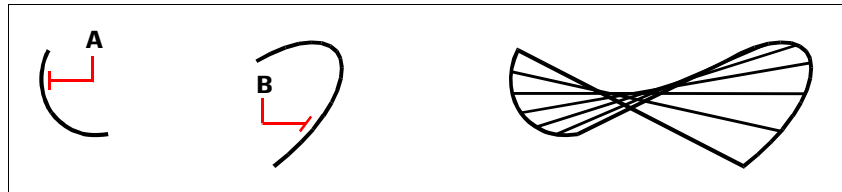
Creating ruled surface meshes

You can create a ruled surface, which is a three-dimensional polygon mesh that approximates the surface between two existing entities. You select the two entities that define the ruled surface. These entities can be arcs, circles, lines, points, or polylines.

To create a ruled surface mesh

Advanced experience level

- 1 Do one of the following to choose Ruled Surface ():
 - On the ribbon, choose Draw 3D > Ruled Surface (in Draw 3D Meshes).
 - On the menu, choose Draw > 3D Meshes > Ruled Surface.
 - On the Draw 3D Meshes toolbar, click the Ruled Surface tool.
 - Type *rulesurf* and then press Enter.
- 2 Select the first defining entity.
- 3 Select the second defining entity.



Select the first (A) and second (B) defining entities.

The resulting ruled surface mesh.

The value of the Number of M-Direction Surfaces controls the density of the mesh.

Choose *Tools > Drawing Settings*, click the *3D Settings* tab, click the *Surfaces* tab, then under *Surface Settings*, change the *Number Of M-Direction Surfaces* value.


Creating extruded meshes

You can create an extruded mesh, which is a three-dimensional polygon mesh that approximates the surface generated by extruding a profile curve along a path. You select the two entities that define the path curve and direction vector. The length of the direction vector determines the distance the path curve is moved along the direction vector. The extruded entity can be an arc, circle, line, or polyline. You can choose a line or open polyline as the direction vector. The resulting mesh consists of a series of parallel polygonal planes running along the specified path.


You can create two types of extruded meshes:

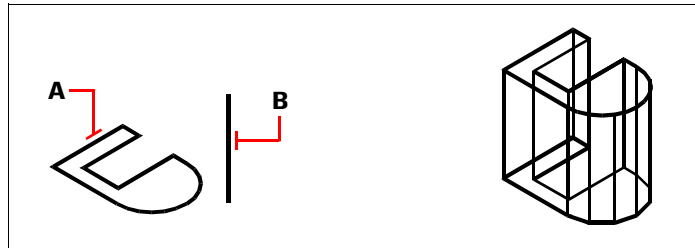
- Facet model meshes are smart meshes that can be modified with advanced features such as boolean operations. They are similar to solids but are more lightweight and don't have mass properties.
- Surface meshes are simple meshes.

To create an extruded facet model mesh

- 1 Do one of the following to choose Extrude ():
 - On the ribbon, choose Draw 3D > Extrude (in Facet Model Meshes).
 - On the menu, choose Draw > Facet Model Meshes > Extruded.
 - On the Facet Model Meshes toolbar, click the Extrude tool.
 - Type *fmextrude* and then press Enter.
- 2 Select the entity to extrude.
- 3 Select the extrusion path, specify the direction, or specify the height.

To create an extruded surface mesh

- 1 Do one of the following to choose Extruded Surface ():
 - On the ribbon, choose Draw 3D > Extruded Surface (in Draw 3D Meshes).
 - On the menu, choose Draw > 3D Meshes > Extruded Surface.
 - On the Draw 3D Meshes toolbar, click the Extruded Surface tool.
 - Type *ai_tabsurf* and then press Enter.
- 2 Select the entity to extrude.
- 3 Select the extrusion path.



Select the entity to extrude (A) and the extrusion path (B).

The resulting extruded surface mesh.

The value of the Number of M-Direction Surfaces controls the density of the mesh.

Choose *Tools > Drawing Settings*, click the *3D Settings* tab, click the *Surfaces* tab, then under *Surface Settings*, change the *Number Of M-Direction Surfaces*.

An extruded mesh is different from an extruded solid.

If you want to extrude an entity and convert it to a three-dimensional solid, use the *Extrude* command. Note that you can also use the *Extrude* command to create an extruded surface. For more details, see “Creating extruded solids or surfaces” on page 735 in this chapter.

Creating revolved meshes


You can create a revolved mesh, which is a three-dimensional polygon mesh that approximates the surface generated by rotating a two-dimensional profile around an axis. You select the two entities that define the profile and the axis. You also specify the starting angle and the number of degrees to revolve the profile.

Revolving the profile 360 degrees creates a closed three-dimensional mesh.


You can create two types of revolved meshes:

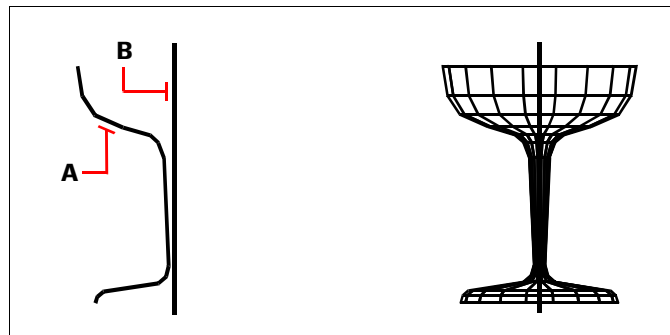
- Facet model meshes are smart meshes that can be modified with advanced features such as boolean operations. They are similar to solids but are more lightweight and don't have mass properties.
- Surface meshes are simple meshes. Note that the Number Of M-Direction Surfaces value determines the mesh density (the number of mesh segments) in the M-direction (around the axis of revolution). The N-Direction Mesh Density value determines the mesh density (the number of mesh segments) in the N-direction (along the axis of revolution).

To create a revolved facet model mesh

- 1 Do one of the following to choose Revolve (- On the ribbon, choose Draw 3D > Revolve (in Facet Model Meshes).
- On the menu, choose Draw > Facet Model Meshes > Revolve.
- On the Facet Model Meshes toolbar, click the Revolve tool.
- Type *fmrevolve* and then press Enter.
- 2 Select the entity to revolve.
- 3 Do one of the following to define the axis of revolution:
 - Specify a start point and an end point.
 - Choose Entity and press Enter to select an entity that determines the axis.
 - Choose Xaxis to select the x-axis.
 - Choose Yaxis to select the y-axis.
- 4 Specify the angle of revolution.

To create a revolved surface mesh

- 1 Do one of the following to choose Revolved Surface ():
 - On the ribbon, choose Draw 3D > Revolved Surface (in Draw 3D Meshes).
 - On the menu, choose Draw > 3D Meshes > Revolved Surface.
 - On the Draw 3D Meshes toolbar, click the Revolved Surface tool.
 - Type *revsurf* and then press Enter.
- 2 Select the entity to revolve.
- 3 Select the entity to be used as the axis of revolution.
- 4 Specify the starting angle.
- 5 Specify the number of degrees to revolve the entity.



Select the entity to be revolved (A) and the axis of revolution (B).

The resulting revolved surface mesh.

The values of the Number of M-Direction Surfaces and N-Direction Mesh Density control the density of the mesh.

Choose Tools > Drawing Settings, click the 3D Settings tab, click the Surfaces tab, then under Surface Settings, change the Number Of M-Direction Surfaces and N-Direction Mesh Density values.

Revolved surfaces can also be created using the Revolve command.


The Revolve command is typically used for creating revolved three-dimensional solids, but it can also be used to create a revolved surface. For more details, see “Creating revolved solids or surfaces” on page 736 in this chapter.

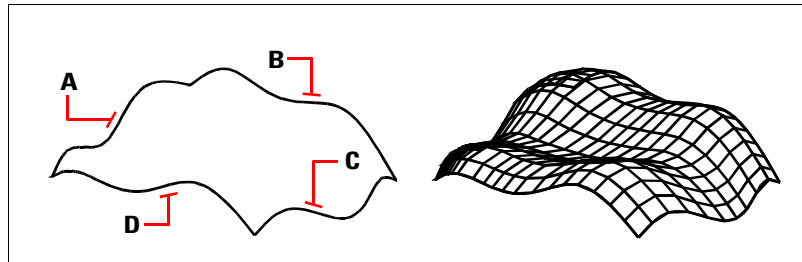
Creating edge-defined Coons surface patch meshes

You can create a surface called a Coons surface patch, which is a mesh connecting four edges. You select the entities that define the edges. Edge entities can be arcs, splines, lines, or polylines. The four edge entities must be touching and form a closed loop. A patch is a bicubic surface (one curve extends in the M-direction and the other in the N-direction) interpolated between the four adjoining edges. You can select the edges in any order. The first edge you select determines the M-direction of the mesh.

To create an edge-defined Coons surface patch mesh

Advanced experience level

- 1 Do one of the following to choose Coons Surface ():
 - On the ribbon, choose Draw 3D > Coons Surface (in Draw 3D Meshes).
 - On the menu, choose Draw > 3D Meshes > Coons Surface.
 - On the Draw 3D Meshes toolbar, click the Coons Surface tool.
 - Type *edgesurf* and then press Enter.
- 2 Select the first edge.
- 3 Select the second, third, and fourth edges.



Select the entities to be used as the four edges (A, B, C, and D).

The resulting Coons surface patch mesh.


The values of the Number of M-Direction Surfaces and N-Direction Mesh Density control the density of the mesh.

Choose Tools > Drawing Settings, click the 3D Settings tab, click the Surfaces tab, then under Surface Settings, change the Number Of M-Direction Surfaces and N-Direction Mesh Density values.


Creating boxes

You can create rectangular boxes, or cubes. A box consists of six rectangular surface planes. The base of the box is always parallel with the xy plane of the current UCS. You position the box by specifying either a corner or the center of the box. You determine the size of the box by either specifying a second corner and the height; defining the box to be a cube and then providing its length; or specifying the length, width, and height.


To create a box as an ACIS solid

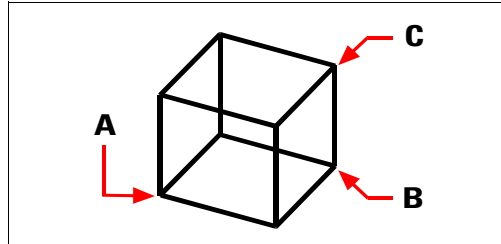
- 1 Do one of the following to choose Box ():
 - On the ribbon, choose Draw 3D > Box (in Draw 3D Solids).
 - On the menu, choose Draw > 3D Solids > Box.
 - On the Draw 3D Solids toolbar, click the Box tool.
 - Type *box* and then press Enter.
- 2 Specify the first corner of the base.
- 3 Specify the opposite corner of the base.
- 4 Specify the height.

To create a box as a Facet Model mesh

- 1 Do one of the following to choose Box ():
 - On the ribbon, choose Draw 3D > Box (in Facet Model Meshes).
 - On the menu, choose Draw > Facet Model Meshes > Box.
 - On the Facet Model Meshes toolbar, click the Box tool.
 - Type *fnbox* and then press Enter.
- 2 Specify the first corner of the base.
- 3 Specify the opposite corner of the base.
- 4 Specify the height.

To create a box as a 3D mesh

- 1 Do one of the following to choose Box ():
 - On the ribbon, choose Draw 3D > Box (in Draw 3D Meshes).
 - On the menu, choose Draw > 3D Meshes > Box.
 - On the Draw 3D Meshes toolbar, click the Box tool.
 - Type *ai_box* and then press enter.
- 2 Specify the first corner of the base.
- 3 Specify the opposite corner of the base.
- 4 Specify the height.




First corner of the base (A), the opposite corner of the base (B), and the height (C).


Creating wedges

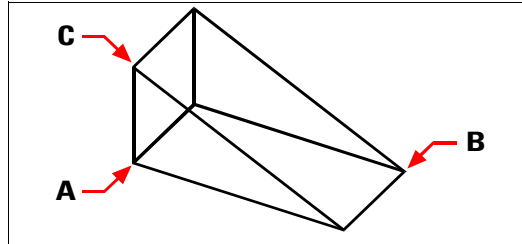
You can create three-dimensional wedges consisting of five surface planes. The base of the wedge is always parallel with the xy plane of the current UCS with the sloped face opposite the first corner. The height is always parallel with the z-axis. You position the wedge by specifying either a corner or the center of the wedge. You determine the size of the wedge by either specifying a second corner and the height; defining the wedge based on a cube having a given length; or specifying the length, width, and height.

To create a wedge as an ACIS solid

- 1 Do one of the following to choose Wedge ():
 - On the ribbon, choose Draw 3D > Wedge (in Draw 3D Solids).
 - On the menu, choose Draw > 3D Solids > Wedge.
 - On the Draw 3D Solids toolbar, click the Wedge tool.
 - Type *wedge* and then press Enter.
- 2 Specify the first corner of the base.
- 3 Specify the opposite corner of the base.
- 4 Specify the height.

To create a wedge as a 3D mesh

- 1 Do one of the following to choose Wedge ():
 - On the ribbon, choose Draw 3D > Wedge (in Draw 3D Meshes).
 - On the menu, choose Draw > 3D Meshes > Wedge.
 - On the Draw 3D Meshes toolbar, click the Wedge tool.
 - Type *ai_wedge* and then press Enter.
- 2 Specify the first corner of the base.
- 3 Specify the opposite corner of the base.
- 4 Specify the height.




First corner of the base (A), the opposite corner of the base (B), and the height (C).


Creating cones

You can create three-dimensional cones defined by a circular base and tapering to a point perpendicular to the base. The base of the cone is always parallel with the xy plane of the current UCS; the height of the cone is always parallel with the z-axis. You position the cone by specifying the center of the base. You determine the size of the cone by specifying either the radius or the diameter of the base and the height.


To create a cone as an ACIS solid

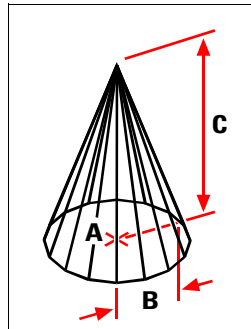
- 1 Do one of the following to choose Cone ():
 - On the ribbon, choose Draw 3D > Cone (in Draw 3D Solids).
 - On the menu, choose Draw > 3D Solids > Cone.
 - On the Draw 3D Solids toolbar, click the Cone tool.
 - Type *cone* and then press Enter.
- 2 Specify the center of the base of the cone.
- 3 Specify the radius or diameter.
- 4 Specify the height by doing one of the following:
 - Enter the height of the cone, or select a point to indicate the height.
 - Choose Apex to draw a cone with an offset top, and then select a point for the tip of the cone.
 - Choose 2Point to specify the height by picking two points, and then select the two points.

To create a cone as a Facet Model mesh

- 1 Do one of the following to choose Cone ():
 - On the ribbon, choose Draw 3D > Cone (in Facet Model Meshes).
 - On the menu, choose Draw > Facet Model Meshes > Cone.
 - On the Facet Model Meshes toolbar, click the Cone tool.
 - Type *fmcone* and then press Enter.
- 2 Specify the center of the base of the cone.
- 3 Specify the base radius or diameter.
- 4 Specify the height.

To create a cone as a 3D mesh

- 1 Do one of the following to choose Cone ():
 - On the ribbon, choose Draw 3D > Cone (in Draw 3D Meshes).
 - On the menu, choose Draw > 3D Meshes > Cone.
 - On the Draw 3D Meshes toolbar, click the Cone tool.
 - Type *ai_cone* and then press Enter.
- 2 Specify the center of the base of the cone.
- 3 Specify the radius or diameter.
- 4 Specify the height.




Center of the base (A), the radius of the base (B), and the height (C).


Creating pyramids

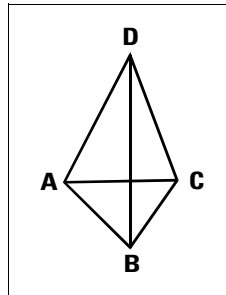
You can create tetrahedrons (three-sided pyramids) or four-sided pyramids. The sides of the resulting pyramid can meet at a point (the apex) or can form a three- or four-edged top. The sides of a four-sided pyramid can also meet along a ridge defined by two points. The base of the pyramid is always parallel with the xy plane of the current UCS. You position the pyramid by specifying a corner of the base. You determine the size of the pyramid by specifying the base points and either the apex, the corners of the top surface, or the endpoints of the ridge.

To create a tetrahedron as an ACIS solid

- 1 Do one of the following to choose Pyramid ():
 - On the ribbon, choose Draw 3D > Pyramid (in Draw 3D Solids).
 - On the menu, choose Draw > 3D Solids > Pyramid.
 - On the Draw 3D Solids toolbar, click the Pyramid tool.
 - Type *pyramid* and then press Enter.
- 2 Specify the first point for the base of the pyramid.
- 3 Specify the second and third points.
- 4 In the prompt box, choose Tetrahedron.
- 5 Specify the apex of the tetrahedron.


To create a tetrahedron as a 3D mesh

- 1 Do one of the following to choose Pyramid ():
 - On the ribbon, choose Draw 3D > Pyramid (in Draw 3D Meshes).
 - On the menu, choose Draw > 3D Meshes > Pyramid.
 - On the Draw 3D Meshes toolbar, click the Pyramid tool.
 - Type *ai_pyramid* and then press Enter.
- 2 Specify the first point for the base of the pyramid.
- 3 Specify the second and third points.
- 4 In the prompt box, choose Tetrahedron.
- 5 Specify the apex of the tetrahedron.




The first point (A), second point (B), and third point (C) of the base, and the apex (D).


To create a pyramid as a Facet Model mesh

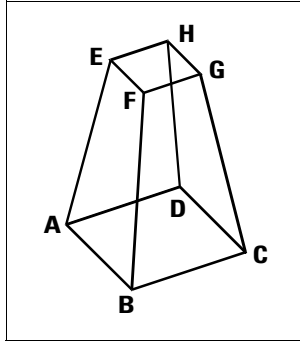
- 1 Do one of the following to choose Pyramid ():
 - On the ribbon, choose Draw 3D > Pyramid (in Facet Model Meshes).
 - On the menu, choose Draw > Facet Model > Pyramid.
 - On the Facet Model Meshes toolbar, click the Pyramid tool.
 - Type *fm_pyramid* and then press Enter.
- 2 Specify the center point for the base of the pyramid.
- 3 Specify the radius for the base of the pyramid.
- 4 Specify the height.

To create a pyramid with a planar top as an ACIS solid

- 1 Do one of the following to choose Pyramid ():
 - On the ribbon, choose Draw 3D > Pyramid (in Draw 3D Solids).
 - On the menu, choose Draw > 3D Solids > Pyramid.
 - On the Draw 3D Solids toolbar, click the Pyramid tool.
 - Type *pyramid* and then press Enter.
- 2 Specify the first point for the base of the pyramid.
- 3 Specify the second, third, and fourth points.
- 4 In the prompt box, choose Top Surface.
- 5 Specify the first point on the top surface of the pyramid.
- 6 Specify the second, third, and fourth points.

To create a pyramid with a planar top as a 3D mesh

- 1 Do one of the following to choose Pyramid ():
 - On the ribbon, choose Draw 3D > Pyramid (in Draw 3D Meshes).
 - On the menu, choose Draw > 3D Meshes > Pyramid.
 - On the Draw 3D Meshes toolbar, click the Pyramid tool.
 - Type *ai_pyramid* and then press Enter.
- 2 Specify the first point for the base of the pyramid.
- 3 Specify the second, third, and fourth points.
- 4 In the prompt box, choose Top Surface.
- 5 Specify the first point on the top surface of the pyramid.
- 6 Specify the second, third, and fourth points.




The first point (A), second point (B), third point (C), and fourth point (D) of the base, and the first point (E), second point (F), third point (G), and fourth point (H) of the top surface.


Creating cylinders

You can create cylinders defined by a circular base. The base of a cylinder is always parallel with the xy plane of the current UCS; the height of a cylinder is always parallel with the z-axis. You position a cylinder by specifying the center of the base. You determine the size of a cylinder by specifying either the radius or diameter of the base and the height.


To create a cylinder as an ACIS solid

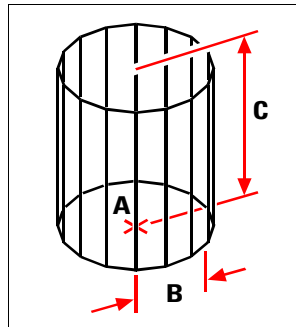
- 1 Do one of the following to choose Cylinder ():
 - On the ribbon, choose Draw 3D > Cylinder (in Draw 3D Solids).
 - On the menu, choose Draw > 3D Solids > Cylinder.
 - On the Draw 3D Solids toolbar, click the Cylinder tool.
 - Type *cylinder* and then press Enter.
- 2 Specify the center of the base of the cylinder.
- 3 Specify the radius or diameter.
- 4 Specify the height by doing one of the following:
 - Enter the height of the cylinder, or select a point to indicate the height.
 - Choose Center of Second End to draw a cylinder with a slanted wall, and then specify the center point for the second end.
 - Choose 2Point to specify the height by picking two points, and then select the two points.

To create a cylinder as a Facet Model mesh

- 1 Do one of the following to choose Cylinder ():
 - On the ribbon, choose Draw 3D > Cylinder (in Facet Model Meshes).
 - On the menu, choose Draw > Facet Model > Cylinder.
 - On the Facet Model Meshes toolbar, click the Cylinder tool.
 - Type *fmcylinder* and then press Enter.
- 2 Specify the center of the base of the cylinder.
- 3 Specify the radius or diameter.
- 4 Specify the height.

To create a cylinder as a 3D mesh

- 1 Do one of the following to choose Cylinder ():
 - On the ribbon, choose Draw 3D > Cylinder (in Draw 3D Meshes).
 - On the menu, choose Draw > 3D Meshes > Cylinder.
 - On the Draw 3D Meshes toolbar, click the Cylinder tool.
 - Type *ai_cylinder* and then press Enter.
- 2 Specify the center of the base of the cylinder.
- 3 Specify the radius or diameter.
- 4 Specify the height.




Center of the base (A), radius of the base (B), and the height (C).


Creating spheres

You can create spheres. The latitude lines of a sphere are always parallel with the xy plane of the current UCS; the central axis is always parallel with the z-axis. You position a sphere by specifying its center point. You determine the size of a sphere by specifying either its radius or its diameter.


To create a sphere as an ACIS solid

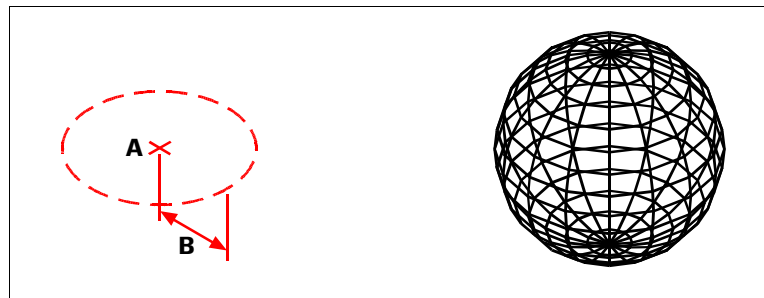
- 1 Do one of the following to choose Sphere ():
 - On the ribbon, choose Draw 3D > Sphere (in Draw 3D Solids).
 - On the menu, choose Draw > 3D Solids > Sphere.
 - On the Draw 3D Solids toolbar, click the Sphere tool.
 - Type *sphere* and then press Enter.
- 2 Specify the center of the sphere.
- 3 Specify the radius or diameter.

To create a sphere as a Facet Model mesh

- 1 Do one of the following to choose Sphere ():
 - On the ribbon, choose Draw 3D > Sphere (in Facet Model Meshes).
 - On the menu, choose Draw > Facet Model > Sphere.
 - On the Facet Model Meshes toolbar, click the Sphere tool.
 - Type *fmsphere* and then press Enter.
- 2 Specify the center of the sphere.
- 3 Specify the radius or diameter.

To create a sphere as a 3D mesh

- 1 Do one of the following to choose Sphere ():
 - On the ribbon, choose Draw 3D > Sphere (in Draw 3D Meshes).
 - On the menu, choose Draw > 3D Meshes > Sphere.
 - On the Draw 3D Meshes toolbar, click the Sphere tool.
 - Type *ai_sphere* and then press Enter.
- 2 Specify the center of the sphere.
- 3 Specify the radius or diameter.
- 4 Specify the number of longitudinal sections that are perpendicular to the xy plane.
- 5 Specify the number of latitudinal sections that are parallel to the xy plane.




Center (A) and radius (B) of the sphere.

The resulting sphere.


Creating dishes

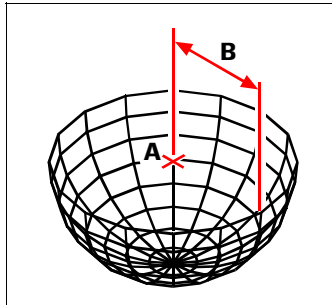
You can create a three-dimensional dish. The latitude lines of a dish are always parallel with the xy plane of the current UCS; the central axis is always parallel with the z-axis. You position a dish by specifying its center point. You determine the size of a dish by specifying either its radius or its diameter.

To create a dish as an ACIS solid

- 1 Do one of the following to choose Dish ():
 - On the ribbon, choose Draw 3D > Dish (in Draw 3D Solids).
 - On the menu, choose Draw > 3D Solids > Dish.
 - On the Draw 3D Solids toolbar, click the Dish tool.
 - Type *dish* and then press Enter.
- 2 Specify the center of the dish.
- 3 Specify the radius or diameter.

To create a dish as a 3D mesh

- 1 Do one of the following to choose Dish ():
 - On the ribbon, choose Draw 3D > Dish (in Draw 3D Meshes).
 - On the menu, choose Draw > 3D Meshes > Dish.
 - On the Draw 3D Meshes toolbar, click the Dish tool.
 - Type *ai_dish* and then press Enter.
- 2 Specify the center of the dish.
- 3 Specify the radius or diameter.



Center (A) and radius (B) of the dish.

Creating domes

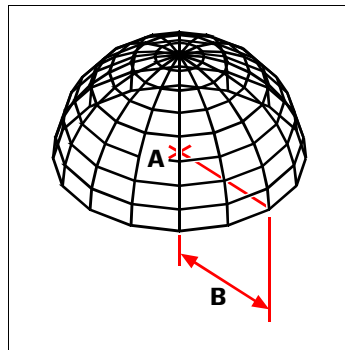
You can create a three-dimensional dome. The latitude lines of a dome are always parallel with the xy plane of the current UCS; the central axis is always parallel with the z-axis. You position a dome by specifying its center point. You determine the size of a dome by specifying either its radius or its diameter.

To create a dome as an ACIS solid

- 1 Do one of the following to choose Dome (🟢):
 - On the ribbon, choose Draw 3D > Dome (in Draw 3D Solids).
 - On the menu, choose Draw > 3D Solids > Dome.
 - On the Draw 3D Solids toolbar, click the Dome tool.
 - Type *dome* and then press Enter.
- 2 Specify the center of the dome.
- 3 Specify the radius or diameter.

To create a dome as a 3D mesh

- 1 Do one of the following to choose Dome (🟡):
 - On the ribbon, choose Draw 3D > Dome (in Draw 3D Meshes).
 - On the menu, choose Draw > 3D Meshes > Dome.
 - On the Draw 3D Meshes toolbar, click the Dome tool.
 - Type *ai_dome* and then press Enter.
- 2 Specify the center of the dome.
- 3 Specify the radius or diameter.




Center (A) and radius (B) of the dome.


Creating tori

You can create a three-dimensional donut or ring-shaped entity known as a torus. The diameter of a ring is always parallel with the xy plane of the current UCS. A torus is constructed by revolving a circle about a line drawn in the plane of the circle and parallel with the z-axis of the current UCS. You position a torus by specifying its center point. You determine the size of a torus by specifying its overall diameter or radius and the diameter or radius of the tube (the circle being revolved).


To create a torus as an ACIS solid

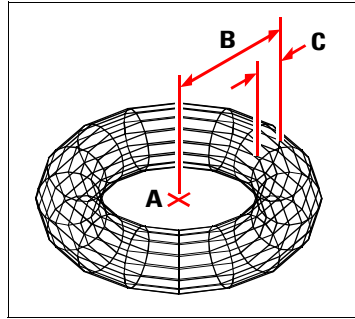
- 1 Do one of the following to choose Torus ():
 - On the ribbon, choose Draw 3D > Torus (in Draw 3D Solids).
 - On the menu, choose Draw > 3D Solids > Torus.
 - On the Draw 3D Solids toolbar, click the Torus tool.
 - Type *torus* and then press Enter.
- 2 Specify the center of the whole torus.
- 3 Specify the radius or diameter of the whole torus.
- 4 Specify the radius or diameter of the body of the torus.

To create a torus as a Facet Model mesh

- 1 Do one of the following to choose Torus ():
 - On the ribbon, choose Draw 3D > Torus (in Facet Model Meshes).
 - On the menu, choose Draw > Facet Model Meshes > Torus.
 - On the Facet Model Meshes toolbar, click the Torus tool.
 - Type *fmtorus* and then press Enter.
- 2 Specify the center of the whole torus.
- 3 Specify the radius or diameter of the whole torus.
- 4 Specify the radius or diameter of the body (tube) of the torus.

To create a torus as a 3D mesh

- 1 Do one of the following to choose Torus ():
 - On the ribbon, choose Draw 3D > Torus (in Draw 3D Meshes).
 - On the menu, choose Draw > 3D Meshes > Torus.
 - On the Draw 3D Meshes toolbar, click the Torus tool.
 - Type *ai_torus* and then press Enter.
- 2 Specify the center of the whole torus.
- 3 Specify the radius or diameter of the whole torus.
- 4 Specify the radius or diameter of the body of the torus.
- 5 Specify the number of longitudinal sections that are perpendicular to the xy plane.
- 6 Specify the number of latitudinal sections that are parallel to the xy plane.




Center (A) and radius of the whole torus (B), and the radius of the body (C).

Creating pipes

You can create a three-dimensional pipe, which is a hollow cylinder or tube. Pipes are created by specifying the base center, radius or diameter, wall thickness, and length. Pipes are created as Facet Model meshes.


To create a pipe as a Facet Model mesh

- 1 Do one of the following to choose Pipe ():
 - On the ribbon, choose Draw 3D > Pipe (in Facet Model Meshes).
 - On the menu, choose Draw > Facet Model Meshes > Pipe.
 - On the Facet Model Meshes toolbar, click the Pipe tool.
 - Type *fmpipe* and then press Enter.
- 2 Specify the base center of the pipe.
- 3 Do one of the following:
 - Enter the radius, or specify a second point to indicate the radius.
 - Choose Diameter, and then enter the diameter, or specify a second point to indicate the diameter.
- 4 Enter the thickness of the pipe wall, or choose 2Point to specify two points that indicate the thickness.
- 5 Enter the length, or specify a point to indicate the length.

Creating helices

You can create a three-dimensional helix, which is a spiral-shaped entity, as a facet model mesh. Helices are created by specifying the base, top, turns, and height or axis endpoint.

To create a helix as a Facet Model mesh

- 1 Do one of the following to choose Helix ():
 - On the ribbon, choose Draw 3D > Helix (in Facet Model Meshes).
 - On the menu, choose Draw > Facet Model Meshes > Helix.
 - On the Facet Model Meshes toolbar, click the Helix tool.
 - Type *fmhelix* and then press Enter.
- 2 Specify the radius of the helix body (tube).
- 3 Specify the center point of the helix base.
- 4 Do one of the following to specify the bottom of the helix:
 - Enter the radius, or specify a second point to indicate the radius.
 - Choose Diameter, and then enter the diameter or specify a second point to indicate the diameter.
- 5 Do one of the following to specify the top of the helix:
 - Enter the radius, or specify a second point to indicate the radius.
 - Choose Diameter, and then enter the diameter or specify a second point to indicate the diameter.
- 6 Optionally define the turns of the helix:
 - Choose Turns, then enter the number of full turns. The maximum is 500.
 - Choose turn Height, then specify the height of one full turn.
 - Choose tWist, then specify the direction of helix turns, which can be clockwise (CW) or counterclockwise (CCW).
- 7 Do one of the following:
 - Specify the height of the helix.
 - Choose Axis endpoint, then specify the endpoint of the helix, which determines the position and direction of the helix.


Creating regions

You can convert a closed entity into a two-dimensional region. After you create a region, you can modify it using the various three-dimensional tools. For example, you can create a region from a square, and then extrude the square to create a three-dimensional cube.

You can create regions from closed entities, such as polylines, polygons, circles, ellipses, closed splines, and donuts. You can also select open entities such as lines and arcs, if they have common endpoints and they form a closed area.

Creating regions typically has no visible effect on a drawing. However, if the original entity had a width or lineweight, that information is lost when you create the region.

To create a region

- 1 Do one of the following to choose Region (

The command bar displays a message that describes how many regions were created.


The Boundary command can also be used to create regions.

For more details, see “Defining boundaries” on page 209.


Creating planar surfaces

You can create planar surfaces by drawing a rectangle or selecting existing closed, planar entities such as polylines, polygons, circles, ellipses, closed splines, and donuts. You can also select open entities such as lines and arcs, if they have common endpoints and they form a closed area.

To create a rectangular planar surface

- 1 Do one of the following to choose Planar Surface ():
 - On the ribbon, choose Draw 3D > Planar Surface (in Surfaces).
 - On the menu, choose Draw > Surfaces > Planar Surface.
 - On the Draw 3D Surfaces toolbar, click the Planar Surface tool.
 - Type *planesurf* and then press Enter.
- 2 Specify a point for the first corner.
- 3 Specify a point for the opposite corner.



To create a planar surface from existing entities

- 1 Do one of the following to choose Planar Surface ():
 - On the ribbon, choose Draw 3D > Planar Surface (in Surfaces).
 - On the menu, choose Draw > Surfaces > Planar Surface.
 - On the Draw 3D Surfaces toolbar, click the Planar Surface tool.
 - Type *planesurf* and then press Enter.
- 2 Choose Entity.
- 3 Select closed, planar entities, then press Enter.

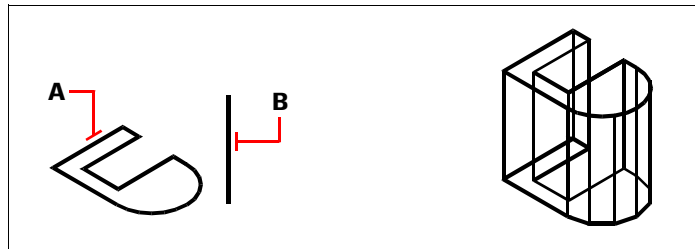
Creating extruded solids or surfaces

You can create three-dimensional solids or surfaces by extruding closed entities, such as polylines, polygons, circles, ellipses, closed splines, donuts, and regions. You can extrude the entity along a selected path, or you can specify its height and taper angle.

To create an extruded solid or surface

- 1 Do one of the following to choose Extrude () ():
 - On the ribbon, choose Draw 3D > Extrude (in Draw 3D Solids or in Surfaces).
 - On the menu, choose Draw > 3D Solids > Extrude or Draw > Surfaces > Extrude.
 - On the Draw 3D Solids or the Draw 3D Surfaces toolbar, click the Extrude tool.
 - Type *extrude* and then press Enter.
- 2 Select the entity to extrude.

If you want to create an extruded surface instead of a solid, choose Mode. If an open profile is selected, a surface is created even if Mode is set to Solid.
- 3 Select the extrusion path, or specify the height.





Select the entity to extrude (A) and the extrusion path (B).

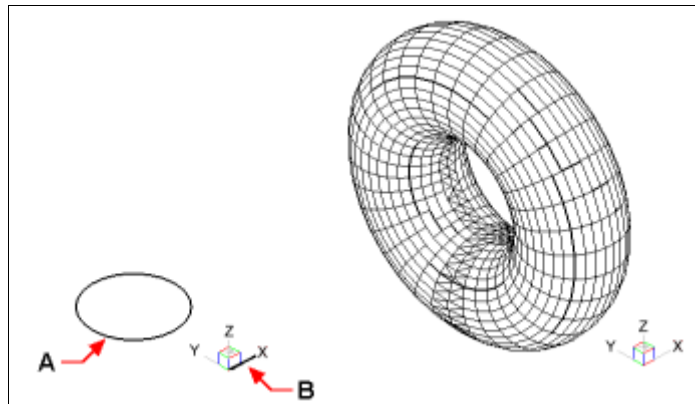
The resulting extruded solid.

Creating revolved solids or surfaces

You can create three-dimensional solids or surfaces by revolving closed entities, such as polylines, polygons, circles, ellipses, and regions. You can revolve the entity about a defined axis, line, polyline, or two points.

To create a revolved solid or surface

- 1 Do one of the following to choose Revolve () ():
 - On the ribbon, choose Draw 3D > Revolve (in Draw 3D Solids or in Surfaces).
 - On the menu, choose Draw > 3D Solids > Revolve or Draw > Surfaces > Revolve.
 - On the Draw 3D Solids or the Draw 3D Surfaces toolbar, click the Revolve tool.
 - Type *revolve* and then press Enter.
- 2 Select the entity to revolve.
If you want to create a revolved surface instead of a solid, choose Mode. If an open profile is selected, a surface is created even if Mode is set to Solid.
- 3 Do one of the following to define the axis of revolution:
 - Specify a start point and an end point.
 - Type *e* and press Enter to select an entity that determines the axis.
 - Type *x* and press Enter to select the x-axis.
 - Type *y* and press Enter to select the y-axis.
- 4 Specify the angle of revolution.



Select the entity to revolve (A) and the axis (B) about which to revolve it, followed by the angle of revolution.

The resulting revolved solid.

Creating lofted solids and surfaces



Creates a three-dimensional solid or surface between two or more cross sections. Cross sections can be open or closed entities. Open cross sections create three-dimensional surfaces. Closed cross sections create three-dimensional solids or surfaces, depending on the specified mode.

Cross sections can be 2D polylines, lines, arcs, circles, ellipses, elliptical arcs, 2D splines, helices, traces, edges of entities, faces of a solid or surface, points of the first or last cross section, regions, and 2D solids.

Guides can be 2D polylines with a single segment, 3D polylines, lines, arcs, elliptical arcs, 2D and 3D splines, and edges of entities.

Paths can be 2D and 3D polylines, lines, arcs, circles, ellipses, elliptical arcs, 2D and 3D splines, helices, and edges of entities.

To create a lofted solid or surface

- 1 Do one of the following to choose Loft ( ):
 - On the ribbon, choose Draw 3D > Loft (in Draw 3D Solids or in Surfaces).
 - On the menu, choose Draw > 3D Solids > Loft or Draw > Surfaces > Loft.
 - On the Draw 3D Solids or the Draw 3D Surfaces toolbar, click the Loft tool.
 - Type *loft* and then press Enter.
- 2 Select the cross sections in the order you want them to loft. You must select at least two cross sections.
- 3 If desired, choose Point to taper the loft entity. Select the taper point on a cross section, then press Enter.
- 4 If desired, choose Join multiple edges to create a cross section from edges. Select the edges, which must share start and end points, to be considered as a cross section. When done selecting edges, press Enter.
- 5 If needed, choose Mode to change whether a three-dimensional solid or surface is created, then press enter.
- 6 Press Enter to continue.
- 7 Choose one of the following:
 - Guides — Creates the loft entity using guide curves which help shape the entity. Select the guide curves for the loft entity. Guide curves must intersect each cross section and begin and end at the first and last cross section. You can also combine multiple edges to form a guide.
 - Path — Creates the loft entity along a path. Select the path for the lofted entity. The path must intersect the solid or surface on all planes.
 - Cross sections only — Creates the loft entity between the cross sections without using guides or paths.
 - Settings — Opens the Loft Settings dialog box to specify various settings.



Creating swept solids and surfaces

You can create three-dimensional solids or surfaces by sweeping an entity along a path.

Entities that you can sweep include 2D polylines, lines, arcs, circles, ellipses, elliptical arcs, 2D and 3D splines, 3D solid faces, and 2D solids.

Entities that can be the path include 2D and 3D polylines, lines, arcs, circles, ellipses, elliptical arcs, 2D and 3D splines, helices, and edges of solids, surfaces, or meshes.


To create a swept solid or surface

- 1 Do one of the following to choose Sweep () ():
 - On the ribbon, choose Draw 3D > Sweep (in Draw 3D Solids or in Surfaces).
 - On the menu, choose Draw > 3D Solids > Sweep or Draw > Surfaces > Sweep.
 - On the Draw 3D Solids or the Draw 3D Surfaces toolbar, click the Sweep tool.
 - Type *sweep* and then press Enter.
- 2 Select one or more entities to sweep.
You can choose Mode to change whether a three-dimensional solid or surface is created.
- 3 Select the path.
- 4 If desired, choose any of the following options:
 - Alignment — Aligns the sweep entity to be perpendicular (normal) to the tangent direction of the sweep path.
 - Base point — Determines the base point of the sweep entity.
 - Scale — Determines the scale factor for the sweep entity. If there are multiple sweep entities, the scale factor is applied to each entity. You can also choose Reference to select reference points in the drawing for scaling.
 - Twist — Determines the degrees in which to rotate the sweep entity along the path. If there are multiple sweep entities, the twist angle is applied to each entity. You can also choose Bank to determine whether the sweep entity being swept rotates along a 3D path such as a 3D polyline, spline, or helix.


Creating polysolids

You can create three-dimensional solids with a rectangular profile from a new polyline that you draw or from an existing line, arc, polyline, or circle.

To create a polysolid without converting any entities

- 1 Do one of the following to choose Polysolid (- On the ribbon, choose Draw 3D > Polysolid (in Draw 3D Solids).
- On the menu, choose Draw > 3D Solids > Polysolid.
- On the Draw 3D Solids toolbar, click the Polysolid tool.
- Type *polysolid* and then press Enter.
- 2 Select the start point of the polysolid.
- 3 Continue selecting points. You can choose Arc to create an arc segment. You can also choose Undo to erase the previous point.
- 4 Choose Height and specify the height of the polysolid. The default height is specified by the PSOLHEIGHT system variable.
- 5 Choose width and specify the width of the polysolid. The default height is specified by the PSOLWIDTH system variable.
- 6 Choose Justify to specify whether the width and height should be justified to the left, center, or right. Justification orientation is determined according to the direction of the first profile segment.

To create a polysolid from an existing entity

- 1 Do one of the following to choose Polysolid (- On the ribbon, choose Draw 3D > Polysolid (in Draw 3D Solids).
- On the menu, choose Draw > 3D Solids > Polysolid.
- On the Draw 3D Solids toolbar, click the Polysolid tool.
- Type *polysolid* and then press Enter.
- 2 Choose Entity, then press Enter.
- 3 Select the entity you want to convert to a polysolid. You can select a line, arc, polyline, or circle.
- 4 Choose Height and specify the height of the polysolid. The default height is specified by the PSOLHEIGHT system variable.
- 5 Choose width and specify the width of the polysolid. The default height is specified by the PSOLWIDTH system variable.
- 6 Choose Justify to specify whether the width and height should be justified to the left, center, or right. Justification is determined according to the direction of the first profile segment.

Creating composite three-dimensional entities

You can create composite three-dimensional facet model meshes by combining, subtracting, and finding the intersection of two or more facet model meshes.


Composite three-dimensional entities can also be created using 3D solids.

Combining three-dimensional entities


Combine two or more facet model meshes to create one facet model mesh.

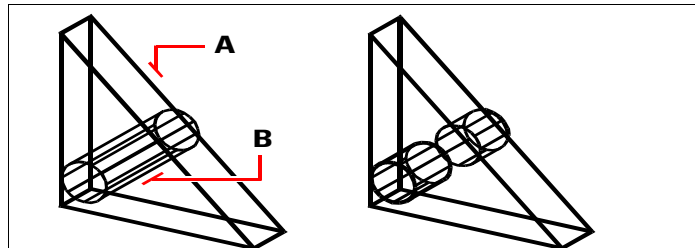
You can also combine two or more 3D solids.

To combine facet model meshes

- 1 Do one of the following to choose Union ():
 - On the ribbon, choose Draw 3D > Union (in Facet Model Meshes).
 - On the menu, choose Draw > Facet Model Meshes > Union.
 - On the Facet Model Meshes toolbar, click the Union tool.
 - Type *fmunion* and then press Enter.
- 2 Select the facet model meshes to combine.

To combine solids

- 1 Do one of the following to choose Union ():
 - On the ribbon, choose Draw 3D > Union (in Solid Editing).
 - On the menu, choose Modify > Solid Editing > Union.
 - On the Solids Editing toolbar, click the Union tool.
 - Type *union* and then press Enter.
- 2 Select the 3D solids to combine.



Select the entities to combine (A and B).


The resulting combined entity.

Subtracting three-dimensional entities


Subtract two or more facet model meshes to create one facet model mesh.

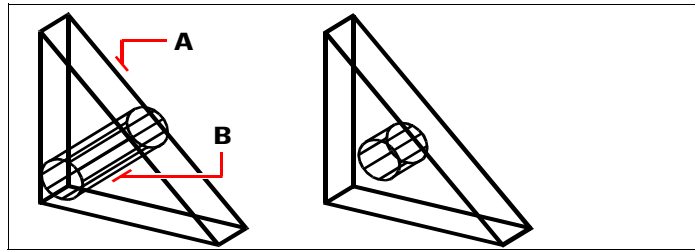
You can also subtract two or more 3D solids.

To subtract facet model meshes

- 1 Do one of the following to choose Subtract ():
 - On the ribbon, choose Draw 3D > Subtract (in Facet Model Meshes).
 - On the menu, choose Draw > Facet Model Meshes > Subtract.
 - On the Facet Model Meshes toolbar, click the Subtract tool.
 - Type *fmsubtract* and then press Enter.
- 2 Select the facet model meshes to subtract from one another.

To subtract solids

- 1 Do one of the following to choose Subtract ():
 - On the ribbon, choose Draw 3D > Subtract (in Solid Editing).
 - On the menu, choose Modify > Solid Editing > Subtract.
 - On the Solids Editing toolbar, click the Subtract tool.
 - Type *subtract* and then press Enter.
- 2 Select the 3D solids to subtract from one another.



Select the entities to subtract (A and B).


The resulting entity.

Intersecting three-dimensional entities


Create a composite three-dimensional entity from the intersection of two or more facet model meshes. Areas outside of the intersection are removed.

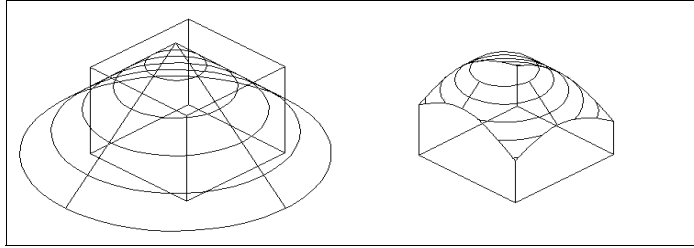
You can also intersect two or more 3D solids.

To intersect facet model meshes

- 1 Do one of the following to choose Intersect ():
 - On the ribbon, choose Draw 3D > Intersect (in Facet Model Meshes).
 - On the menu, choose Draw > Facet Model Meshes > Intersect.
 - On the Facet Model Meshes toolbar, click the Intersect tool.
 - Type *fmintersect* and then press Enter.
- 2 Select the facet model meshes to intersect.

To intersect solids

- 1 Do one of the following to choose Intersect ():
 - On the ribbon, choose Draw 3D > Intersect (in Solid Editing).
 - On the menu, choose Modify > Solid Editing > Intersect.
 - On the Solids Editing toolbar, click the Intersect tool.
 - Type *intersect* and then press Enter.
- 2 Select the 3D solids to intersect.



Select the entities to combine (A and B).


The resulting combined entity.

Creating three-dimensional entities from extracted curves


You can create curves from the contours of three-dimensional surfaces—and from solids if your program version supports them—and use those curves to experiment with non-mesh shapes. The resulting curves depend on the original entity's shape and can be lines, polylines, arcs, circles, or splines.

When finished experimenting with the resulting curves, you can use the Loft command to create a solid or surface from the revised curves.

To extract curves

- 1 Do one of the following to choose Extract Isolines ():
 - On the ribbon, choose Draw 3D > Extract Isolines (in Curves).
 - On the menu, choose Modify > 3D Operations > Extract Isolines.
 - Type *surfextractcurve* and then press Enter.
- 2 Select the surface, solid, or solid face that you want to extract curves from. To select a solid face, press Ctrl + click.
- 3 Optionally choose any of the following:
 - **Chain** Extracts curves from adjacent faces of the entity you select.
 - **Direction** Changes the direction of the extracted curves. By default, curves are created in the U direction.
- 4 Move the mouse over the selected entity, and then click to create the curve that is being previewed.
- 5 Continue creating curves, and press Enter when done.

To extract splines that pass through all specified points on the surface

- 1 Do one of the following to choose Extract Isolines ():
 - On the ribbon, choose Draw 3D > Extract Isolines (in Curves).
 - On the menu, choose Modify > 3D Operations > Extract Isolines.
 - Type *surfextractcurve* and then press Enter.
- 2 Select the surface, solid, or solid face that you want to create a custom curve from. To select a solid face, press Ctrl + click.
- 3 Choose Spline points.
- 4 Specify a point for the spline to pass through.
- 5 Continue specifying points, and then press Enter when done.

Converting three-dimensional entities

Various types of three-dimensional entities can be converted into other entity types:


- Convert to 3D surfaces — Select closed 2D entities, 3D solids, regions, lines, arcs and open polylines with thickness, mesh entities and 3D faces to convert to 3D surfaces.
- Convert to 3D meshes — Select 3D solids, 3D faces, polyface meshes and polygon meshes, regions, and closed polylines to convert to 3D meshes.
- Convert to polyface meshes — Select 3D solids to convert to polyface meshes.
- Convert to 3D solids — Select various entity types and convert to 3D solids.

Converting solids to polyface meshes

Converting three-dimensional entities to surfaces

You can convert the following entities into three-dimensional surfaces: closed 2D entities, 3D solids, regions, lines, arcs and open polylines with thickness, mesh entities and 3D faces.

To convert a three-dimensional entity to a surface

- 1 Do one of the following to choose Convert to Surface (

Converting three-dimensional entities to meshes

You can convert the following entities into three-dimensional surfaces: 3D solids, 3D faces, polyface meshes and polygon meshes, regions, and closed polylines.

To convert a three-dimensional entity to a mesh


- 1 Do one of the following to choose Convert to Mesh (

- 2 Choose Settings to view or change settings for the conversion:
 - **Show Edges** Click this check box to display the edges that form the mesh.
 - **Subdivide Level** Enter a value from 0 to 3 to determine the level of detail. The higher the number, the greater the detail applied to the converted mesh.
- 3 Click OK.
- 4 Select the entities you want to convert, then press Enter when done.

Converting solids to polyface meshes

You can convert three-dimensional solids to polyface meshes using the 3D Convert command.

To convert a solid

- 1 Do one of the following to choose 3D Convert ():
 - On the ribbon, choose Draw 3D > 3D Convert (in Convert).
 - On the menu, choose Modify > 3D Convert > 3D Convert.
 - On the Solids Editing toolbar, click the 3D Convert tool.
 - Type *3dconvert* and press Enter.
- 2 Select the entities you want to convert.

Converting surfaces and solids to NURBS surfaces


Use the Convert to NURBS command to convert solids and surfaces that are created with the Convert to Surface command to NURBS surfaces.

Meshes created with the Convert to Mesh command can also be converted to NURBS surfaces if you use the Convert to Surface or Convert to Solid command first.

Select a line to convert it to a spline.

The Convert to NURBS command is typically used to convert solids and surfaces to NURBS surfaces, but it can also be used to convert two-dimensional entities (for example, lines, polylines, arcs, circles, etc.) to splines.

To convert entities to NURBS surfaces

- 1 Do one of the following to choose Convert to NURBS ():
 - On the ribbon, choose Draw 3D > Convert to NURBS (in Convert).
 - On the menu, choose Modify > 3D Convert > Convert to NURBS.
 - On the Solids Editing toolbar, click the Convert to NURBS tool.
 - Type *convtonurbs* and press Enter.
- 2 Select the entities you want to convert.

NURBS surfaces can display with control vertices.


Use the *Show CV* command to display control vertices; use the *Hide CV* command to hide them.

Converting entities to solids


The Convert to Solid command converts several types of entities to solids: closed mesh entities, closed 3D surfaces, and closed circles and polylines with thickness.

You can also apply thickness to surface and mesh entities and convert them to three-dimensional solids using the Thicken command.

To convert closed entities to a solid using Convert to Solid

- 1 Do one of the following to choose Convert to Solid (- On the ribbon, choose Draw 3D > Convert to Solid (in Solid Editing).
- On the menu, choose Modify > Solid Editing > Convert to Solid.
- On the Solids Editing toolbar, click the Convert to Solid tool.
- Type *convtosolid* and press Enter.
- 2 Select the entities that you want to convert to solids, which can include:
 - Closed meshes
 - Closed 3D surfaces
 - Circles with thickness
 - Closed polylines with thickness and a non-changing width
- 3 When you are finished selecting entities, press Enter.

To convert a surface or mesh to a solid using Thicken

- 1 Do one of the following to choose Thicken (- On the ribbon, choose Draw 3D > Thicken (in Solid Editing).
- On the menu, choose Modify > Solid Editing > Thicken.
- On the Solids Editing toolbar, click the Thicken tool.
- Type *thicken* and press Enter.
- 2 Select the surface or mesh entity you want to convert and apply thickness to.
- 3 If you selected a mesh entity, choose Facet Surface. The mesh is converted to a facet surface first.
- 4 Enter the thickness value.

Editing in three dimensions


You can copy, move, rotate, array, mirror, and align two-dimensional and three-dimensional entities in both two-dimensional and three-dimensional space. You can also change and edit properties of three-dimensional entities much like you change and edit properties for two-dimensional entities. When you modify three-dimensional entities in two-dimensional space, you modify the entity in relation to the current UCS.

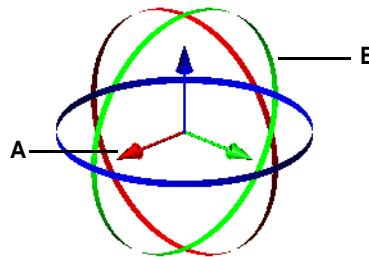
Moving and rotating in three dimensions

Use the 3D Positioner command to move and rotate entities interactively in three dimensional space.

A visual aid, or gizmo, displays. Click one of its arrows to move entities along the corresponding x, y, or z-axis, or click one of its orbits to rotate entities relative to the corresponding axis. Movement is restricted to the axis you select.

To move or rotate entities in three dimensions

- 1 Do one of the following to choose 3D Positioner ():
 - On the ribbon, choose Home > 3D Positioner (in Modify); Edit > 3D Positioner (in Modify).
 - On the menu, choose Modify > 3D Operations > 3D Positioner.
 - On the Modify toolbar, click the 3D Positioner tool.
 - Type *3dpositioner* and then press Enter.
- 2 Select the entities you want to move or rotate, and then press Enter.
- 3 Specify a point relative to which the selected entities are moved or rotated.
- 4 To move selected entities, click the arrow that corresponds to the axis you want to move along. To rotate selected entities, click the orbit that corresponds to the axis you want to rotate around.
- 5 Specify a new location.




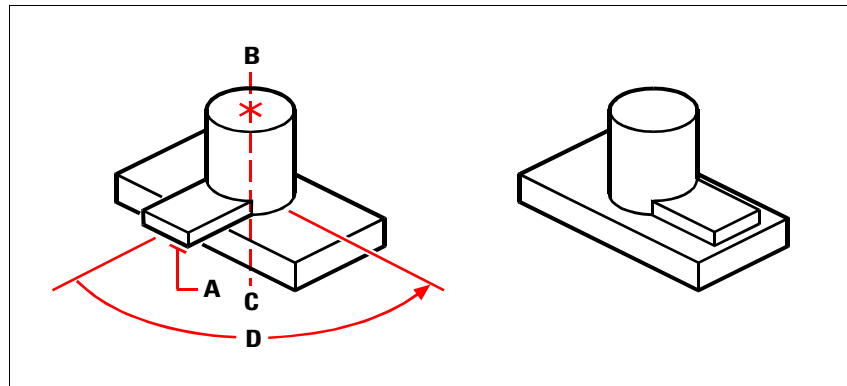
A Click and drag an arrow to move selected entities along the corresponding axis.

B Click and drag an orbit to rotate selected entities relative to the corresponding axis.

Another way to rotate entities about a specified axis in three dimensional space is using the 3D Rotate command. First you select the entities to rotate and then define the axis of rotation either by specifying two points; selecting an existing entity; aligning the axis with the x-, y-, or z-axis of the current UCS; or aligning the axis with the current view.

To rotate an entity in three dimensions using the 3D Rotate command

- 1 Do one of the following to choose 3D Rotate ():
 - On the ribbon, choose Edit > 3D Rotate (in Modify).
 - On the menu, choose Modify > 3D Rotate.
 - On the Modify toolbar, click the 3D Rotate tool.
 - Type *rotate3D* and then press Enter.
- 2 Select the entities to rotate, and then press Enter.
- 3 Choose from one of the following options: Entity, Last, View, Xaxis, Yaxis, Zaxis.
- 4 Specify the rotation angle.
- 5 Specify the reference angle.



Select the entities to rotate (A), specify the endpoints of the axis of rotation (B and C), and then specify the rotation angle (D).

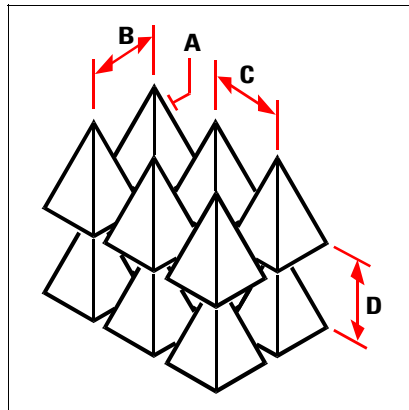
Result after rotating the entities.

Arraying in three dimensions

You can copy selected entities and arrange them in a three-dimensional rectangular or polar (circular) pattern. For a rectangular array, you control the number of copies in the array by specifying the number of rows and columns and the number of levels. You also specify the distance between each. For a polar array, you specify the axis around which to array the entities, the number of copies of the entities to create, and the angle subtended by the resulting array.


To create a three-dimensional rectangular array

- 1 Do one of the following to choose 3D Array:
 - On the ribbon, choose Edit > 3D Array (in Modify).
 - On the menu, choose Modify > 3D Array.
 - On the Modify toolbar, click the 3D Array tool.
 - Type *3Darray* and then press Enter.
- 2 Select the entities, and then press Enter.
- 3 In the prompt box, choose Rectangular.
- 4 Type the number of rows in the array.
- 5 Type the number of columns.
- 6 Type the number of levels.
- 7 Specify the vertical distance between the rows.
- 8 Specify the horizontal distance between the columns.
- 9 Specify the depth between the levels.

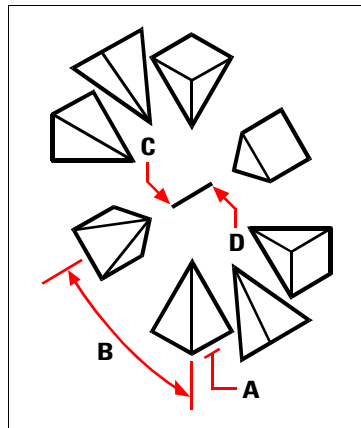


To create a three-dimensional rectangular array, select the entity to copy (A), type the number of rows, columns, and levels, and then specify the distance between each row (B), column (C), and level (D).

To create a three-dimensional polar array

- 1 Do one of the following to choose 3D Array ():
 - On the ribbon, choose Edit > 3D Array (in Modify).
 - On the menu, choose Modify > 3D Array.
 - On the Modify toolbar, click the 3D Array tool.
 - Type *3Darray* and then press Enter.
- 2 Select the entities, and then press Enter.
- 3 In the prompt box, choose Polar.
- 4 Type the number of copies to make, including the original selection set.
- 5 Specify the angle the array is to fill, from 0 to 360 degrees.

The default setting for the angle is 360 degrees. Positive values create the array in a counterclockwise direction; negative values create the array in a clockwise direction.
- 6 In the prompt box, choose one of the following:
 - Yes-Rotate Entities to rotate entities as they are arrayed.
 - No-Do Not Rotate to retain the original orientation of each copy as it is arrayed.
- 7 Specify the center point of the array.
- 8 Specify a second point along the central axis of the array.




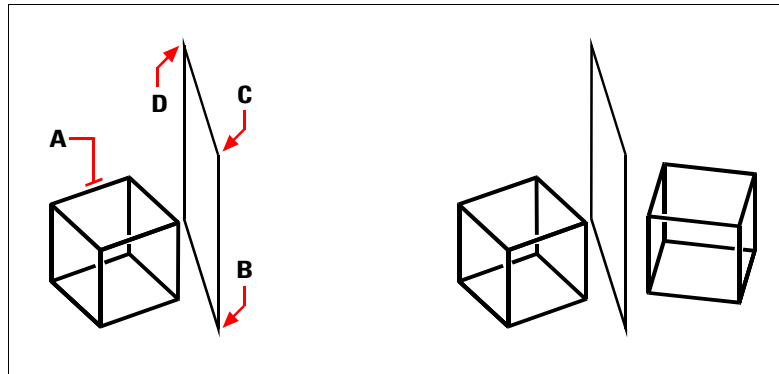
To create a three-dimensional polar array, select the entity to copy (A), type the number of copies to make, specify the angle the array is to fill (B), and then specify the center point of the array (C) and a second point along the central axis of the array (D).

Mirroring in three dimensions

You can create a mirror image of selected entities in three-dimensional space. You mirror the entities about a mirror plane that you define by either specifying three points; selecting an existing two-dimensional planar entity; aligning the plane parallel with the xy, yz, or xz plane of the current UCS; or aligning the plane with the current view. You can delete or retain the original entities.

To mirror an entity about a three-dimensional plane

- 1 Do one of the following to choose 3D Mirror ():
 - On the ribbon, choose Edit > 3D Mirror (in Modify).
 - On the menu, choose Modify > 3D Mirror.
 - On the Modify toolbar, click the 3D Mirror tool.
 - Type *mirror3D* and then press Enter.
- 2 Select the entities, and then press Enter.
- 3 In the prompt box, choose 3 Points, or press Enter to select the default.
- 4 Specify the first point on the mirror plane.
- 5 Specify the second and third points on the plane.
- 6 In the prompt box, choose one of the following:
 - Yes-Delete Entities to delete the original entities.
 - No-Keep Entities to retain the original entities.



Select the entity to mirror (A), and then specify the first point (B), second point (C), and third point (D) defining the mirror plane.

The resulting mirrored entity.


Aligning in three dimensions

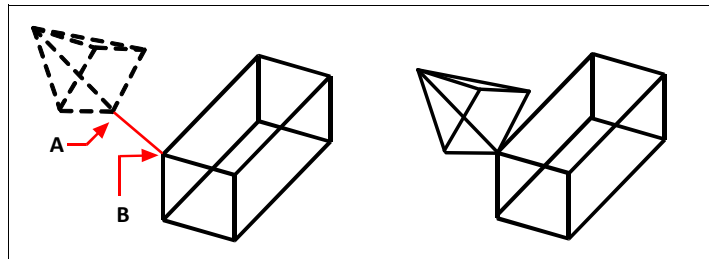
You can align one or more selected entities with other entities in three-dimensional space. First you select the entities you want to move and align, then you specify pairs of points to move and align the selected entities:

- One pair of points — Moves selected entities.

- Two pairs of points — Moves then rotates selected entities. The second pair of points can also determine the scale of moved entities (scaling is available only when using two pairs of points).
- Three pairs of points — Moves, rotates, then again rotates selected entities.

To align entities by moving them


- 1 Do one of the following to choose Align ():
 - On the ribbon, choose Edit > Align (in Modify).
 - On the menu, choose Modify > Align.
 - On the Modify toolbar, click the Align tool.
 - Type *align* and then press Enter.
- 2 Select the entities that will be moved, and then press Enter. Do not include destination entities in the selection set.
- 3 Specify the first source point.
- 4 Specify the first destination point, then press Enter.

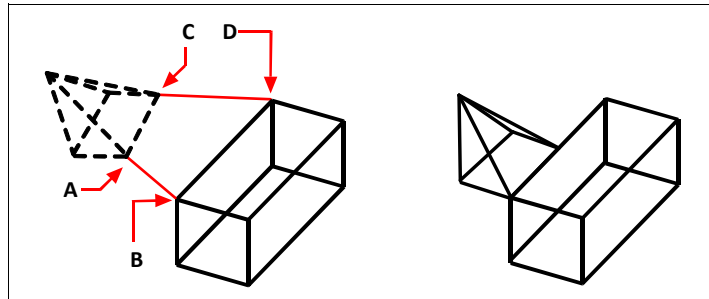


Select one or more entities to move, and then specify the first source point (A) and the first destination point (B).

The resulting entities.

To align entities by moving and rotating them


- 1 Do one of the following to choose Align (- On the ribbon, choose Edit > Align (in Modify).
- On the menu, choose Modify > Align.
- On the Modify toolbar, click the Align tool.
- Type *align* and then press Enter.
- 2 Select the entities that will be moved and rotated, and then press Enter. Do not include destination entities in the selection set.
- 3 Specify the first source point then the first destination point.
- 4 Specify the second source point then the second destination point, then press Enter.
- 5 If desired, choose Yes to scale the moved entities. Otherwise, choose No. If scaling, the selected entities scale proportionately so that the distance between the two source points is the same as the distance between the two destination points. The scaling option is available only when selecting two pairs of points.

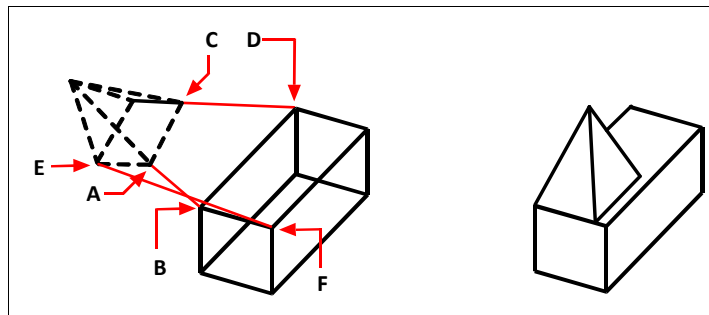


Select one or more entities to move and rotate, and then specify the first source point (A), the first destination point (B), the second source point (C), and the second destination point (D).

The resulting entities, not scaled.

To align entities by moving and then rotating them twice

- 1 Do one of the following to choose Align (- On the ribbon, choose Edit > Align (in Modify).
- On the menu, choose Modify > Align.
- On the Modify toolbar, click the Align tool.
- Type *align* and then press Enter.
- 2 Select the entities that will be moved and rotated, and then press Enter. Do not include destination entities in the selection set.
- 3 Specify the first source point then the first destination point.
- 4 Specify the second source point then the second destination point.
- 5 Specify the third source point then the third destination point, then press Enter.



Select one or more entities to move and rotate, and then specify the first source point (A), the first destination point (B), the second source point (C), the second destination point (D), the third source point (E), and the third destination point (F).


The resulting entities.

Projecting and trimming entities on a solid, surface, or region

You can modify a solid, surface, or region by projecting geometry onto it. For example, you can project holes drawn on a different plane onto a three-dimensional mechanical part and automatically trim the holes from the part.

You can project points, lines, arcs, circles, ellipses, 2D and 3D polylines, splines, and helices onto a three-dimensional solid, surface, or region using a direction that you specify.


To project entities onto a solid, surface, or region

1 (Optional) To trim the resulting projection from the three-dimensional entity, choose Auto Trim ():

- On the ribbon, choose Draw 3D > Auto Trim (in Curves).
- On the menu, choose Modify > 3D Operations > Auto Trim.

Use a system variable.

Set the SURFACEAUTOTRIM system variable to 1 to automatically trim projections from three-dimensional entities.

2 Do one of the following to choose Project Geometry ():

- On the ribbon, choose Draw 3D > Project Geometry (in Curves).
- On the menu, choose Modify > 3D Operations > Project Geometry.
- Type *projectgeometry* and then press Enter.

3 Select the entities that you want to project, and press Enter when done.

4 Select the surface, solid, or region onto which you want the selected entities to be projected, then press Enter.

5 Specify the direction by choosing one of the following:

- View — Creates a projection based on the current view.
- UCS — Creates a projection along the z-axis of the current UCS.
- Points — Creates a projection along a path between a start and end point that you specify.

Editing three-dimensional solids


You can edit three-dimensional solids in several unique ways, including: chamfer, fillet, section, and slice. You can also modify individual faces and edges of solids, as well as imprint, separate, shell, and check solids.

You can edit three-dimensional ACIS solids, including: boxes, cones, cylinders, dishes, domes, pyramids, spheres, tori, and wedges.


Chamfering and filleting solids

You can chamfer or fillet a three-dimensional solid much like you chamfer or fillet a two-dimensional entity.

To chamfer a solid

- 1 Do one of the following to choose Chamfer ():
 - On the ribbon, choose Edit > Chamfer (in Modify).
 - On the menu, choose Modify > Chamfer.
 - On the Modify toolbar, click the Chamfer tool.
 - Type *chamfer* and then press Enter.
- 2 Select the edge of the base surface to chamfer. (One of two surfaces adjacent to the selected edge will be highlighted.)
- 3 Do one of the following:
 - To select a different surface, type *n* and press Enter.
 - To use the current surface, press Enter.
- 4 Specify the base surface distance (measured from the selected edge to the base surface).
- 5 Specify the adjacent surface distance (measured from the selected edge to the adjacent surface).
- 6 Do one of the following:
 - Specify the edges to chamfer.
 - To select all edges around the base surface, type *l* and press Enter.

To fillet a solid

- 1 Do one of the following to choose Fillet ():
 - On the ribbon, choose Edit > Fillet (in Modify).
 - On the menu, choose Modify > Fillet.
 - On the Modify toolbar, click the Fillet tool.
 - Type *fillet* and then press Enter.
- 2 Select the edge of the solid to fillet.
- 3 Specify the fillet radius.
- 4 Select additional edges to fillet, and press Enter to fillet.


Sectioning and slicing solids

You can section or slice a three-dimensional solid, region, or body (typically a sheet).


When you section a solid, you obtain an “inside view” by creating a cross-section through the solid as a region or block. When you section a region or body, the resulting intersections are curves.

When you slice a solid, region, or body, you create a new entity by cutting the original entity and removing a specific side.

To section an entity

- 1 Do one of the following to choose Section ():
 - On the ribbon, choose Draw 3D > Section (in Draw 3D Solids).
 - On the menu, choose Draw > 3D Solids > Section.
 - On the Draw 3D Solids toolbar, click the Section tool.
 - Type *section* and then press Enter.
- 2 Select the entities to cross-section.
- 3 Do one of the following:
 - Specify three points to define the cross-section plane. (The first point defines the origin, while the second point defines the x-axis and the third point defines the y-axis.)
 - Type *o* and press Enter to select an entity that defines the cross-sectional plane.
 - Specify an axis by typing the appropriate letter and pressing Enter.


To slice an entity

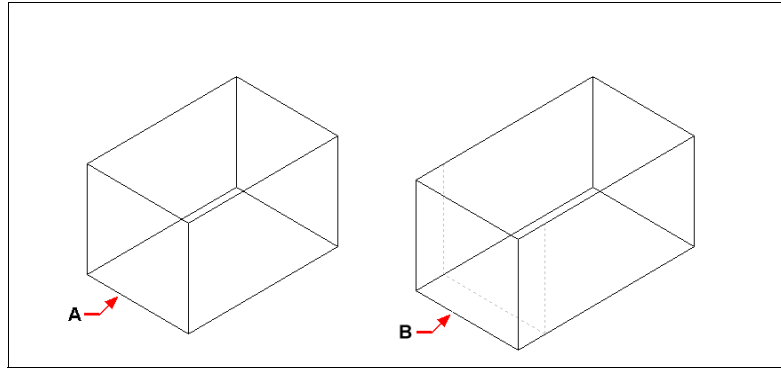
- 1 Do one of the following to choose Slice ():
 - On the ribbon, choose Draw 3D > Slice (in Draw 3D Solids).
 - On the menu, choose Draw > 3D Solids > Slice.
 - On the Draw 3D Solids toolbar, click the Slice tool.
 - Type *slice* and then press Enter.
- 2 Select the entities to slice.
- 3 Do one of the following:
 - Specify three points to define the cross-section plane. (The first point defines the origin, while the second point defines the x-axis and the third point defines the y-axis.)
 - Type *o* and press Enter to select an entity that defines the cross-sectional plane.
 - Specify an axis by typing the appropriate letter and pressing Enter.
- 4 Specify which side to retain, or type *b* to retain both sides.

Modifying faces

You can edit three-dimension solids by extruding, moving, rotating, offsetting, tapering, deleting, or copying individual faces. You can also change the color of individual faces.

*Extruding solid faces***To extrude a solid face**

- 1 Do one of the following to choose Extrude Face ():
 - On the ribbon, choose Draw 3D > Extrude Face (in Solid Editing).
 - On the menu, choose Modify > Solid Editing > Extrude Face.
 - On the Solids Editing toolbar, click the Extrude Face tool.
- 2 Select the entity with the face you want to extrude.
- 3 Select the face(s) to extrude, and press Enter.
- 4 Do one of the following:
 - Specify the height of extrusion.
 - Type *p* and press Enter to select a path for extrusion.
- 5 If you specified a height, specify a taper angle.




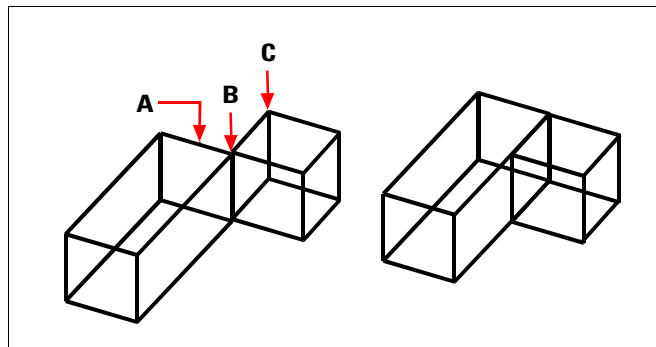
Select the entity to extrude, and then specify the face(s) to extrude (A), and the height of extrusion or path.

The resulting entity with the face extruded to position (B).

Moving solid faces

To move a solid face


- 1 Do one of the following to choose Move Face ():
 - On the ribbon, choose Draw 3D > Move Face (in Solid Editing).
 - On the menu, choose Modify > Solid Editing > Move Face.
 - On the Solids Editing toolbar, click the Move Face tool.
- 2 Select the entity with the face you want to move.
- 3 Select the face(s) to move, and press Enter.
- 4 Specify a base point.
- 5 Specify an end point.

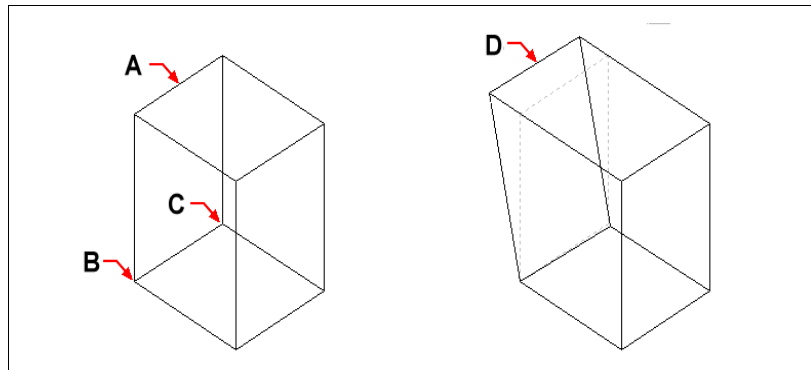


Select the entity, and then specify the face(s) to move (A), the base point (B), and the end point (C).

The resulting entity with the face moved to the new position.

*Rotating solid faces***To rotate a solid face**


- 1 Do one of the following to choose Rotate Face ():
 - On the ribbon, choose Draw 3D > Rotate Face (in Solid Editing).
 - On the menu, choose Modify > Solid Editing > Rotate Face.
 - On the Solids Editing toolbar, click the Rotate Face tool.
- 2 Select the entity with the face you want to rotate.
- 3 Select the face(s) to rotate, and press Enter.
- 4 Specify a base point.
- 5 Specify another point on the rotation axis.
- 6 Specify the rotation angle.

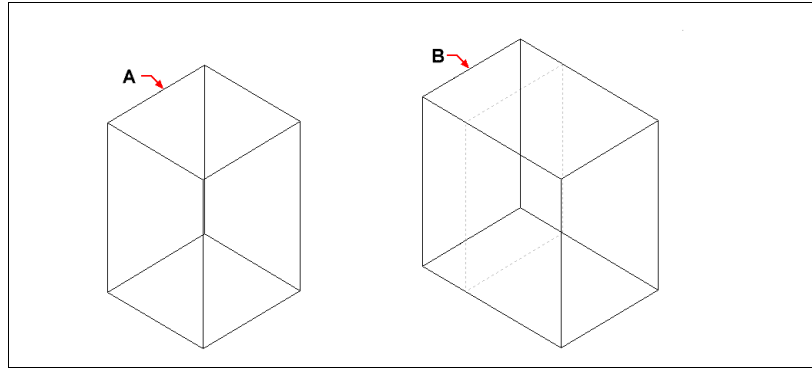


Select the entity, and then specify the face(s) to rotate (A), the base point (B), a second point on the rotation axis (C), and a rotation angle.

The resulting entity with the face rotated to position (D).

*Offsetting solid faces***To offset a solid face**

- 1 Do one of the following to choose Offset Face ():
 - On the ribbon, choose Draw 3D > Offset Face (in Solid Editing).
 - On the menu, choose Modify > Solid Editing > Offset Face.
 - On the Solids Editing toolbar, click the Offset Face tool.
- 2 Select the entity with the face you want to offset.
- 3 Select the face(s) to offset, and press Enter.
- 4 Specify an offset distance.




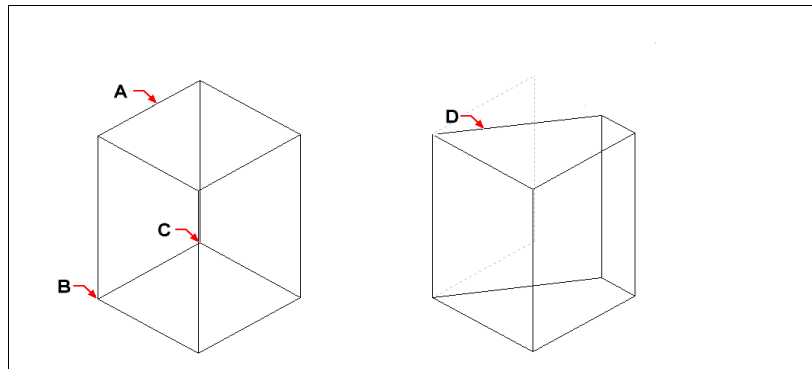
Select the entity, and then specify the face(s) to offset (A) and the distance to offset.

The resulting entity with the face offset to position (B).

Tapering solid faces

To taper a solid face


- 1 Do one of the following to choose Taper Face ():
 - On the ribbon, choose Draw 3D > Taper Face (in Solid Editing).
 - On the menu, choose Modify > Solid Editing > Taper Faces.
 - On the Solids Editing toolbar, click the Taper Face tool.
- 2 Select the entity with the face you want to taper.
- 3 Select the face(s) to taper, and press Enter.
- 4 Specify a base point.
- 5 Specify another point along the axis.
- 6 Specify a taper angle.

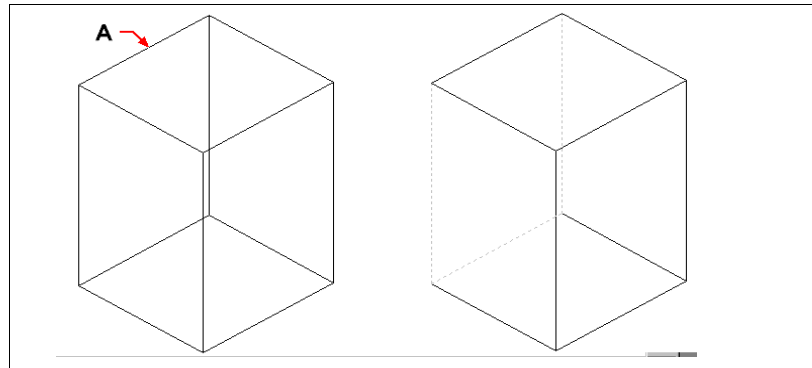


Select the entity, and then specify the face(s) to taper (A), a base point, a second point along the taper axis, and a taper angle.

The resulting entity with the face tapered to position (D).

*Deleting solid faces***To delete a solid face**


- 1 Do one of the following to choose Delete Face ():
 - On the ribbon, choose Draw 3D > Delete Face (in Solid Editing).
 - On the menu, choose Modify > Solid Editing > Delete Face.
 - On the Solids Editing toolbar, click the Delete Face tool.
- 2 Select the entity with the face you want to delete.
- 3 Select the face(s) to delete, and press Enter.

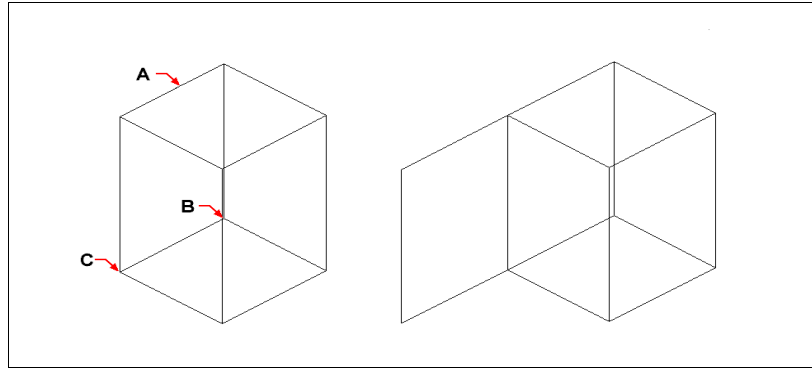


Select the entity, and then specify the face(s) to delete (A).

The resulting entity with the face deleted.

*Copying solid faces***To copy a solid face**

- 1 Do one of the following to choose Copy Face ():
 - On the ribbon, choose Draw 3D > Copy Face (in Solid Editing).
 - On the menu, choose Modify > Solid Editing > Copy Face.
 - On the Solids Editing toolbar, click the Copy Face tool.
- 2 Select the entity with the face you want to copy.
- 3 Select the face(s) to copy, and press Enter.
- 4 Specify a base point.
- 5 Specify an end point.




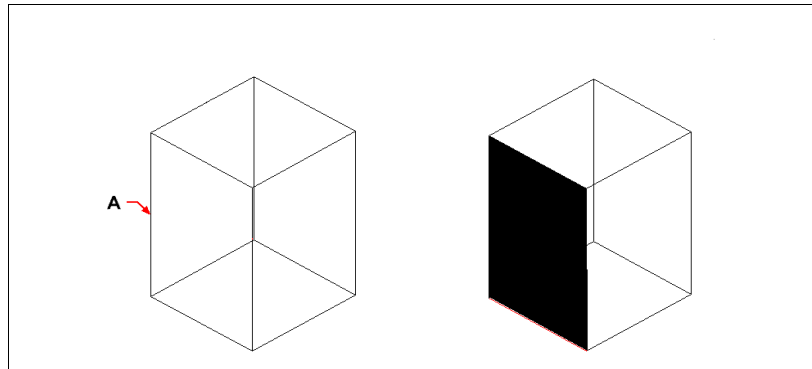
Select the entity, and then specify the face(s) to copy (A), the base point (B), and the end point (C).

The resulting entity with the face copied.

Coloring solid faces

To color a face

- 1 Do one of the following to choose Color Face ():
 - On the ribbon, choose Draw 3D > Color Face (in Solid Editing).
 - On the menu, choose Modify > Solid Editing > Color Face.
 - On the Solids Editing toolbar, click the Color Face tool.
- 2 Select the entity with the face you want to color.
- 3 Select the face(s) to color, and press Enter.
- 4 Specify a color.




Select the entity, and then specify the face(s) to color (A).

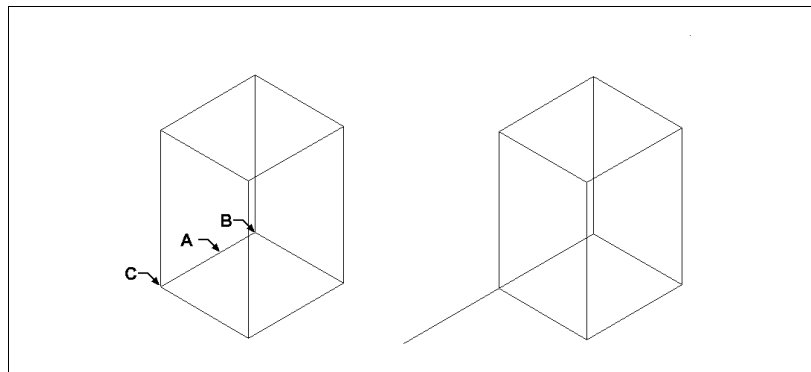
The resulting entity with the face colored.

Modifying edges

In addition to modifying faces of solids, you can modify individual edges. You can copy individual edges or change the color of individual edges.

To copy an edge


- 1 Do one of the following to choose Copy Edge ():
 - On the ribbon, choose Draw 3D > Copy Edge (in Solid Editing).
 - On the menu, choose Modify > Solid Editing > Copy Edge.
 - On the Solids Editing toolbar, click the Copy Edge tool.
- 2 Select the entity with the edge you want to copy.
- 3 Select the edge(s) to copy, and press Enter.
- 4 Specify a base point.
- 5 Specify an end point.



Select the entity, and then specify the edge(s) to copy (A), the base point (B), and the end point (C).

The resulting entity with the edge copied.


To color an edge

- 1 Do one of the following to choose Color Edge ():
 - On the ribbon, choose Draw 3D > Color Edge (in Solid Editing).
 - On the menu, choose Modify > Solid Editing > Color Edge.
 - On the Solids Editing toolbar, click the Color Edge tool.
- 2 Select the entity with the edge you want to color.
- 3 Select the edge(s) to color, and press Enter.
- 4 Specify a color.

Imprinting solids

You can modify the face of a solid by imprinting another entity on it. For example, you can imprint a line, arc, or polyline onto the face of a box.


To imprint a solid entity

- 1 Do one of the following to choose Imprint ():
 - On the ribbon, choose Draw 3D > Imprint (in Solid Editing).
 - On the menu, choose Modify > Solid Editing > Imprint.
 - On the Solids Editing toolbar, click the Imprint tool.
- 2 Select the solid entity you want to imprint.
- 3 Select the entity you want to imprint on the solid.

Separating solids

You can separate solids that have been combined. After you separate them, they are separated into individual solids.


To separate solids

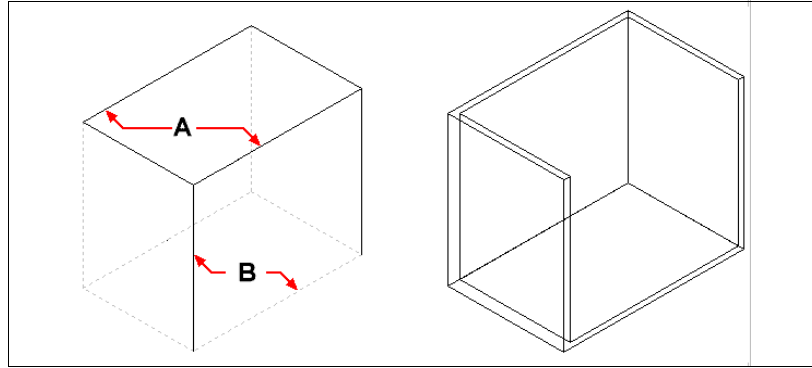
- 1 Do one of the following to choose Separate ():
 - On the ribbon, choose Draw 3D > Separate (in Solid Editing).
 - On the menu, choose Modify > Solid Editing > Separate.
 - On the Solids Editing toolbar, click the Separate tool.
- 2 Select the solid you want to separate.

Shelling solids

You can create a shell or a hollow thin wall from your 3D solid entity. PTCAD offsets existing faces to create new faces.

To shell a solid

- 1 Do one of the following to choose Shell ():
 - On the ribbon, choose Draw 3D > Shell (in Solid Editing).
 - On the menu, choose Modify > Solid Editing > Shell.
 - On the Solids Editing toolbar, click the Shell tool.
- 2 Select the entity you want to shell.
- 3 Remove any faces you don't want to include.
- 4 Specify an offset distance.




Select the entity, and then select faces to remove (A) and (B), then specify an offset distance.

The resulting shelled entity.

Cleaning solids

You can remove redundant edges or vertices from solids when they are not needed.


To clean a solid

- 1 Do one of the following to choose Clean ():
 - On the ribbon, choose Draw 3D > Clean (in Solid Editing).
 - On the menu, choose Modify > Solid Editing > Clean.
 - On the Solids Editing toolbar, click the Clean tool.
- 2 Select the entity you want to clean.

Checking solids

You can check whether a selected entity is a valid three dimensional ACIS solid. If it is a valid 3D solid, you can modify the entity using the 3D solid editing commands; if not, you cannot edit the entity using these commands.

To check a solid

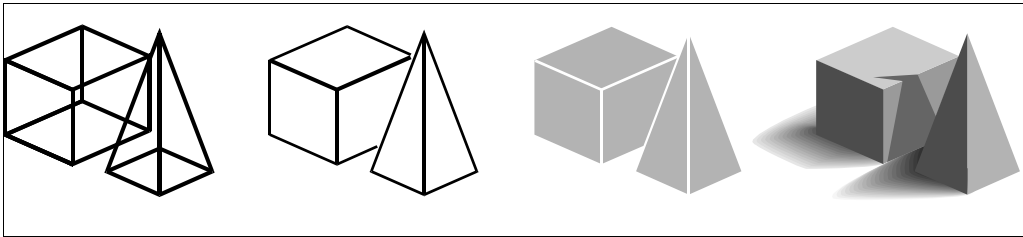
- 1 Do one of the following to choose Check ():
 - On the ribbon, choose Draw 3D > Check (in Solid Editing).
 - On the menu, choose Modify > Solid Editing > Check.
 - On the Solids Editing toolbar, click the Check tool.
- 2 Select the entities to check.

Hiding, shading, and rendering

As you create three-dimensional entities, the program displays both wireframe and surface models in wireframe view, which makes it difficult to visualize your three-dimensional models. To better visualize the model, you can remove all the lines that are hidden behind other entities or surfaces when seen from the current viewpoint.

Shading goes a step further by removing hidden lines and then assigning flat colors to the visible surfaces, making them appear solid. Shaded images are useful when you want to quickly visualize your model as a solid entity, though they lack depth and definition.

Rendering provides an even more realistic image of your model, complete with light sources, shadows, surface material properties, and reflections, giving your model a photo-realistic look. As shown in the following illustrations, when you render a model, the program removes hidden lines and then shades the surfaces as though they were illuminated from imaginary light sources.



Wire-frame model.

Hidden-line image.

Shaded image.

Rendered image.

Understanding dynamic and static hiding, shading, and rendering

PTCAD includes two ways to hide, shade, and render drawings, each with its own advantages:

- **Dynamic visual styles** — Drawings display a hidden-line, shadowed, or rendered view in real time while you draw and edit. You can add materials, lights, and a background to the drawing, and their effects also display dynamically. Visual styles are based on the OpenGL and OpenGL ES graphics devices.
- **Static images** — Static hidden-line, shadowed, and rendered images are created for the current view. While not dynamic, you can create static images using more options than visual styles, such as sun shadows, reflections, refactoring, and the ability to edit materials. Static images are based on DaVinci rendering.

Using dynamic visual styles for hiding, shading, and rendering

Visual styles can change the display of a drawing in real-time as you draw and edit a drawing. As you work, you can also add materials to entities, lighting to the scene, and a background to the current view, all of which display with visual styles that support rendering, such as Realistic and Shaded.

Visual styles can be managed using the Explorer. For more details, see “Working with visual styles” on page 338.

Displaying a drawing with a visual style

You can choose from a variety of styles according to your needs. For example, if you’re just starting a new drawing, you might want to view it using a wireframe mode, which looks similar to a model made out of wire and displays faster than more complex visual styles. Then as your drawing advances, you can switch to a more detailed visual style.

The following visual styles are available:

- 2D Wireframe — Drawing displays in two dimensions with all lines drawn at the edge of entities.
- Conceptual — Drawing displays in three dimensions with surfaces shaded in conceptual colors.
- Hidden — Drawing displays in three dimensions with all lines drawn at the edge of entities except those behind surfaces.
- Realistic — Drawing displays in three dimensions with surfaces filled with assigned materials.
- Shaded — Drawing displays in three dimensions with surfaces shaded.
- Shaded with Edges — Drawing displays in three dimensions with surfaces shaded and lines drawn at the edge of surfaces.
- Shades of Gray — Drawing displays in three dimensions with surfaces shaded in monochromatic shades of gray.
- Sketchy — Drawing displays in two and three dimensions with a freehand sketch effect.
- Wireframe — Drawing displays in three dimensions with all lines drawn at the edge of entities.
- X-ray — Drawing displays in three dimensions with surfaces partially transparent.

The more detail that displays on the screen, the greater the impact on performance.

To change the visual style

- 1 Do one of the following:
 - On the ribbon, choose View and in Visual Styles, make your selection.
 - On the menu, choose View > Visual Styles, then make your selection.
 - On the View toolbar, click your selection.
 - Type *vscurrent*, press Enter, then make your selection.

Adding materials that display with visual styles

Materials that display with visual styles are managed using the Explore Materials command. After you import materials to the drawing, you can assign them to entities and layers, and their effects are seen using the Realistic visual style.

Use the Realistic visual style to display materials assigned to entities.

For more details about using the Materials Explorer, see “Working with materials” on page 362.

Adding lighting effects that display with visual styles


PTCAD applies default lighting to all models when the drawing is displayed using Realistic, Shaded, Shaded with Edges, and Shades of Gray visual styles.

If you want to create a specific lighting scene, you can add your own lights that specify the placement of lights, light color, and light intensity to determine how your drawing or scene is lit. Lights can be placed either outside the field of view or inside to illuminate different areas of the scene.


There are several ways to add custom lighting to your drawing:

- **Point Light command** A point light shines light from its location in all directions. Point lights are entities in a drawing and their effects appear for shaded or rendered visual styles in PTCAD and in third-party software such as Artisan Renderer.
- **Spot Light command** A spot light shines light in the shape of a cone from its location towards the direction you specify. Spot lights are entities in drawings and their effects appear for shaded or rendered visual styles in PTCAD and in third-party software such as Artisan Renderer.

To create a point light

- 1 Do one of the following to choose Point Light ():
 - On the ribbon, choose View > Point Light (in Lights).
 - On the menu, choose View > Lights > Point Light.
 - Type *pointlight* and then press Enter.
- 2 If default lighting is currently turned on, PTCAD prompts you to turn it off so you can see the effect of your new lights. Click Yes to turn off default lighting.
- 3 Enter the x-, y-, and z-coordinates of where to place the point light, or click the location in the drawing.

To create a spot light

- 1 Do one of the following to choose Spot Light ():
 - On the ribbon, choose View > Spot Light (in Lights).
 - On the menu, choose View > Lights > Spot Light.
 - Type *spotlight* and then press Enter.
- 2 If default lighting is currently turned on, PTCAD prompts you to turn it off so you can see the effect of your new lights. Click Yes to turn off default lighting.
- 3 Enter the x-, y-, and z-coordinates of where to place the spot light, or click the location in the drawing.
- 4 Then enter the x-, y-, and z-coordinates of the location you want to shine light, or click the location in the drawing.

Edit point lights and spot lights.

You can move and copy point lights and spot lights in your drawing, just as you would any other entity. To specify settings for the point or spot light, such as color and attenuation, right-click the light, choose Properties, and make your selections in the Properties pane. And when you select a spot light, grips display allowing you to change the light and target location, cone size, and fall-off size.

Lights of older file formats can be updated to the current format.


If you open a drawing that contains lights that were created in an older format, type CONVERTOLDLIGHTS at the command prompt to update them to the current format.

Adding a background that displays with visual styles

Backgrounds display for the current view when using any visual style except 2D Wireframe. You can choose a solid-color, gradient, or image background.

Backgrounds can also be assigned to named views and displayed with the same visual styles. For more details, see “Setting the background for a named view” on page 692 in this chapter.

To set the background for the current view

- 1 Do one of the following to choose Background ():
 - On the ribbon, choose View > Background (in Views).
 - On the menu, choose View > Background.
 - Type *background* and then press Enter.

- 2 Choose one of the following:
 - Solid
 - Gradient
 - Image
- 3 In the Background dialog box, make your selections. Click [?] to see details about each option.
- 4 Click OK.

Creating static hidden-line, shaded, and rendered images

Creating hidden-line images

Creating a hidden-line image of your drawing removes all the lines that are hidden behind other surfaces when seen from your vantage point. When you remove hidden lines or shade a model, the program treats the entities differently, depending on how you created them. Wireframe models always appear transparent, because they have no surfaces. Surface models appear filled, with surfaces applied to all visible sides.

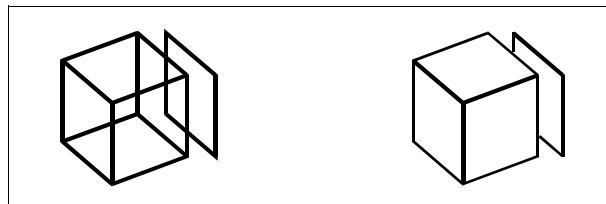
You can also control the hidden-line view on an entity-by-entity basis by showing and hiding edges of selected 3D faces, meshes, and polyface meshes.

To create a hidden-line image

- Type *hide* and then press Enter.

Use a visual style.


Choosing View > Visual Styles > Hidden is similar to using the Hide command. For more details, see “Displaying a drawing with a visual style” on page 768.



Before creating a hidden-line image.

After removing hidden lines.


To hide edges of 3D faces, meshes, and polyface meshes

- 1 Do one of the following to choose Hide Edges ():
 - On the ribbon, choose Draw 3D > Hide Edges (in Draw 3D Meshes).
 - On the menu, choose Draw > 3D Meshes > Hide Edges.
 - Type *hideedges* and then press Enter.
- 2 Select the entities with edges that you want to hide, then press Enter when done.

Individual edges can be hidden for 3D faces.

The Hide Edges command hides all edges of selected entities, but if you are working with 3D faces, you can also use the Edge command to hide individual edges.

To show edges of 3D faces, meshes, and polyface meshes

- 1 Do one of the following to choose Show Edges ():
 - On the ribbon, choose Draw 3D > Show Edges (in Draw 3D Meshes).
 - On the menu, choose Draw > 3D Meshes > Show Edges.
 - Type *showedges* and then press Enter.
- 2 Select the entities with edges that you want to show, then press Enter when done.

Individual edges can be displayed for 3D faces.

The Show Edges command shows all edges of selected entities, but if you are working with 3D faces, you can also use the Edge command to show individual edges.

Creating shaded images

Creating a shaded image of your drawing removes hidden lines and then applies shading to the visible surfaces based on their entity color. Because they are intended to provide a quick visualization, shaded images do not have a light source and use continuous colors across surfaces, causing them to appear flat and unrealistic.

To create a shaded image

- Type *shade* and then press Enter.

To control the appearance of the shaded image, choose Tools > Drawing Settings, and then click the 3D Settings tab and select the options you want. You can shade the surfaces and edges of the model in four ways:

- Faces shaded; edges not highlighted.
- Faces shaded; edges highlighted in the background color.
- Faces filled in the background color; edges drawn using the entity color (similar to a hidden-line view).
- Faces filled using the entity color; edges highlighted in the background color.

Use a visual style.

Choosing a shaded visual style (View > Visual Styles) is similar to using the Shade command. For more details, see “Displaying a drawing with a visual style” on page 768.

Creating photo-realistic rendered images


Creating a rendered image of your drawing removes hidden lines and then shades the surface as though it were illuminated from multiple light sources.

Full rendering creates a photo-realistic image of your model, complete with light sources, shadows, surface material properties, and reflections. You can illuminate your image with spotlights, distant lighting to simulate sunshine, and ambient light. If you choose not to customize the light sources, the program generates default light sources for you.

Rays from these imaginary light sources are traced as they reflect off and refract through the surfaces of the model, a process called *ray tracing*. Ray tracing determines where shadows fall and how reflections on shiny materials such as metal and glass appear. You can modify the reflective properties of the materials that make up your model to control how the light rays reflect off its surfaces.


Full rendering automatically creates a base on which your model is displayed, if you don't already have one, so it does not appear suspended in space. A background is also automatically added to the image. A background such as a cloudy sky or an imported raster graphic such as a stone wall can also be added behind the image, making it even more realistic.

To create a quickly rendered image

Do one of the following to choose Render ():

- On the ribbon, choose View > Render Settings (in Rendering).
- On the menu, choose View > Rendering > Render.
- On the Rendering toolbar, click the Render tool.
- Type *render* and then press Enter.

To create a fully rendered image

Do one of the following to choose Full Render ():

- On the ribbon, choose View > Render Settings (in Rendering).
- On the menu, choose View > Rendering > Full Render.
- On the Rendering toolbar, click the Full Render tool.
- Type *fullrender* and then press Enter.



Fully rendered image.

Use a visual style.

Choosing the Realistic visual style (View > Visual Styles) is similar to using the render commands. For more details, see “Displaying a drawing with a visual style” on page 768.




Creating custom rendered images

Before you use the Full Render command to create a rendered image of a drawing, you can apply materials, a background, and lighting (including shadows) to customize your drawing:

- **Materials** Specify materials for different surfaces and define how the materials map to those surfaces. Predefined materials are available in the materials library, which can be customized further using the built-in editor. You can choose to use procedural or bitmap materials.
- **Backgrounds** Specify the background or backdrop for a rendered image. Several predefined backgrounds are available. By default, no background is used and it appears black. The background is an infinite, planar surface and reflects off of any reflective surfaces in your model. The background is not affected by lighting however, so no shadows or highlights are seen on the background.

- **Lighting** Specify the placement of lights, light color, and light intensity to determine how your drawing or scene is lit, including shadows and reflections in the scene. Lights can be placed either outside the field of view or inside to illuminate different areas of the scene. Several predefined lighting controls are available, including ambient lighting, diffuse lighting, specular reflections, specular highlights, and transparency.

To apply materials, backgrounds, and lighting

- 1 Do one of the following to choose Materials () , Backgrounds () , or Lighting ():
 - On the ribbon, choose View > Materials, Backgrounds, or Lighting (in Rendering).
 - On the menu, choose View > Rendering, and then choose Materials, Backgrounds, or Lighting.
 - On the Rendering toolbar, click the Materials, Backgrounds, or Lighting tool.
 - Type *materials*, *backgrounds*, or *lighting* and then press Enter.
- 2 Make your selections.


You can specify additional rendering options.

Choose View > Rendering > Render Settings. For more information about creating custom rendered images, click Help in any of the Render dialog boxes.

Saving a rendered image

You can save a rendered image to a different format — either a bitmap (.bmp), JPEG (.jpg), TIFF (.tif), TrueVision TGA (.tga), or Portable Network Graphic (.png). If you want to print the rendered image, you can print the saved image from another graphics program.

To save a rendered image of your drawing


- 1 Create a rendered image of the drawing.
- 2 Do one of the following to choose Render Settings ():
 - On the ribbon, choose View > Render Settings (in Rendering).
 - On the menu, choose View > Rendering > Render Settings.
 - On the Rendering toolbar, click the Render Settings tool.
 - Type *setrender* and then press Enter.
- 3 On the Rendering tab, click Save Last Image.
- 4 Enter a file name and path.
- 5 In Save As Type, choose the file format.
- 6 Click Save.

Rendering in Artisan Renderer

Similar to the Full Render command in PTCAD, Artisan Renderer allows you to create a photo-realistic image of your model. However, many users prefer using Artisan Renderer to speed up the design process with access to a wide range of pre-set materials and lighting setups, along with the ability to create custom realistic materials.


For more details about Artisan Renderer, choose Help > Help in Artisan Renderer.

To create a rendered image in Artisan Renderer

- 1 Save the drawing.
- 2 Do one of the following to choose Artisan Render (- On the ribbon, choose View > Artisan (in Rendering).
- On the menu, choose View > Rendering > Artisan.
- On the Artisan Rendering toolbar, click the Artisan tool.
- Type *artisan* and then press Enter.
- 3 If the Artisan Settings dialog box displays, choose from the following options:
 - **Units** Select the units.
 - **Language** Select the language. To use the operating system language, choose Use System Locale.
 - **Facet resolution** Set the resolution of shaded three dimensional entities and faces when exporting to Artisan Renderer.
 - **Smoothing angle** Set the angle of smoothing applied to crease edges when exporting to Artisan Renderer.
 - **Show on Startup** Choose to display the Artisan Settings dialog each time you use the Artisan or Artisan Sync command in PTCAD.
- 4 Click OK.

Artisan Renderer opens and displays your model for you to add materials and other effects. For more details about using Artisan Renderer, choose Help > Help in Artisan Renderer.

To synchronize your latest model with Artisan Renderer

- 1 Save the drawing.
- 2 Do one of the following to choose Artisan Sync (- On the ribbon, choose View > Artisan Sync (in Rendering).
- On the menu, choose View > Rendering > Artisan Sync.
- On the Artisan Rendering toolbar, click the Artisan Sync tool.
- Type *artisan-sync* and then press Enter.

Artisan Renderer is updated with the latest model from PTCAD, which is helpful if you're working in PTCAD and Artisan Renderer at the same time and you change the model in PTCAD.

Working with other programs and computers

PTCAD offers great flexibility in its capability to be used with other programs. You can include an PTCAD drawing in a Microsoft® Word document or insert a Microsoft® Excel spreadsheet containing a parts list into an PTCAD drawing. To include PTCAD drawings in other programs and documents from other programs in PTCAD drawings, you either link or embed them. You can also save Intelli-CAD drawings in other file formats that can be used directly with other programs or send PTCAD drawings to coworkers via e-mail.

This section explains how to:

- Save and view snapshots.
- Use object linking and embedding.
- Export PTCAD drawings to other file formats.
- Convert one or more drawings to other file formats.
- Use PTCAD data on other computers.
- Use PTCAD with the Internet.

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Saving and viewing snapshots

You can save snapshots of a drawing to view later. A snapshot saves the current drawing in either .emf, .wmf, or .sld files exactly as it appears on the screen. A snapshot is not a drawing file. You cannot edit or print the snapshot; you can only view it.

You can use snapshots in the following ways:

- Add slide (.sld) files to image tile menus.
- Make presentations by showing snapshots of your drawings.
- Reference a snapshot of a drawing while working on a different drawing.
- Present a number of snapshots as a slide show by using scripts.


For details about using slide (.sld) files with image tile menus, see “Customizing image tile menus” on page 912.

Creating snapshots

You create a snapshot by saving the current view as a snapshot. A snapshot does not include any entities on layers that are not currently visible. The contents of the snapshot also depend on the current drawing space. In model space, the snapshot shows only the current viewport. In paper space, the snapshot contains all visible viewports.

If you are creating a slide (.sld) file to use in an image tile menu, make sure the image fills the screen before you create the snapshot. Image tiles display with a 3:2 ratio in image tile menus.

To create a snapshot

- 1 Display the drawing exactly as you want to capture it as a snapshot.
- 2 Do one of the following to choose Make Snapshot (- On the ribbon, choose Tools > Make Snapshot (in Manage).
- On the menu, choose Tools > Make Snapshot.
- On the Tools toolbar, click the Make Snapshot tool.
- Type *msnapshot* and then press Enter.
- 3 In the dialog box that displays, specify the name of the snapshot file you want to create.
- 4 From the Files Of Types list, choose either .emf, .wmf, or .sld.
- 5 Click Save.


The current drawing remains on the screen, and the snapshot is saved to the directory that you specify. Slide (.sld) files that are used in image tile menus should be located in the program’s installation folder.

Viewing snapshots

You can view previously saved snapshots, and you can also view snapshots created using AutoCAD.

When you view a snapshot, it temporarily replaces the current drawing with the external snapshot file. When you refresh the display of the current drawing (by redrawing, panning, zooming, minimizing, maximizing, or tiling), the snapshot image disappears, and you are returned to the current drawing.

To view a snapshot

- 1 Do one of the following:
 - On the ribbon, choose Tools > View Snapshot (in Manage).
 - Choose Tools > View Snapshot.
 - On the Tools toolbar, click the View Snapshot tool ().
 - Type *vsnapshot* and then press Enter.
- 2 In the dialog box that displays, specify the name of the snapshot file you want to view.
- 3 Click Open.

PTCAD displays the snapshot in the current drawing window.

Using data from other programs in PTCAD drawings

You can include data from other programs in PTCAD drawings by using either embedding or linking. The method you choose depends on the type of object or file you want to include in your PTCAD drawing and what you want to do with it after it is there.

Embedding objects into drawings


Embed an object into your PTCAD drawing when you want to keep all the data you work with in one file or if you want to transfer the file to other computers. You can embed data from programs that support object linking and embedding.

For example, if you want to distribute data about a department's computer equipment along with an PTCAD drawing of the department's floor plan, you can embed a Microsoft® Excel spreadsheet into the floor plan.

When you embed data from another program, PTCAD becomes the container for that data. The object embedded in the PTCAD drawing becomes part of the PTCAD file. When you edit the data, you open its program from within the PTCAD drawing.


Any changes you make to the embedded data exist only in the PTCAD drawing, so it is not necessary to keep that data in a separate file. If the data does exist in a separate file, the original file does not change when you modify the embedded object in PTCAD. Also, changes to the original file do not affect the embedded object in the PTCAD drawing.

To embed another program's object into an PTCAD drawing

- 1 Open the file that contains the data you want.
- 2 In the file, select the data you want to embed in the PTCAD drawing.
- 3 Choose that program's command to place data on the Clipboard.
Usually, you choose Edit > Copy.
- 4 In the PTCAD window, display the drawing in which you want to embed the object.
- 5 Do one of the following to choose Paste :
 - On the ribbon, choose Home > Paste (in Clipboard) or choose Edit > Paste (in Modify)
 - On the menu, choose Edit > Paste.
 - On the Standard toolbar, click the Paste tool.
 - Type *paste* and then press Enter.


The data on the Clipboard is pasted into the drawing as an embedded object. The object appears in the center of the view, but you can select and move it by moving the cursor.

To embed an object from an existing file within PTCAD

- 1 Do one of the following to choose OLE Object ():
 - On the ribbon, choose Insert > OLE Object (in Data).
 - On the menu, choose Insert > OLE Object.
 - Type *insertobj* and then press Enter.
- 2 In the Insert Object dialog box, click Create From File.
- 3 Specify the file by doing one of the following:
 - Type a path and file name in the File box.
 - Click Browse to select a file.
- 4 Select Display As Icon if you want that program's icon to appear in the drawing instead of the data.
- 5 Click OK.

The first page of the file appears in the PTCAD drawing, unless you chose to display it as an icon. You can select the object and drag to reposition it.

To create a new embedded object from within PTCAD

- 1 Do one of the following to choose OLE Object ():
 - On the ribbon, choose Insert > OLE Object (in Data).
 - On the menu, choose Insert > OLE Object.
 - Type *insertobj* and then press Enter.
- 2 In the Insert Object dialog box, click Create New.
- 3 From the Object Type list, select the type of object you want to create, and then click OK.

The program for creating that object opens within PTCAD. If the program is compatible with ActiveX, it opens in place (within the other program) in the PTCAD drawing; otherwise, the program opens in its full window.

- 4 Create the object in the other program.
- 5 If the program is running within the other document (in place), click anywhere outside the embedded object to close the program.

If the program is running in its full window, choose File > Exit.

Linking objects to drawings

If another program supports ActiveX, you can link its data to PTCAD drawings. Use linking when you want to include the same data in many files. When you update the data, all links to other files reflect the changes.

For example, if you created your company logo in an ActiveX-compatible drawing program, and you want to include it in the title block of every drawing you create with PTCAD, you can link it to each PTCAD drawing. When you change the original logo in the drawing program, the PTCAD drawing updates automatically.

When you link data from another program, the PTCAD drawing stores only a reference to the location of the file in which you created the data. You link data from a saved file, so PTCAD can find the data and display it.

Because linking adds only a reference to a file, the data does not significantly increase the file size of the PTCAD drawing. However, links require some maintenance. If you move any of the linked files, you need to update the links. In addition, if you want to transport linked data, you must include all linked files.

You can update a linked object automatically every time you open the drawing, or you can do so only when you specify. Anytime a link is updated, changes made to the object in its original file also appear in the PTCAD drawing, and the changes also appear in the original file if they were made through PTCAD.


To link a file to an PTCAD drawing

- 1** Save the original file.

Because a link consists of a reference to the original file, you must save the file before you can link to it.

- 2** In the original file, select the data you want in the PTCAD drawing.
- 3** Choose that program's command to place data on the Clipboard.
Usually, you choose Edit > Copy.
- 4** Display the PTCAD drawing to which you want to link the file.
- 5** In PTCAD, choose Edit > Paste Special.
- 6** In the Paste Special dialog box, select Paste Link.
- 7** Click OK.

To create a linked object from within PTCAD

- 1 Display the PTCAD drawing in which you want to display the linked object.
- 2 Do one of the following to choose OLE Object (- On the ribbon, choose Insert > OLE Object (in Data).
- On the menu, choose Insert > OLE Object
- Type *insertobj* and then press Enter.
- 3 In the Insert Object dialog box, click Create From File.
- 4 Specify the file by doing one of the following:
 - Type a path and file name in the File box.
 - Click Browse to select the file using a file dialog box.
- 5 Select the Link check box.
- 6 Select Display As Icon if you want that program's icon to appear in the drawing instead of the data.
- 7 Click OK.

The first page appears in the PTCAD drawing, unless you chose to display it as an icon. To reposition the object, select and drag it.

Editing an embedded or linked object from within PTCAD

You can modify an embedded or linked object in its original program from within PTCAD. When you modify an embedded object, you change the object only in PTCAD, not its original file (if you pasted the object from an existing file). When you modify a linked file, however, you open and change the original file.

Most programs include a submenu of actions you can perform on an embedded or linked ActiveX object. Usually, the commands for editing ActiveX objects are Edit and Open. If the object is embedded and its program supports in-place editing, the Edit command opens the object in place. The Open command opens the object in the full program window. In PTCAD, this command appears at the bottom of the Edit menu.

To edit an embedded or linked object

- In the PTCAD drawing, double-click the object.

If the object is embedded and the program in which you created the object supports in-place editing, the object opens in place.

If the object is linked, or if its program does not support in-place editing, the other program opens in its full window and displays the object.

Importing files created in other formats


You can import files that have the following formats:

- ACIS format — Three-dimensional Spatial Technologies ACIS solids saved as an .sat file.
- DWF format — Autodesk® Design Web Format (used with .dwf files) is used to distribute a drawing for others to view in a Web browser, review, and edit. The DWF format uses the .dwf file extension.
- DWT format — Drawing templates contain predefined settings that you can reuse when you create new drawings. Drawing templates use the .dwt file extension.
- DXF format — Autodesk Drawing Exchange Format is an ASCII description of a drawing file with a .dxf file extension.
- DXB format — Autodesk Drawing Exchange Format is a binary description of a drawing file with a .dxb file extension.
- DGN format — Drawing files used with Bentley Microstation. The DGN format uses the .dgn file extension.
- DAE format — Collada files are an interactive three-dimensional graphics file format used by 3D graphics applications (three-dimensional entities are exported, including ACIS entities). Collada files use the .dae file extension.
- FMB format — Facet Model Binary data used by PTCAD to store facet model mesh data.
- IFC format — Drawing files in the IFC (Industry Foundation Classes) format used for building and construction data.
- IGES format — Model data in the IGES (Initial Graphics Exchange Specification) format as an .iges or .igs file.
- Map files — Map files are .shp, .sdf and .sqlite shapefiles that store geometric location and attribute information of geographic features.
- OBJ format — Model data in OBJ (Wavefront Technologies object) format as an .obj file.
- PDF format — Model data used with Adobe® Acrobat® Reader® and Adobe® Acrobat.
- STEP format — Model data in the STEP (Standard for the Exchange of Product) format as an .step, .ste or .stp file.
- STL format — Model data in an STL (Stereolithography) format as an .stl file, which is common when working with 3D printers.
- TXT or CSV format — Point data located in a text or comma-separated value format file can be imported as point entities, vertices of a polyline, or vertices of a spline.
- XLS format — Data in a table from an .xls or .xlsx Microsoft Excel file.

Importing a DXF, DWF, DWT, DGN, or DAE format file

Importing .dxf files, two-dimensional .dwf files, .dwt, .dae, and .dgn files is similar to opening a standard drawing file.


To import a DXF, DWF, DWT, DGN, or DAE format file

- 1 Do one of the following to choose Import ():
 - On the ribbon, choose the Application button then choose Import, or choose Insert > Import.
 - On the menu, choose File > Import > Import.
 - On the Standard toolbar, click the Open tool.
 - Type *import* and then press Enter.
- 2 Choose the folder that contains the drawing.
- 3 In Files of Type, choose the type of drawing you want to import.
- 4 Choose the file you want to open.
- 5 Click Open.

Importing a DXB format file

A DXB format file is a drawing file used for building and construction data.

To import a DXB format file


- 1 Do one of the following to choose DXB In ():
 - On the ribbon, choose the Application button > Import > DXB In, or choose Insert > DXB In (in Import).
 - On the menu, choose File > Import > DXB In or Insert > Drawing Exchange Binary.
 - Type *dxbin* and then press Enter.
- 2 Choose the file you want to open.
- 3 Click Open.

Importing a PDF format file or underlay


A PDF format file can contain two- and three-dimensional model data. The format is used to distribute models that can be viewed with Adobe® Acrobat® Reader® and Adobe® Acrobat.

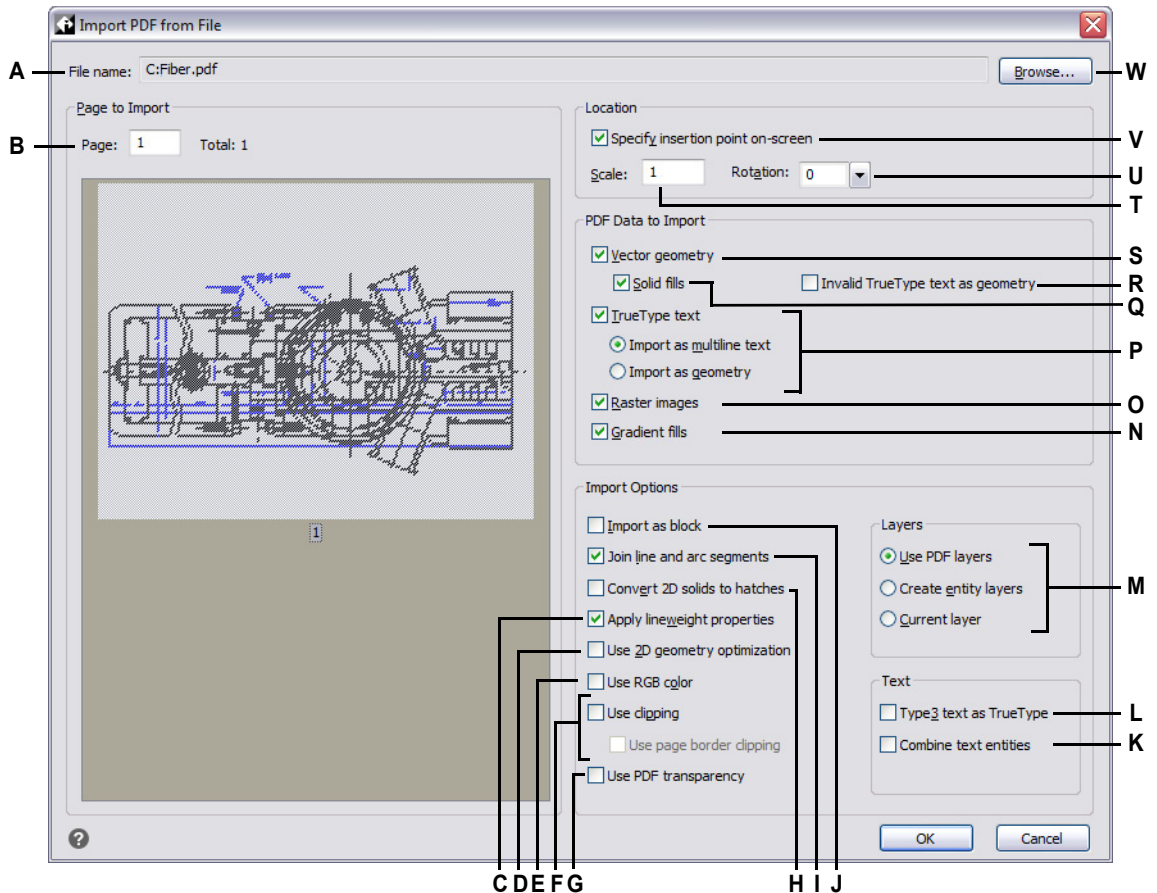
You can import content from either a .pdf file or a .pdf underlay that already exists in your drawing.

To import content from a .pdf file

- 1 Do one of the following to choose PDF In (- On the ribbon, choose the Application button > Import > PDF In, or choose Insert > PDF In (in Import).
- On the menu, choose File > Import > PDF In.
- Type *pdfin* and then press Enter.
- 2 Choose File, then press Enter.
- 3 Choose the file you want to open, then click Open.
- 4 In the Import PDF from File dialog box, make your selections.
- 5 Click OK.
- 6 Specify the insertion point by clicking in the drawing area or entering coordinates.

To import content from a PDF underlay

- 1 Do one of the following to choose PDF In (- On the ribbon, choose the Application button > Import > PDF In, or choose Insert > PDF In (in Import).
- On the menu, choose File > Import > PDF In.
- Type *pdfin* and then press Enter.
- 2 In the drawing, select the frame of the PDF underlay you want to import.
The PDFFRAME system variable must be set to 1 or 2 to select its frame.
- 3 Specify the portion of the underlay to import:
 - Rectangular area — Specify two diagonal corners of the rectangular area to import.
 - Polygonal area — Choose Polygonal, then pick at least three points to define the polygonal area to import.
 - Entire area — Choose All to import all data contained in the .pdf underlay.
- 4 Choose Settings to specify more options, then in the Import PDF from Underlay dialog box, make your selections, and click OK when done.
The dialog box that displays is similar to the Import PDF from File dialog box.
- 5 Choose what to do with the underlay:
 - Keep — Keep the .pdf underlay unchanged.
 - Detach — Remove the .pdf underlay from the drawing.
 - Unload — Hide the .pdf underlay temporarily.




- A Displays the path and filename of the .pdf file to import.
- B Enter the number of the page to import from the specified file.
- C Mark to retain linewidth properties of imported entities.
- D Mark to interpret entities that resemble arcs, circles, and ellipses as such within a tolerance.
- E Mark to use RGB colors during the import. If unmarked, colors from the palette are used.
- F Mark to enable clipping. If marked, you can also enable clipping at the border of the page.
- G Mark to keep transparency of geometric data unchanged.
- H Mark to import two-dimensional solid entities as solid-filled hatches.
- I Mark to combine contiguous line and arc segments into a polyline.
- J Mark to import the .pdf content as a block.
- K Mark to combine text entities that use the same font and are located on the same line into a single text entity.
- L Mark to import text entities that use Type3 fonts as entities that use TrueType fonts.
- M Select how to create layers.
- N Mark to import gradient fills and save them as .png files.
- O Mark to save raster images as .png files and attach them.
- P Mark to import text entities that use TrueType fonts. Then choose whether the resulting text entities are multiline text or geometry (polylines and solid-filled hatches).
- Q Mark to import all solid-filled areas. Available only when Vector Geometry is selected.
- R Mark to import text entities that use invalid TrueType fonts as polylines and solid-filled hatches.
- S Mark to import vector geometry (such as linear paths, Bezier curves, and solid-filled areas) as polylines and two-dimensional solids. If unmarked, it is imported as solid-filled hatches.
- T Enter the scale factor for the imported .pdf content.
- U Enter the rotation angle for the imported .pdf content.
- V Select to specify the insertion point for the .pdf content directly in the drawing.
- W Click to select a .pdf file to import.

Importing an IFC format file

An IFC format file typically contains a 3D model of a building that can be used for visualization, design, analysis, and to generate building construction documents.

Importing an .ifc file creates separate entities for each item in the file and converts those items to AEC entities where possible, depending on your selections.

To import an IFC format file

- 1 Do one of the following to choose IFC In:
 - On the ribbon, choose the Application button > Import > IFC In () , or choose Insert > IFC In (in Import).
 - On the menu, choose File > Import > IFC In.
 - Type *ifcimport* and then press Enter.
- 2 Choose the file you want to import.
- 3 Click Open.
- 4 In the IFC Import Options dialog box that displays, mark the elements you want to import from the .ifc file. For details about each option, click [?].
- 5 When finished selection import options, click OK.
- 6 Select an insertion point in the drawing.
- 7 Enter the scale or specify the drawing units type in which to insert the .ifc file, then press Enter.
- 8 Enter the rotation to use for the insertion, then press Enter.

Importing an ACIS format file

An ACIS format file contains three-dimensional solids, regions, or bodies that are saved as an ASCII .sat file.


To import an ACIS format file

- 1 Do one of the following:
 - On the ribbon, choose the Application button > Import > ACIS In, or choose Insert > ACIS In (in Import).
 - On the menu, choose File > Import > ACIS In.
 - Type *acisin*, and then press Enter.
- 2 Choose the directory containing the .sat file.
- 3 Choose the .sat file you want to open.
- 4 Click Open.

Importing an IGES format file

The IGES (Initial Graphics Exchange Specification) format is an exchange format that typically contains 3D surfaces saved in an .iges or .igs file. IGES data is imported as meshes, or it is imported as three-dimensional solids if your program version supports the Spatial® ACIS® modeler.

To import an IGES format file


- 1 Do one of the following to choose IGES In (- On the ribbon, choose the Application button > Import > IGES In, or choose Insert > IGES In (in Import).
- On the menu, choose File > Import > IGES In.
- Type *igesimport* and then press Enter.
- 2 Choose the .iges or .igs file you want to import.
- 3 Click Open.

Processing begins. When the selected file is ready to be imported a balloon message displays in the status bar.
- 4 Click the file name in the balloon message to complete the import.
- 5 Zoom to extents to see the imported entities.

Importing a STEP format file

STEP (Standard for the Exchange of Product) is an ISO standard exchange format that typically contains 3D objects saved in an .step, .ste, or .stp file. STEP data is imported as meshes, or it is imported as three-dimensional solids if your program version supports the Spatial® ACIS® modeler.

To import an STEP format file

- 1 Do one of the following to choose STEP In (- On the ribbon, choose the Application button > Import > STEP In, or choose Insert > STEP In (in Import).
- On the menu, choose File > Import > STEP In.
- Type *stepimport* and then press Enter.
- 2 Choose the .step, .ste, or .stp file you want to import.
- 3 Click Open.


Processing begins. When the selected file is ready to be imported a balloon message displays in the status bar.
- 4 Click the file name in the balloon message to complete the import.
- 5 Zoom to extents to see the imported entities.

Importing an OBJ format file

OBJ (Wavefront Technologies object) is a format that is typically used in three-dimensional modelers and for three-dimensional printing. It contains geometry such as points, lines, texture vertices, faces, curves, and surfaces.

OBJ model data is imported as mesh entities.


To import an OBJ format file

- 1 Do one of the following to choose OBJ In ():
 - On the ribbon, choose the Application button > Import > OBJ In, or choose Insert > OBJ In (in Import).
 - On the menu, choose File > Import > OBJ In.
 - Type *objin* and then press Enter.
- 2 Choose the .obj file you want to import.
- 3 Click Open.
Processing begins.
- 4 Zoom to extents to see the imported entities.

Importing an FMB format file

FMB (Facet Model Binary) is a format that stores facet model mesh data. FMB data is imported as facet model mesh entities.


To import an FMB format file

- 1 Do one of the following to choose FMB In ():
 - On the ribbon, choose the Application button > Import > FMB In, or choose Insert > FMB In (in Import).
 - On the menu, choose File > Import > FMB In.
 - Type *fmbin* and then press Enter.
- 2 Choose the .fmb file you want to import.
- 3 Click Open.
Processing begins.
- 4 Zoom to extents to see the imported entities.

Importing an STL format file

The STL (Stereolithography) format typically contains 3D solids and meshes for working with 3D printers. STL format files are saved in .stl files.

To import an STL format file

- 1 Do one of the following to choose STL In (- On the ribbon, choose the Application button > Import > STL In, or choose Insert > STL In (in Import).
- On the menu, choose File > Import > STL In.
- Type *stlin* and then press Enter.
- 2 Choose the .stl file you want to import.
- 3 Click Open.
Processing begins.
- 4 Zoom to extents to see the imported entities.

Importing a TXT or CSV format file containing points

Point data in a text or comma-separated file can be imported as point entities, vertices of a polyline, or vertices of a spline.

If you import the point data as point entities, the appearance of the imported points is determined by the PDMODE system variable.


The following is an example of point data in a .csv file:

```
1.1,2.2,3.3
5.5,6.6,7.7
```

The first line of a file can include a text description (e.g., column titles), but all other lines should contain at least three numbers representing x-, y- and z-coordinates. Commas, semicolons, tabs, and spaces can be used to separate values.

If you import point data as polylines or splines, blank lines in the file indicate the end of one entity and the beginning of the next.

To import a TXT or CSV file containing points


- 1 Do one of the following to choose Import Points (- On the ribbon, choose Insert > Import Points (in Data); Express Tools > Import Points (in Tools).
- On the menu, choose Insert > Import Points or Express Tools > Tools > Import Points.
- Type *impoint* and then press Enter.

- 2 Choose the .txt or .csv file that contains the points you want to import.
- 3 Click Open.
- 4 Choose one of the following:
 - **Points** Choose Points to import points as point entities. Then press Enter to use the default point size or enter a new value. The point size is stored in the PDSIZE system variable.
 - **Polyline** Choose Polyline to import points as polyline vertices. Then choose Yes to import points as vertices of a closed polyline or No to import them as vertices of an open polyline.
 - **Spline** Choose Spline to import points as spline vertices. Then choose Yes to import points as vertices of a closed spline or No to import them as vertices of an open spline.

Importing an XLS format file

Data can be imported from an .xls or .xlsx file created using Microsoft® Excel® or another spreadsheet program. The imported data fills a new table that is created automatically during import, and the new table is formatted automatically to match the format of the original spreadsheet.


To import an XLS format file

- 1 Do one of the following to choose XLS In ():
 - On the ribbon, choose Insert > XLS In (in Data).
 - On the menu, choose Insert > XLS In.
 - Type *xlsimport* and then press Enter.
- 2 Choose the .xls or .xlsx file that contains the data you want to import.
- 3 Click Open.
- 4 If the specified file contains more than one sheet, enter the name of the sheet to import. If necessary, type ? and press Enter to see a list of sheet names.
- 5 Specify the insertion point for the upper-left corner of the table.


Importing a map file

You can import map files that are .shp, .sdf, and .sqlite shapefiles and also map data from PostgreSQL, MySQL, and WFS servers. Shapefiles are used for storing geometric location and attribute information of geographic features.

To import a shapefile file

- 1 Do one of the following to choose Map Import (- On the ribbon, choose Insert > Map Import (in Data).
- On the menu, choose Insert > Map Import.
- Type *mapimport* and then press Enter.
- 2 Choose SDF, SQLITE, or SHP, depending on the shapefile you want to import.
- 3 In the dialog box that displays, select the file, then click Open.
- 4 Specify the name of the input layer with data that you want to import, then press Enter.
- 5 Specify the name of the table to import data from, then press Enter.
- 6 Specify the coordinate system for the imported map data. If a coordinate system was already assigned to the current drawing by typing *adesetcrdsys* in the command bar, the imported map data is converted to the coordinate system of the drawing.
- 7 Enter coordinate system: Specify the coordinate system for the imported map data. If a coordinate system was assigned to the current drawing, the imported map data is converted to the coordinate system of the drawing.
- 8 Choose Yes to import non-geometry data from the source file, or choose No to only import geometry.
- 9 Specify the target layer in the drawing for imported data, then press Enter.
- 10 Choose Yes to import three-dimensional geometry data in three dimensions, or choose No to convert it to two dimensions.
- 11 Choose Yes to import polygons as closed polylines, or choose No to import as polygons.

To import data from a PostgreSQL, MySQL, and WFS server

- 1 Do one of the following to choose Map Import (- On the ribbon, choose Insert > Map Import (in Data).
- On the menu, choose Insert > Map Import.
- Type *mapimport* and then press Enter.
- 2 Choose POSTGRESQL, MYSQL, or WFS, depending on the type of server you want to connect to.
- 3 Follow the prompts, which vary according to the server.

Using PTCAD data in other programs

You can use any of the following methods to include PTCAD data in a document created in another program:

- Embedding
- Linking
- Dragging
- Exporting
- E-mailing

The method you choose depends on the capabilities of the other program and how you want to work with the PTCAD data after you've placed it in the other document.

Each method except exporting uses ActiveX to integrate data from different programs. *With ActiveX, you can open PTCAD drawings from within the other program to modify the PTCAD drawings.*

Embedding drawings

When you embed an PTCAD drawing, it becomes part of the other program's document file. When you edit the drawing, you edit only the version that is embedded in the other document.

Embedding is useful when you don't want to maintain a link to the PTCAD drawing for the data you include in the other document. Edits made to the new drawing do not affect the original drawing. To transfer the file to other computers, you can transfer all the data in one file, but embedded objects increase the file size.

From within a document in a program that supports ActiveX, such as Microsoft® Word, you can either create a new embedded PTCAD drawing or embed an existing PTCAD drawing.

To create an PTCAD drawing in another document

- 1 In the document, choose Insert > Object (or the equivalent command for that program).
- 2 In the dialog box, click the options for creating a new file.
- 3 Under Object Type, choose PTCAD Drawing, and then click OK.
- 4 Create the PTCAD drawing.
- 5 If PTCAD is running in its own window, choose File > Exit.
If PTCAD is running within the other document (in place), click somewhere in the document outside the PTCAD drawing to close PTCAD.

- 6 To edit the PTCAD drawing from within the document, double-click the drawing.


You can also embed an existing PTCAD drawing from within another document.

Follow step 1 in the preceding procedure, and then click the option for creating an object from an existing file.

To embed selected PTCAD entities

- 1 In PTCAD, select the entities you want to embed.
- 2 Choose Edit > Copy (or press Ctrl+C).
- 3 Open the document in which you want to embed the entities.
- 4 Choose Edit > Paste (or the equivalent command).

To embed an entire PTCAD drawing

- 1 Open the document in which you want to embed the drawing.
- 2 Do one of the following to choose OLE Object (- On the ribbon, choose Insert > OLE Object (in Data).
- On the menu, choose Insert > OLE Object
- Type *insertobj* and then press Enter.
- 3 Click Create From File.
- 4 Click Browse, and then choose the file you want to embed.
- 5 Click Insert, and then click OK.

Editing an embedded PTCAD object in place

In many ActiveX-compatible programs you can edit an embedded PTCAD object without leaving the program (or container application). This is called in-place editing. A different set of PTCAD menus and toolbars temporarily replaces most of the menus and controls in the active window while you edit the PTCAD object.

To edit an embedded PTCAD object in place

- 1 In the container application, double-click the embedded PTCAD object.
A different set of PTCAD menus and controls appears.
- 2 Edit the PTCAD drawing.
- 3 Click anywhere outside the drawing window to exit the in-place editing controls.

Linking drawings

When you link an PTCAD drawing to another document, the other document contains only a reference to the PTCAD drawing file, rather than the actual drawing. You link data in a saved PTCAD file so that the other program can find the data and display it.

Linking works well when you want to include the same PTCAD data in more than one document. When you update the data, you need update it in only one location. The versions that are linked to other documents reflect the changes automatically.

Linking an PTCAD file to another document does not increase the file size the way embedding an PTCAD object does. However, links require more maintenance. To transport the data, you must make sure to transfer all linked files to the other computer.

To link an PTCAD file to another document

- 1 Open the drawing you want to link.

Because a link is a reference to a file, the referenced file must be saved on a local or remote disk.

If you haven't saved the drawing you want to link, choose File > Save.

- 2 In the other program, open the document in which you want to include the PTCAD drawing.
- 3 Choose that program's command for inserting objects.

In Microsoft® Office programs, choose Insert > Object. In the Object dialog box, click the Create From File tab. Specify the name of the drawing file you want to link. Select the Link To File check box, and then click OK.

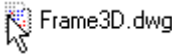
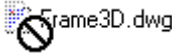
The drawing appears in the document, with a link to the original PTCAD file.

Dragging PTCAD drawings into other programs

If the other program in which you want to include PTCAD drawings is compatible with ActiveX, an alternative to pasting drawings with menu commands is to drag drawing file icons from Windows Explorer into the other document. Dragging and dropping drawings does not use the Clipboard, so data on the Clipboard is not affected.

When you drag an PTCAD drawing file from Windows Explorer, you link or embed the entire drawing in the other document. When you drag the file, the cursor changes in response to the action you take.

How the cursor changes

Cursor appearance	Action
 Frame3D.dwg	Drag to embed the selected file to the other document.
 Frame3D.dwg	Cannot drop drawings in that document.

For easy drag-and-dropping, position the application windows side by side.

Before you drag a drawing, position the Windows Explorer window and the other program's window so you can see the file icon and the document in which you want to drop it.

To drag and embed drawings into another document

- Select the icon for the drawing file, and then drag the drawing into the document.

Exporting drawings

You can save or export PTCAD drawings in a number of different formats for use with other programs. When you save a drawing in a different format, the program saves all the entities in the drawing to the new file. Or, you can choose which entities are included in the new file.

File formats that can be exported

The following table describes the file formats that can be exported.

Export formats

Format	File extension	Details
Bitmap	.bmp	Graphics file
Collaborative Design Activity (Collada)	.dae	Interactive three-dimensional graphics file format used by 3D graphics applications (three-dimensional entities are exported, including ACIS entities)
Comma-Separated Values	.csv	Text file that stores table data without formatting
Design Drawing	.dgn	Distribute your drawing to others for review, editing, and markup using other CAD programs such as Bentley® Microstation®
Design Web Format	.dwf	Distribute your drawing to others for using with Autodesk® software and tools
Enhanced Windows Metafile	.emf	Graphics file
Facet Model Binary	.fmb	Facet model data file
Industry Foundation Classes	.ifc	Distribute your drawing to others for review, editing, and markup in other BIM applications
Portable Document Format	.pdf	Distribute your drawing in 2D or 3D to others for using with Adobe® Acrobat® Reader® and Adobe® Acrobat
Scalable Vector Graphics	.svg	Graphics file and Web development language
Stereolithography	.stl	Graphics file used for prototyping and 3D printing three-dimensional models
Wavefront Technologies Object Format	.obj	File used for 3D modeling and 3D printing
Windows Metafile	.wmf	Graphics file

You can also export ACIS solids, regions, and surfaces to an ASCII file (*.sat) that you can use in other programs.

Exporting to a BMP, EMF, WMF, DAE, or SVG format file

Exporting to a file is similar to saving a standard file.


To export a drawing to a .bmp, .emf, .wmf, .dae, or .svg file

- 1 Do one of the following:
 - On the ribbon, choose the Application button then choose Export > Export, or choose Output > Export (in Export).
 - On the menu, choose File > Export > Export.
 - Type *export* and then press Enter.
- 2 In the Export Drawing As File dialog box, under Save As Type, choose the file format.
- 3 Specify the name of the file you want to create.
- 4 Click Save.
- 5 If the selection prompt box displays, choose the entity-selection method, and then create the selection set.
- 6 When you have finished selecting entities, press Enter.

Exporting to an STL format file

You can export three-dimensional solids to an STL (Stereolithography) file with an .stl extension. The STL format is commonly used when working with 3D printers.

To export to an STL format file


- 1 Do one of the following to choose STL Out (- On the ribbon, choose the Application button then choose Export > STL Out or Output > STL Out (in Export).
- On the menu, choose File > Export > STL Out.
- Type *stlout* and then press Enter.
- 2 Specify the name of the file you want to create.
- 3 Click Save.
- 4 Choose the entity-selection method, and then create a selection set that contains the entities you want to export.
- 5 Choose Yes to export to a binary .stl file. Choose No to export to an ASCII .stl file.
- 6 Zoom to extents to see the imported entities.

Exporting to a PDF format file

PDF format files allow you to distribute your drawing to others for viewing in Adobe® Acrobat® Reader®, which is free software that users can download. PDF files can also be viewed, reviewed, and edited in Adobe® Acrobat.

You can export drawings to two-dimensional and three-dimensional PDF files.

To export a drawing to a .pdf file

- 1 Do one of the following to choose PDF Out :
 - On the ribbon, choose the Application button then choose Export > PDF Out, or choose Output > PDF Out (in Export).
 - On the menu, choose File > Export > PDF Out.
 - Type *pdfexport* and then press Enter.
- 2 In Save As Type, choose Portable Document Format (pdf).
- 3 Specify the name of the file you want to create.
- 4 Click Save.
- 5 Mark the layouts and/or the model to export.
- 6 Select the page setup you want to assign to each layout or the model when exported.
- 7 Select the layer state you want to assign to each layout or the model when exported.
- 8 To create a 3D .pdf file instead of 2D, mark Include Layer Information and then mark Create 3D PDF File.
- 9 Choose additional options for how you want to export to the .pdf file. Click [?] to see details about each option.
- 10 Click Export.

You can also print and publish to .pdf files.


If instead of exporting you need to print to a .pdf file, you can print or publish directly to a .pdf file. One advantage of publishing is that you can print from multiple drawings to a single .pdf file. For details about printing to .pdf, see “Selecting a printer or plotter” on page 637. For details about publishing to .pdf, see “Publishing a sheet list to the printer named in page setup” on page 677.

Exporting to a DWF format file

DWF format files allow you to publish your drawings so they can be viewed on the Internet using a Web browser. PTCAD exports your drawing to a Design Web Format (.dwf) file, which can be viewed in a Web browser if Autodesk Design Review is also installed on the computer. Design Review is a free tool from Autodesk.

You can export your drawing to a 2D .dwf file or a 3D .dwf file. 2D .dwf files have smaller file sizes, but cannot be viewed in three dimensions. 3D .dwf files can be viewed in three dimensions using the Autodesk® Design Review, but have larger file sizes.


To export a drawing to a .dwf file

- 1 Do one of the following to choose DWF Out :
 - On the ribbon, choose the Application button then choose Export > DWF Out, or choose Output > DWF Out (in Export).
 - On the menu, choose File > Export > DWF Out.
 - Type *dwfout* and then press Enter.
- 2 In Save As Type, choose Design Web Format (dwf).
- 3 Specify the name of the file you want to create.
- 4 Click Save.
- 5 Choose the entity-selection method, and then create a selection set that contains the entities you want to export.
- 6 Choose how you want to export:
 - **DWF File Version** — Choose the file version you want. Version 4.2 can export entities on the Model tab only (no layouts). Version 5.5 can export the current layout only. Version 6.0 can export the current layout or all layouts.
 - **DWF File Format** — Choose the desired file format. Compressed binary files have a smaller file size than uncompressed binary files (both are 2D .dwf files). ASCII files have the largest file size for 2D .dwf files, but 3D .dwf files have larger file sized and can be viewed in three dimensions.
 - **Layout to Export** — Choose whether to export the current layout only, or all layouts in the drawing.
- 7 Click OK.
- 8 When you have finished selecting entities, press Enter.

Exporting to a DGN format file

DGN format files allow you to distribute your drawing to others for review, editing, and markup using other CAD programs such as Bentley® Microstation®. Files are exported to DGN version 8 files.

To export a drawing to a .dgn file


- 1 Do one of the following to choose DGN Out ( - On the ribbon, choose the Application button then choose Export > DGN Out, or choose Output > DGN Out (in Export).
 - On the menu, choose File > Export > DGN Out.
 - Type *dgnexport* and then press Enter.
 - 2 Specify the name of the file you want to create.
 - 3 Click Save.
 - 4 At the prompt, choose a master unit for the .dgn file such as kilometers or feet.
 - 5 At the prompt, choose a sub unit for the .dgn file such as inches or mils.
- The drawing is exported.

Exporting to an IFC format file

The IFC (Industry Foundation Classes) format is used for exchanging building and construction data. It typically contains a 3D model of a building that can be used for visualization, design, analysis, and to generate building construction documents.

You can export AEC entities, .ifc underlays, and .rvt underlays to an IFC format file.


To export a drawing to an IFC format file

- 1 Do one of the following to choose IFC Out (- On the ribbon, choose the Application button then choose Export > IFC Out, or choose Output > IFC Out (in Export).
- On the menu, choose File > Export > IFC Out.
- Type *ifcexport* and then press Enter.
- 2 Specify the name of the file you want to create.
- 3 Click Save.
- 4 In the IFC Export Options dialog box, make your selections. Click [?] for more details about each option.
- 5 Click OK.

Exporting to a CSV format file

Table data can be exported from a drawing to the Comma Separated Values (CSV) format. The resulting .csv file contains table contents only without formatting.

To export a table to a CSV format file


- 1 Do one of the following to choose CSV Out (- On the ribbon, choose the Application button then choose Export > CSV Out, or choose Output > CSV Out (in Export).
- On the menu, choose File > Export > CSV Out.
- Type *tableexport* and then press Enter.
- 2 Select the table you want to export.
- 3 Specify the name of the file you want to create.
- 4 Click Save.

Exporting to an OBJ format file

The Wavefront Technologies OBJ format is commonly used for three-dimensional models and three-dimensional printing. It stores information about geometry such as points, lines, texture vertices, faces, curves, and surfaces.

Color and texture information is exported to a companion .mtl (Material Template Library) file format which is referenced from within the .obj file.


To export a drawing to an OBJ format file

- 1 Do one of the following to choose OBJ Out (- On the ribbon, choose the Application button then choose Export > OBJ Out, or choose Output > OBJ Out (in Export).
- On the menu, choose File > Export > OBJ Out.
- Type *objout* and then press Enter.
- 2 Select the entities you want to export.
- 3 Specify the name of the file you want to create.
- 4 Click Save.

Exporting to an FMB format file

Facet model entities can be exported from a drawing to the Facet Modeler Binary (FMB) format. The .fmb file that is created stores facet model data that can be shared with other CAD programs.

To export facet model entities to an FMB format file

- 1 Do one of the following to choose OBJ Out :
 - On the ribbon, choose the Application button then choose Export > FMB Out, or choose Output > FMB Out (in Export).
 - On the menu, choose File > Export > FMB Out.
 - Type *fmbout* and then press Enter.
- 2 Select the facet model entities you want to export.
- 3 Specify the name of the file you want to create.
- 4 Click Save.

Exporting to an ACIS format file

You can export ACIS entities such as surfaces, regions, and solids to an ACIS format file in ASCII (SAT) format.

To export an ACIS format file

- 1 Do one of the following:
 - On the ribbon, choose the Application button then choose Export > ACIS Out, or choose Output > ACIS Out (in Export).
 - On the menu, choose File > Export > ACIS Out.
 - Type *acisout* and then press Enter.
- 2 Select the ACIS entities you want to save.
- 3 Specify the name of the file you want to create.
- 4 Click Save.


Converting drawings to other file versions and formats

Several types of drawings can be converted to other file versions and formats. You can convert a batch of drawings in a folder or specify a single drawing to convert.

The following formats can be converted from and converted to:

- Autodesk DWG™ format — Autodesk Drawing Format is a drawing with a .dwg file extension.
- Autodesk DXF™ format — Autodesk Drawing Exchange Format is an ASCII description of a drawing file with a .dxf file extension.
- DGN format — Drawing files used with Bentley® Microstation®. The DGN format uses the .dgn file extension.


To convert a single drawing

- 1 Do one of the following to choose PTCAD Converter ()
 - On the ribbon, choose the Application button then choose Drawing Utilities > PTCAD Converter, or choose Tools > PTCAD Converter (in Manage).
 - On the menu, choose File > PTCAD Converter.
 - Type *intelliconvert* and then press Enter.
- 2 Choose Single file, then click Next.
- 3 Select an input file. Click [...] to browse to the location.
- 4 Name an output file. Click [...] to browse to the location.
- 5 In Convert to version, select the file format and version for the output file.
- 6 If you are converting to a .dgn file, select any of the following:
 - **Master unit** — Assigns the selected master unit to the output .dgn file.
 - **Sub unit** — Assigns the selected sub unit to the output .dgn file.
 - **Bind external references when possible** — Makes external references a permanent part of the .dgn file, similar to a block, if external references are found in the input drawing.
- 7 If you have more files to convert, mark Convert more files.
- 8 Click Finish.

A log file is created automatically if converting to a .dgn file and errors occur.

The file <output_dgnfilename>.log is saved in the same folder where the output .dgn file is created.

To convert a batch of drawings

- 1 Do one of the following to choose PTCAD Converter ()
 - On the ribbon, choose the Application button then choose Drawing Utilities > PTCAD Converter, or choose Tools > PTCAD Converter (in Manage).
 - On the menu, choose File > PTCAD Converter.
 - Type *intelliconvert* and then press Enter.

- 2 Choose Multiple files, then click Next.
- 3 Specify the input files by doing the following:
 - Enter an input folder. Click [...] to browse to the location.
 - To include files located in subfolders within the input folder, mark Process subfolders.
 - Choose which files to include: .dwg, .dxf, and .dgn.
- 4 Click Next.
- 5 Review the list of files that are found, then click Next to proceed using all of the named files. You can also click Back to specify different files.
- 6 Enter an output folder. Click [...] to browse to the location.
- 7 Mark Convert, then select the output format and file version.
- 8 To audit and fix files during the conversion, mark Audit files and fix errors. You can also choose to save audit log files, which will also be saved in the output folder.
- 9 Select what to do, if while processing, files with the same name are found in the output folder:
 - **Replace existing** — Replaces existing files with the new files. To make a backup copy of the existing file before it is overwritten, mark Create backup files (*.bak), and backup files will be created in the same output folder.
 - **Skip existing** — Skips adding a new file if an existing file is found with the same name.
 - **Add suffix to output files** — Adds the specified suffix to all new files created in the output folder.
- 10 If you are converting to a .dgn file, select any of the following:
 - **Master unit** — Assigns the selected master unit to the output .dgn file.
 - **Sub unit** — Assigns the selected sub unit to the output .dgn file.
 - **Bind external references when possible** — Makes external references a permanent part of the .dgn file, similar to a block, if external references are found in the input drawing.
- 11 Click Next.

New files are generated based on your specifications.

A log file is created automatically if converting to a .dgn file and errors occur.

The file <output_dgnfilename>.log is saved in the output folder that was specified for batch file conversion.

Using PTCAD data on other computers

Within PTCAD there are two ways to move drawings and related files to another computer:

- Create a package that contains all related drawings and files. This option uses the eTransmit command and is best for moving multiple drawings or a drawing that has supporting files such as external reference files and font files.
- Send a drawing through e-mail from PTCAD. This option uses the Send Mail command and is best for moving single drawings.


Packaging drawings and files for use on another computer

Use the eTransmit command to package together drawings and their related files, such as external references, materials from Explore Materials, and fonts. The resulting package can be a self-extracting .exe file, a .zip file, or a separate folder that contains the files.

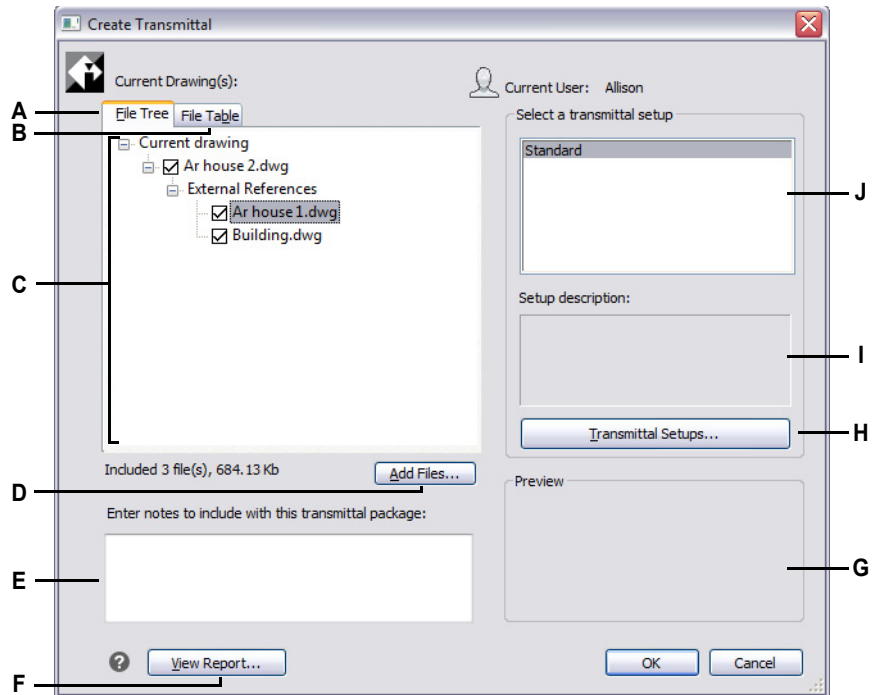
The eTransmit command uses *transmittal setups* to define how packages are created, with options for filenaming, password control, whether to include fonts, and more. The default transmittal setup is named Standard, which you can modify. Or you can create new transmittal setups.

Creating a package that contains drawings and related files

To package drawings and their related files

- 1 Do one of the following to choose eTransmit :
 - On the ribbon, choose the Application button then Publish > eTransmit, or choose Output > eTransmit (in Export).
 - On the menu, choose File > eTransmit.
 - Type *etransmit* and then press Enter.
- 2 Review the available files in the list. The files display in either hierarchical format (click the File Tree tab) or in a table (click the File Table tab).
- 3 Mark the files to include in the package; unmark the files to exclude from the package.
- 4 To include files that are not listed, click Add Files, navigate to and select the desired files, then click Open.
- 5 In Select a Transmittal Setup, choose the transmittal setup that defines how you want the package to be created. The transmittal setup specifies the package file-type (.exe, .zip, or folder), filenaming, and more.
- 6 To view or create a transmittal setup, click Transmittal Setups. For more details, see “Working with transmittal setups” on page 811 in this chapter.

- 7 Optionally enter notes to include in the transmittal report.
- 8 Click OK.
The package is created.




- | | |
|---|---|
| <p>A Click to view files in a hierarchy.</p> <p>B Click to view files in a table.</p> <p>C Mark the files to include in the package; unmark files to exclude.</p> <p>D Click to add files to the list.</p> <p>E Enter details to include in the package report.</p> <p>F Click to view the report that will be included in the package.</p> | <p>G Displays a preview of the file selected in the list.</p> <p>H Click to view, create, and modify transmittal setups.</p> <p>I Displays a description of the selected transmittal setup.</p> <p>J Select the transmittal setup that defines the options you want to use when creating the package.</p> |
|---|---|

Working with transmittal setups

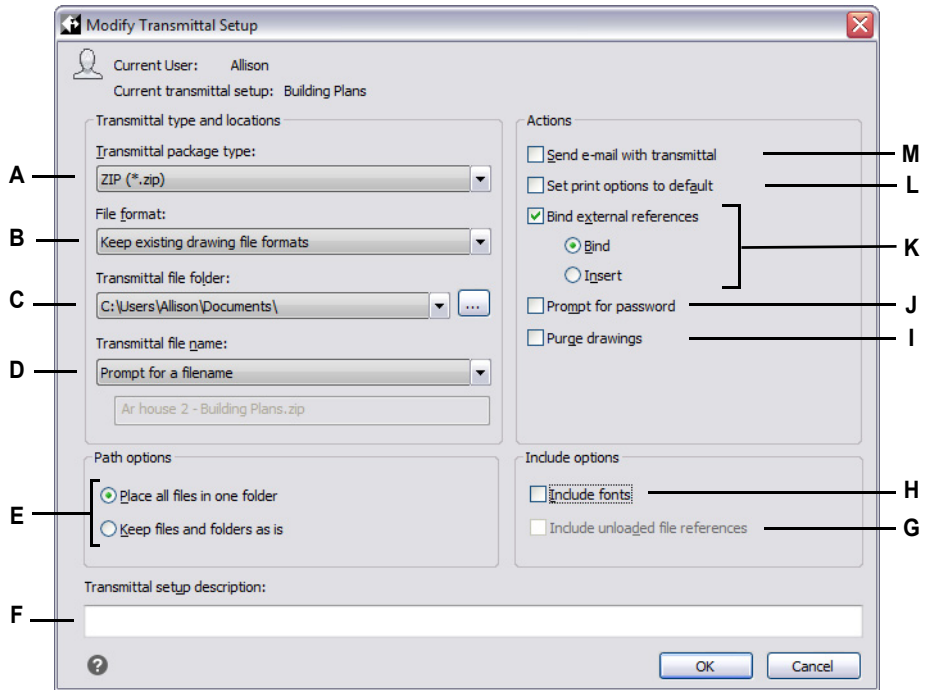
Each time you create a package using the eTransmit command, you choose a transmittal setup to define how the package is created. Each transmittal setup defines the type of package that will be created (.exe, .zip, or folder), filenaming, password control, whether to include font files, and more.

You can create, modify, and delete transmittal setups. Creating more than one transmittal setup is especially helpful if you often need to package files for different recipients with different requirements.

To create or modify a transmittal setup


- 1 Do one of the following to choose eTransmit :
 - On the ribbon, choose the Application button then Publish > eTransmit, or choose Output > eTransmit (in Export).
 - On the menu, choose File > eTransmit.
 - Type *etransmit* and then press Enter.
- 2 Click Transmittal Setups.
- 3 Do one of the following:
 - To create a transmittal setup, click New, enter a name, in Based On select a transmittal setup that has the settings closest to the one you want to create, then click Continue.
 - To modify an existing transmittal setup, select it from the list, then click Modify. The Modify Transmittal Setup dialog box displays.
- 4 In Transmittal Package Type, select the type of package to create when using this transmittal setup. All files can be contained in a new folder, a self-extracting .exe file, or a .zip file.
- 5 In File Format, select the file format for files packaged with this transmittal setup. Files in the package can keep their existing file format, or they can all be converted to the same file format.
- 6 In Transmittal File Folder, enter the default location where this transmittal setup will save packages when they are created. You can type a location, select a recent location from the list, or click [...] to select a folder.
- 7 In Transmittal File Name, select how to name .exe and .zip packages that are created with this transmittal setup. (Available for EXE and ZIP package types.) The following options are available:
 - **Prompt for a filename** — Allows you to enter a filename each time you create a package using this transmittal setup. The prompt displays after clicking OK in the Create Transmittal dialog box.
 - **Overwrite if necessary** — Uses the same filename that you enter each time you create a package using this transmittal setup, and overwrites existing files of the same name. Enter the filename in the field that becomes available.

- **Increment filename if necessary** — Uses the same filename that you enter each time you create a package using this transmittal setup, and increments the filename if an existing file exists with the same name. Enter the filename in the field that becomes available.
- 8 In Path Options, select one of the following:
- **Place all files in one folder** — Saves all files in one folder when files are unpackaged.
 - **Keep files and folders as is** — Saves all files in the original folder structure, which is recreated on the destination computer when files are unpackaged. This can help with file linking when the files are used on another computer.
- 9 Use any of the following options to further define the transmittal setup settings:
- **Transmittal Setup Description** — Enter an optional description for the transmittal setup. The description will display when you select this transmittal setup in the Create Transmittal dialog box.
 - **Send E-Mail with Transmittal** — Select to send the package as an email attachment; the default email application will open after the package is created.
 - **Set Print Options to Default** — Select to remove local printer settings from the package if they will not be applicable on other systems.
 - **Bind External References** — Select to bind or insert all .dwg file external references in the package. Binding makes external references a permanent part of the original drawing file, similar to a block.
 - **Prompt for Password** — Select to prompt for a password when the package is created and when it is unpackaged.
 - **Purge Drawings** — Select to purge all drawings when the package is created.
 - **Include Fonts** — Select to add associated font files (for example, TTF, SHX) in the package.
 - **Include Unloaded File References** — Select to include in the package any unloaded external references, including images and underlays; unloaded files are listed in the Create Transmittal dialog box.




- A Select the type of package the transmittal setup will create.
- B Select a file format option.
- C Specify the default folder location where to save packages. Enter the location, select it, or click [...].
- D Select how to name packages. (Available for EXE and ZIP package types.)
- E Select folder options to use when unpackaging files.
- F Enter a description of the transmittal setup.
- G Select to include any unloaded external references in the package.
- H Select to include font files in the package.
- I Select to purge all drawings when the package is created.
- J Select to protect the package with a password.
- K Select to bind or insert all .dwg file external references in the package.
- L Select to remove local printer settings from the package.
- M Select to send the package as an email attachment.

To rename a transmittal setup

- 1 Do one of the following to choose eTransmit ():
 - On the ribbon, choose the Application button then Publish > eTransmit, or choose Output > eTransmit (in Export).
 - On the menu, choose File > eTransmit.
 - Type *etransmit* and then press Enter.
- 2 Click Transmittal Setups.
- 3 Select the transmittal setup you want to rename.
- 4 Click rename.
- 5 Enter the new name, then press Enter.

To delete a transmittal setup

- 1 Do one of the following to choose eTransmit ():
 - On the ribbon, choose the Application button then Publish > eTransmit, or choose Output > eTransmit (in Export).
 - On the menu, choose File > eTransmit.
 - Type *etransmit* and then press Enter.
- 2 Click Transmittal Setups.
- 3 Select the transmittal setup you want to delete.
- 4 Click Delete, then confirm the deletion.

Sending a drawing through e-mail

You can send an PTCAD drawing to another user via e-mail. PTCAD is compatible with e-mail programs that support the Messaging Application Program Interface (MAPI) protocol.

To include a drawing file in an e-mail message

- 1 While the drawing file is open, do one of the following:
 - On the ribbon, choose the Application button then Publish > Send Mail.
 - On the menu, choose File > Send Mail.
 - Type *mail* and then press Enter.

If your mail program is not already running, it starts; a new e-mail message containing the PTCAD icon and file name appears.

- 2 Address the e-mail, type a message, and send the e-mail message as you would any other message.

To view an PTCAD file sent by e-mail

- Open the e-mail message, and then double-click the PTCAD icon.
software must be installed on the computer used to open drawings from e-mail.

Use the eTransmit command for e-mail if the drawing has supporting files.

If the drawing is dependent on files such as external references or font files, use the eTransmit command to combine the files in a single package and send it by e-mail. When creating the package, use a transmittal setup with the Send Mail with Transmittal option turned on. For more details, see “Packaging drawings and files for use on another computer” on page 809 in this chapter.

Using PTCAD with the Internet

You can use PTCAD to access the Internet and exchange drawing information and perform other tasks, including:

- Add hyperlinks to a drawing.
- Publish drawings to the Internet.
- Drag drawings (.dwg files) directly from a Web site into PTCAD.
- Access the Internet during a drawing session.

Adding hyperlinks to a drawing

In your PTCAD drawings, you can include hyperlinks, which are pointers that take you to another location, such as a Web address or a file on a particular computer. You can attach a hyperlink to any entity in your drawing. Then, when you select that entity, you can open the link and jump to the specific Web address or file location. You can create *absolute* hyperlinks, which store the full path to a file, or *relative* hyperlinks, which store a partial path relative to a base folder or a Uniform Resource Locator (URL).

To create a hyperlink

- 1 Type *hyperlink* and then press Enter.
- 2 Select an entity or entities that you would like to associate with a hyperlink; then press Enter to display the Edit Hyperlink dialog box.
- 3 Do one of the following:
 - Click Browse to specify a file.
 - In the Link to File or URL box, type a Web address.
- 4 If you want to use a common path for all hyperlinks in the drawing, mark Use relative path for hyperlinks. Relative paths are most useful for local files, providing flexibility if you move files to a different folder; you can change multiple hyperlink paths at once, rather than change them all individually.
- 5 Click OK.

The HYPERLINKBASE system variable defines the relative path used for all hyperlinks in the current drawing.

To use the default drawing path, leave the value blank by entering a period (“.”).

To remove a hyperlink

- 1 Type *hyperlink* and then press Enter.
- 2 Select an entity or entities with a hyperlink; then press Enter to display the Edit Hyperlink dialog box.
- 3 Click Remove Link.
- 4 Click OK to close the Edit Hyperlink dialog box.

To access a hyperlink

- Hover the mouse over an entity with a hyperlink, then press Ctrl + click.

To change display options for hyperlinks

- 1 Type *hyperlinkoptions* and then press Enter.
- 2 To display a cursor, tooltip, and shortcut menu for hyperlinks, choose Yes, and then press Enter. Or choose No to turn off the display.

Publishing drawings to the Internet

You can publish your drawings so they can be viewed on the Internet using a Web browser. PTCAD exports your drawing to a Design Web Format (.dwf) file, which can be viewed in a Web browser if Design Review is also installed on the computer. Design Review is a free tool from Autodesk®.

For details about creating a DWF file, see “Click Export.” on page 802 in this chapter.

Inserting drawings from a Web site

Some Web sites are configured to allow you to drag drawings from the Web site directly into your drawing. You can drag-and-drop drawings from any Web site that supports the Autodesk® i-drop technology.

To insert a drawing from a Web site

- 1 Open your Web browser.
- 2 Navigate to a Web site that supports Autodesk® i-drop.
- 3 Position the Web browser and PTCAD windows so they are both visible.
- 4 Click the drawing in your Web browser and drag it to your drawing in PTCAD.

The drawing file is downloaded and inserted into your drawing in PTCAD.

Accessing the PTCAD Web site during a drawing session

In addition to using hyperlinks to access the Internet, you can access a company Web site at any time. At the Web site, you can obtain company information as well as product information and news.

To access the PTCAD Web site

- 1 Do one of the following:
 - On the ribbon, choose Help > PTCAD on the Web.
 - On the menu, choose Help > PTCAD on the Web.
 - Type *onweb* and then press Enter.
- 2 Navigate to the section of your choice on the PTCAD Web site.

Working with digital signatures

Attaching a digital signature to your drawing allows you to sign the drawing and authenticate the signature. First you obtain a digital certificate from a third-party vendor or a self-signed digital certificate, then you simply attach it to your drawings.


Additionally, if you're working with a drawing that has a digital signature from someone else, you can validate the signature.

Attaching digital signatures

Before you can digitally sign a drawing, you need to obtain a digital certificate from a third-party vendor or generate a self-signed digital certificate, then import it to your computer.

After you install your digital signature, you're ready to attach it to one or more drawings.

To attach a digital signature to the current drawing

- 1 Do one of the following to choose Add Digital Signatures (- On the ribbon, choose the Application button then Drawing Utilities > Add Digital Signatures.
- On the menu, choose File > Add Digital Signatures.
- Type *digitalsign* and then press Enter.
- 2 Select Attach Digital Signature After Saving Drawing.
- 3 Select a digital signature from the list of digital signatures that are available on your computer.
- 4 In Comment, add information about the selected digital certificate if needed.
- 5 In Time Stamp, choose a time server to add a time stamp to the drawing.
- 6 Click OK.


To attach a digital signature to multiple drawings

- 1 From the Windows Start menu, choose the program folder where PTCAD is installed, then choose Attach Digital Signatures.
- 2 Select a digital signature from the list of digital signatures that are available on your computer.
- 3 Add drawing files to be digitally signed by doing any of the following:
 - Click Add File, select the drawing files you want to digitally sign, then click Open.
 - Click Add Folder, select a folder that contains the drawing files you want to digitally sign, then click OK. All drawings in the folder will be added to the list.
- 4 In Comment, add information about the selected digital certificate if needed.
- 5 In Time Stamp, choose a time server to add a time stamp to the drawings.
- 6 Click Attach.

Validating a digital signature

If you're working with a drawing that has a digital signature from someone else, you can validate the signature.

To validate an open drawing's digital signature

- 1 Do one of the following to choose Validate Digital Signatures (- On the ribbon, choose the Application button then Drawing Utilities > Validate Digital Signatures.
- On the menu, choose File > Validate Digital Signatures.
- Type *digvalidate* and then press Enter.
- 2 Check the status of the digital signature:
 - Valid — The digital signature is verified by the certificate authority and no changes have been made to the drawing since it was digitally signed.
 - Invalid — The current drawing has been modified since it was digitally signed, the certificate has been revoked by the certificate authority, or the drawing has become corrupted.
 - Not found — A digital signature is not attached to the current drawing.
- 3 Click View Base Signature to view detailed information about the digital signature attached to the drawing. (Unavailable if there is no digital signature attached.)

- 4 If the drawing has external references attached, do any of the following:
 - Mark View Xref Drawings to display the list of all external references attached to the current drawing.
 - Select an external reference and click View Xref Signature to view detailed information about its digital signature. (Unavailable if there is no digital signature attached to the selected external reference.)
- 5 Click OK.

Customizing PTCAD

You can customize PTCAD in a number of ways. For example, you can change the appearance of many aspects of the program and modify the existing menus and toolbars or create new ones. This section explains how to:

- Set the program's preferences.
- Customize the user interface, including tool palettes, menus, toolbars, and the Quick Properties pane.
- Customize keyboard shortcuts, aliases, double-click actions, and image tile menus for frequently used commands.
- Customize entities.
- Create and use scripts.
- Use add-on programs with PTCAD.
- Use additional pointing devices such as a 3D mouse and digitizer tablet.
- Reset the program to its default settings.

Topics in this chapter

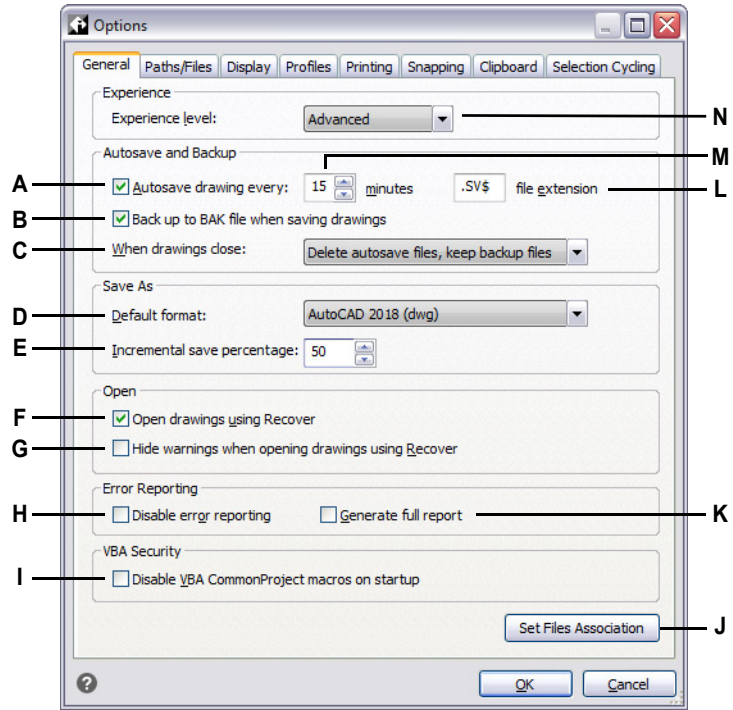
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Setting and changing options

You can change many of the options that control the program's behavior and appearance, such as setting the experience level, specifying file paths and default files, configuring display features, and configuring how certain features work.

Changing the options on the General tab

In the Options dialog box, the General tab contains settings for experience levels, save and open options, and various file options. You can also set VBA security.




- A** Click to save open drawings automatically.
- B** Mark to save an older file version with a .bak extension each time drawings are saved.
- C** Select how to treat autosave and backup files when drawings close.
- D** Select the default file format that displays when saving drawings using the Save As dialog box.
- E** Enter the percentage of a drawing that can be estimated as wasted space before a full save is performed. Enter 0 to turn off incremental saving.
- F** Select to check all drawings for errors when opening them, and attempt recovery, as needed.
- G** Select to hide warning messages when opening drawings. (Available if recover is turned on.)
- H** Click to turn off error reporting when an issue is encountered while running PTCAD.
- I** Click to disable VBA CommonProject macros on startup. (Available in some program versions.)
- J** Click to set filetype associations for PTCAD.
- K** Click to send a complete report of crash data. (Available if error reporting is turned on.)
- L** Type the file extension for autosaved files. (Available if autosave is turned on.)
- M** Enter the autosave frequency. (Available if autosave is turned on.)
- N** Select the experience level.

Setting the experience level

You can control which menus and tools are available by setting the experience level. You can choose from the following experience levels:

- **Beginner** Menus and toolbars display only basic commands.
- **Intermediate** Menus and toolbars display most two-dimensional entity creation and modification commands.
- **Advanced** Menus and toolbars display all available commands.

To set the experience level


- 1 Do one of the following to choose Options (- On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
- On the menu, choose Tools > Options.
- Type *options* and then press Enter.
- 2 Click the General tab.
- 3 Under Experience level, select Beginner, Intermediate, or Advanced.
- 4 When you have finished, click OK.

Saving your drawings automatically

To avoid losing data in the event of a power failure or other system error, save your drawing files often. You also can configure the program to periodically save your drawings automatically. The minutes setting determines the interval between automatic saves. The program restarts this interval timer whenever you save the drawing file.

When autosave is enabled, the program creates a copy of your drawing. The file is saved in the folder specified in Options > Paths/Files for Temporary Files, with the file extension specified in the File Extension box (by default, .SV\$).

To set how drawings are saved automatically

- 1 Do one of the following to choose Options (- On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
- On the menu, choose Tools > Options.
- Type *options* and then press Enter.
- 2 Click the General tab.


- 3 To save your drawings automatically, do the following:
 - Mark Autosave Drawing.
 - In Minutes, enter the frequency that you want to save drawings. Do not use too brief a time if you often work with large drawings, because it can take a minute or more to save a very large drawing.
 - If you want to change the default extension assigned to autosave files, type the new extension in File Extension.
- 4 To save an older version of drawings with a .bak file extension every time a drawing is saved, mark Back Up to BAK File when Saving Drawings.
- 5 In When Drawings Close, choose how to treat autosave and backup files when the program or drawings close.
- 6 When you have finished, click OK.

Setting the default Save As format and incremental save options

You can control the default file format that you want to display in the Save Drawing As dialog box. For example, if you use the Save As command to save most of your drawings in a legacy file format, you can select that file format as the default so you don't have to select it each time you save a drawing using the Save As command. This setting has no effect on saving existing or new drawings using commands other than Save As — PTCAD always saves existing drawings in their current file format and saves new drawings with the most current file format.

You can also enable incremental saving, which saves only the changed parts of drawings instead of re-saving the entire drawing (a full save). This option makes saving drawings, especially large drawings with only a few changes, much faster.

To set the default Save As format and incremental save options

- 1 Do one of the following to choose Options (- On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
- On the menu, choose Tools > Options.
- Type *options* and then press Enter.
- 2 Click the General tab.
- 3 In Default Format, select the default drawing format that displays when using the Save As command. You can always specify a different format in the Save Drawing As dialog box.
- 4 In Incremental Save Percentage, specify what percentage of the drawing file to allocate as blank space for incremental saves. When drawing changes exceed the allocated empty space, a full save is performed the next time a save operation is required. To disable incremental saves, enter 0.

Incremental saves are much faster than a full save but also increase file sizes.

Higher values reduce the number of full saves and increase performance, but also increase the size of the drawing file.


- 5 When you have finished, click OK.

Setting how drawings are opened

There are several options that determine how drawings are opened in PTCAD. You can specify which file extensions are associated with PTCAD, allowing you to open files such as .dwg files automatically using PTCAD.

Additionally, you can set up PTCAD to open drawings automatically using the Recover command, for example, if you are a new PTCAD user and your original drawings were created using different CAD software and those drawings regularly contain errors or damaged data. The Open Drawings using Recover option automatically checks all drawings for errors when using the Open command, and attempts recovery, as needed. Viewing warning messages when opening drawings allows you to know which files are being fixed by PTCAD and what errors have occurred; however, you can also choose to hide the warnings.


To set how drawings are opened

- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options.
 - Type *options* and then press Enter.
- 2 Click the General tab.
- 3 To specify the file types that open automatically using PTCAD, click Set Files Association and make your selections.
- 4 If you want to use the Recover command automatically each time you use the Open command, mark the check box for Open Drawings using Recover.
- 5 If you want to hide warnings when errors are found in a drawing, mark the check box for Hide Warnings when Opening Drawings using Recover. Errors will still be logged in an ASCII file with an .adt extension.
- 6 When you have finished, click OK.

Setting error reporting options

Error reporting occurs when PTCAD encounters a crash issue. You can specify whether error reporting occurs and whether a full report is generated. It is recommended to generate the full report only if requested for troubleshooting purposes. The completed report of crash data can be up to 100MB in size, and while it contains helpful information for troubleshooting issues, is more likely to fail during transmission due to its file size.


To change the options on the General tab

- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options.
 - Type *options* and then press Enter.
- 2 Click the General tab.
- 3 If you want to turn off error reporting, mark the check box for Disable Error Reporting.
- 4 If you want to generate a full report when error reporting is turned on, mark the check box for Generate Full Report.
- 5 When you have finished, click OK.

Disabling VBA CommonProject macros

Each time you start PTCAD, macros are automatically loaded for the Visual Basic Application (VBA) CommonProject. If you do not plan to use VBA, disabling the macros may improve performance. In addition, disabling the macros can enhance security if you are running PTCAD at a low security level.

To change the options on the General tab

- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options.
 - Type *options* and then press Enter.
- 2 Click the General tab.
- 3 If you do not want the CommonProject macros to be loaded when you start PTCAD, under VBA Security, click the check box for Disable VBA CommonProject Macros On Startup.
- 4 When you have finished, click OK.

Changing the options on the Paths/Files tab

You can specify locations for various file types such as those used for drawings, fonts, and menus in the Options dialog box on the Paths/Files tab. You can even specify multiple paths for the same file type. In addition, you can change the names of the default system files that are used for functions such as font mapping and error logging.

Specifying the user paths


You can enter paths to your PTCAD directories by selecting them in the Options dialog box. This feature includes directories for drawings, fonts, help, external references, menus, tool palettes, hatch patterns, blocks, print style tables, print output files, temporary files, templates, and color books. PTCAD searches directories for support files in the following order:

- The PTCAD program directory.
- The current drawing directory.
- The Windows search path.
- The search path specified in the Options dialog box.

You can enter multiple paths for each item.

If, for example, the Drawings item has more than one directory associated with it, click Add to specify additional paths. You can also separate multiple paths with a semicolon if typing them. PTCAD searches the directories in the order in which they are listed.

To specify a user path


- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options.
 - Type *options* and then press Enter.
- 2 Click the Paths/Files tab.
- 3 In the upper half of the dialog box, do one of the following:
 - Right-click and choose from the shortcut menu of options.
 - Click a category to view its search paths, then single-click the path you want to modify, and type the path.

If you do not know the path or directory name, click Browse, and then browse to the location of the directory you want.
- 4 When you have finished, click OK.

Changing the default system files

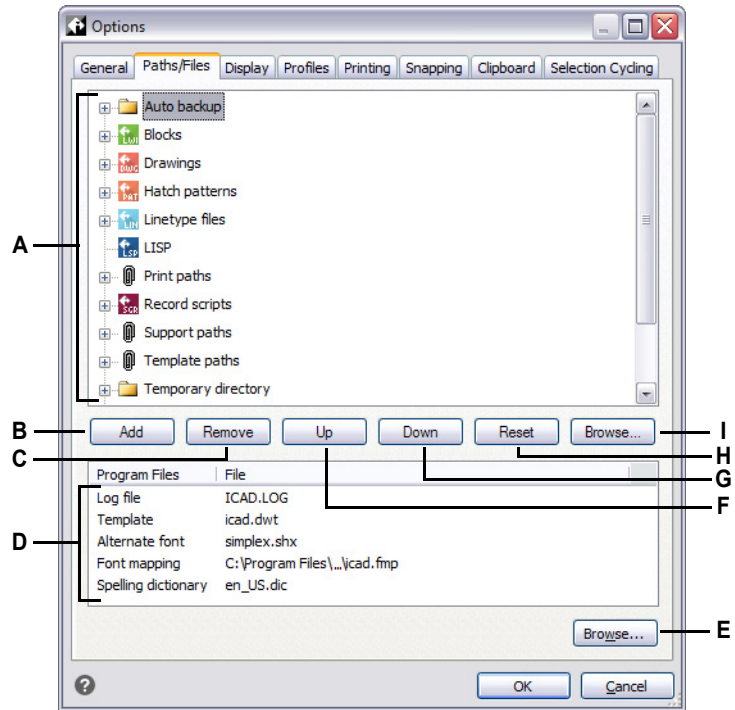
You can change the default system files, including the log file, default template, alternate font, and font mapping file.

To change a default system file

- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options.
 - Type *options* and then press Enter.
- 2 Click the Paths/Files tab.
- 3 In the lower half of the dialog box, under Program Files, do one of the following:
 - Right-click and choose from the shortcut menu of options.
 - Click the file name for the default system file you want to change, and type a new file name.

If you don't know the file name, click Browse, and then browse to the location of the file you want.

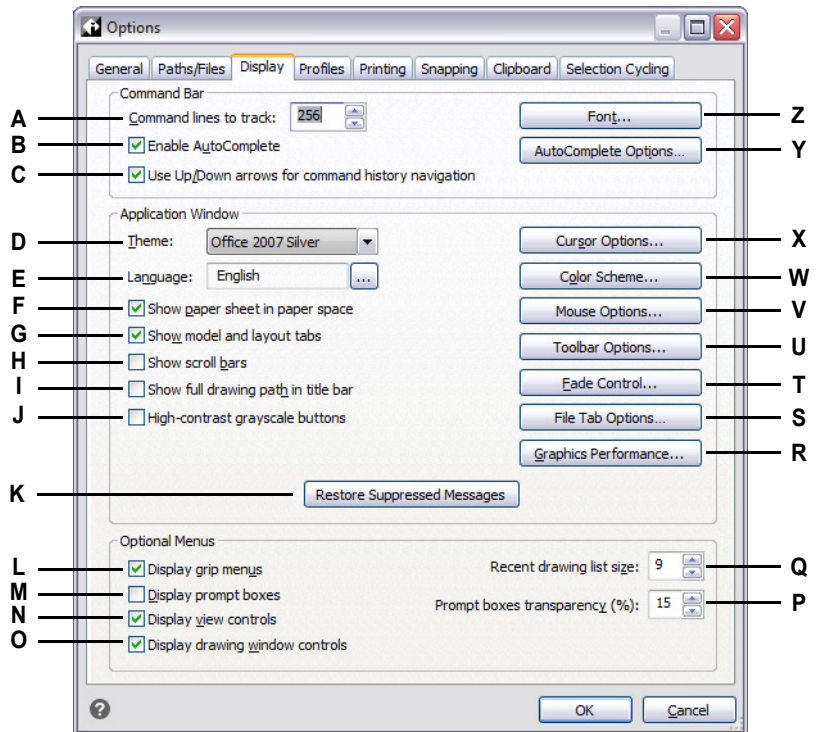
- 4 When you have finished, click OK.



- A** Click to view default paths. Click or press the F2 key to modify a selected path. Double-click a path to open the path in Windows Explorer.
- B** Click or press the Insert key to add a new path for the current category.
- C** Click or press the Delete key to remove the selected path.
- D** Select the default file to change.
- E** Click to locate and specify a new default file.
- F** Click to move the selected path up one position in search order.
- G** Click to move the selected path down one position in search order.
- H** Click to remove custom paths and use the default paths for the selected category.
- I** Click to locate and specify a new path.

Changing the options on the Display tab

In the Options dialog box, the Display tab contains settings for displaying the command bar, PTCAD window, menus, mouse actions, and program language.




- A** Type or select the maximum number of command lines to track.
- B** Select to suggest command names automatically when typing in the command bar.
- C** Select to use Up and Down arrows for scrolling the command history instead of panning.
- D** Select the look for all PTCAD windows.
- E** Displays the language used to show options throughout the program. Click [...] to select language settings.
- F** Select to show or hide a paper sheet in paper space on Layout tabs.
- G** Click to show or hide the Model and Layout tabs.
- H** Click to show or hide scroll bars.
- I** Click to display the drive and folder location of the drawing in the drawing title bar.
- J** Click to display all icons in the user interface in high-contrast grayscale instead of color.
- K** Click to activate any messages that were previously set to not display again.
- L** Click to show or hide grip menus.
- M** Click to show or hide prompt boxes.
- N** Click to show or hide view and visual style controls in the upper-left corner of the current viewport.
- O** Click to show or hide window controls in the upper-right corner of the drawing area.
- P** Type or select the percentage of transparency to apply to prompt boxes when they display.
- Q** Type or select the number of recently used drawings to list on the File menu.
- R** Click to set the graphics device and other performance settings.
- S** Click to specify options for file tabs that can display for each drawing in the main window.
- T** Click to set the fade percentage for items.
- U** Click to specify how tools are displayed on the toolbars and which toolbars display.
- V** Click to select mouse actions.
- W** Click to assign colors to various UI elements of the drawing window.
- X** Click to determine how the pointer displays.
- Y** Click to determine how command names are suggested when typing in the command bar.
- Z** Click to select font options for the command bar.

Setting how the command bar works

The command bar is a dockable window in which you type PTCAD commands and view prompts and other program messages. To display the command bar, use the Command Bar command. To customize how the command bar works, change the options on the Display tab.

To set how the command bar works

- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options.
 - Type *options* and then press Enter.
- 2 Click the Display tab.
- 3 In Command Lines to Track, enter the desired number of previous command and command prompts that are tracked in the command bar. The default value is 256. You can also display the commands in the Prompt History window by pressing F2. To close the window, press F2 again.
- 4 To select font options for displaying the command bar, click Font, then select the font, font style, and font size. Then click OK.
- 5 Mark the Use Up/Down Arrows for Command History Navigation check box if you want to scroll the command history text when using the Up and Down arrows on the keyboard. This can be a convenient way to review and even repeat previous commands. If unchecked, using the keyboard arrows pans your view of the drawing.

Use alternate keyboard shortcuts.

For panning, you can use Alt+arrow keys.

- 6 Mark the Enable AutoComplete check box to use the AutoComplete feature when typing commands in the command bar. For more details about AutoComplete, see “Customizing how suggestions display in the command bar” on page 834 in this chapter.
- 7 When you have finished, click OK.

Colors of command bar items can be customized.

To change the color of various parts of the command bar, use the Options command. For details, see “Setting colors of the main window” on page 840.

Customizing how suggestions display in the command bar


When you type in the command bar, PTCAD suggests names of matching commands as you type. The suggested names appear in an AutoComplete window that automatically opens when you type and closes when you activate a command.

Using AutoComplete is an efficient way to select commands, and it is also a convenient way to view a list of related commands. For example, if you type "LA" in the command bar to work with layers, all layer-related commands that begin with "LA" display in the AutoComplete window.

In addition to command names, suggestions can include names of external commands, system variables, aliases, and LISP functions. Each name displays with a colored icon that indicates its type:

- Red — PTCAD command
- Green — External command
- Yellow — System variable
- Blue — LISP function

To customize how suggestions display in the command bar

- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options.
 - Type *options* and then press Enter.
- 2 Click the Display tab.
- 3 Mark the Enable AutoComplete check box to turn on automatic suggestions of names as you type in the command bar.
- 4 Click AutoComplete Options.

Use a shortcut.

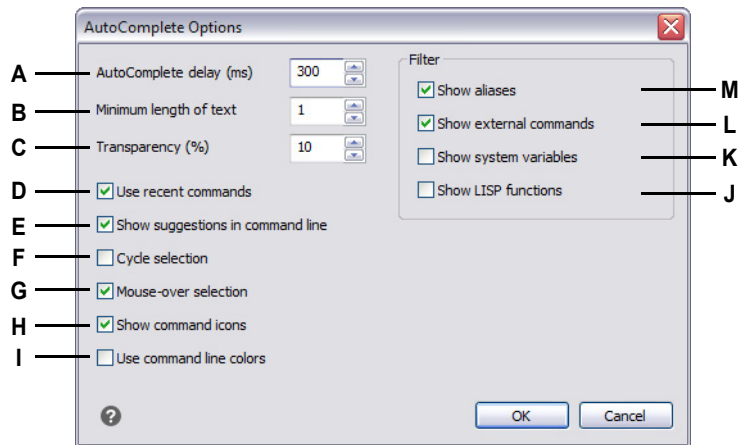
Type INPUTSEARCHOPTIONS and then press enter to access autocomplete options directly from the command bar.

- 5 In AutoComplete Delay, enter the number of milliseconds to wait between keystrokes before displaying the AutoComplete window. The higher the number, the longer the delay allowed between keystrokes (the AutoComplete window displays less frequently).

Fast typists usually increase the timeout setting.

If the AutoComplete window often conflicts with your typing in the command bar, try setting the number of milliseconds to 1,000 or more.

- 6 In Minimum Length of Text, enter the number of letters to be typed in the command bar before displaying the AutoComplete window.
- 7 In Transparency, enter the percentage of transparency, between 0 and 50, in which to display the AutoComplete window. The higher the number, the more transparent the window is. Enter zero for an opaque window.
- 8 In Suggestions List Limit, enter the number of items that can be displayed in the AutoComplete window.
- 9 Determine how selection works:
 - Mark Use Recent Commands to search for and automatically select a recently used command, when possible.
 - Mark Show Suggestions in Command Line to pre-fill the command line with the name as you scroll the list. If turned off, the name does not pre-fill in the command line, however, you can still select the desired name in the AutoComplete window by clicking it or pressing CTRL + Enter.
 - Mark Cycle Selection if you want to allow Arrow key scrolling from the end to beginning (and vice versa) in the list.
 - Mark Mouse-Over Selection to allow selection by hovering the mouse, and not clicking it, over the desired name.
- 10 Mark Show Command Icons to display icons, if available, for each command in the list.
- 11 Mark Use Command Line Colors to display the AutoComplete window using the same background and text colors that are selected for the command bar. For more details about selecting command bar colors, see “Setting colors of the main window” on page 840 in this chapter.
- 12 Mark which items to include in the list of suggested names: aliases, external commands, system variables, and/or LISP functions. If selected, LISP functions display when you enter a parenthesis, “(“, when first typing.
- 13 Click OK.
- 14 Click OK.




- A** Enter the number of milliseconds to wait between keystrokes before displaying the AutoComplete window.
- B** Enter the minimum number of letters to be typed before displaying the AutoComplete window.
- C** Enter the percentage of transparency in which to display the AutoComplete window.
- D** Click to search for and automatically select a recently used command.
- E** Click to pre-fill the command line with the full name as you scroll the list.
- F** Click to allow Arrow key scrolling from the end of the list to the beginning (and vice versa).
- G** Click to allow selection by hovering the mouse over a desired name.
- H** Click to show or hide icons for commands.
- I** Click to use the same colors in the AutoComplete window as specified for the command bar.
- J** Click to include LISP function names.
- K** Click to include system variable names.
- L** Click to include external command names.
- M** Click to include alias names.

Setting the main window options

The main PTCAD window can be customized in many ways to best accommodate your work style. For example, hiding window elements if you do not use them can help increase drawing space in the PTCAD window.

To set the main window options

- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options.
 - Type *options* and then press Enter.
- 2 Click the Display tab.
- 3 In Theme, select the look you want for all PTCAD windows.
- 4 Mark the Show Paper Sheet in Paper Space checkbox if you want to display a bounded sheet of paper for Layout tabs. When unmarked, the paper sheet does not display.
- 5 Mark the Show Model and Layout Tabs checkbox if you want to display the Model tab and Layout tabs in the main window. When unmarked, the tabs do not display, which can be helpful if you only work on the Model tab or if you use the command bar and status bar to switch between tabs.
- 6 Mark the Show Scroll Bars checkbox if you want to display the scroll bars on the right side and bottom of the PTCAD window or viewport. When unmarked, the scroll bars do not display, which can improve performance and can also be helpful if you only use the Pan command to scroll drawings.
- 7 Mark the Show Full Drawing Path in Title Bar check box if you want to display the drive and folder location of the drawing in the drawing title bar in addition to the filename. When unmarked (the default), only the filename displays in the drawing title bar.
- 8 Mark the High-Contrast Grayscale Buttons check box if you want to display all icons in the user interface in high-contrast grayscale instead of color.
- 9 Click Toolbar Options to specify how tools are displayed on the toolbars and which toolbars display. Click OK when finished.
- 10 Click Fade Control to determine the fading percentage for entities and files:
 - **Xref Display** Enter a value or move the slider to specify the fade percentage applied to .dwg file external references.
 - **Entities on Locked Layers** Enter a value or move the slider to specify the fade percentage applied to entities that are located on locked layers.

Use a shortcut.


On the ribbon, choose Locked Layer Fade Control (in Layers), type the percentage, then press Enter.

- **In-place Edit and Annotation Representations** Enter a value or move the slider to specify the fade percentage applied when editing an external reference in place. Fading is applied to the entities of the external reference that are not currently being edited.
- 11 Click OK.
 - 12 Click File Tab Options and select options for displaying drawings on tabs in the main window:
 - **Enable File Tabs** — Select to display each drawing as its own tab in the PTCAD window. If turned off, drawings display as separate windows.
 - **Tab Style** — Select a style in which to display the tabs.
 - **Tab Close Button** — Select to display an [X] for a Close button on the active tab. If turned off, Close buttons do not display on drawing tabs.
 - **Show Icons** — Select to display an icon on each tab that indicates the type of drawing file. If turned off, icons do not display on the drawing tabs.
 - **Top Location** — Select to position the tabs at the top of drawings. If turned off, tabs are positioned at the bottom of drawings.
 - 13 Click OK.
 - 14 Click Restore Suppressed Messages if you want to reactivate any messages that were previously set to not display again.
 - 15 When you have finished, click OK.

Setting how the pointer displays

The pointer can display as a crosshairs instead of the default arrow cursor. If the pointer displays as a crosshairs, you can determine its size and other options.


To set colors of the main drawing window

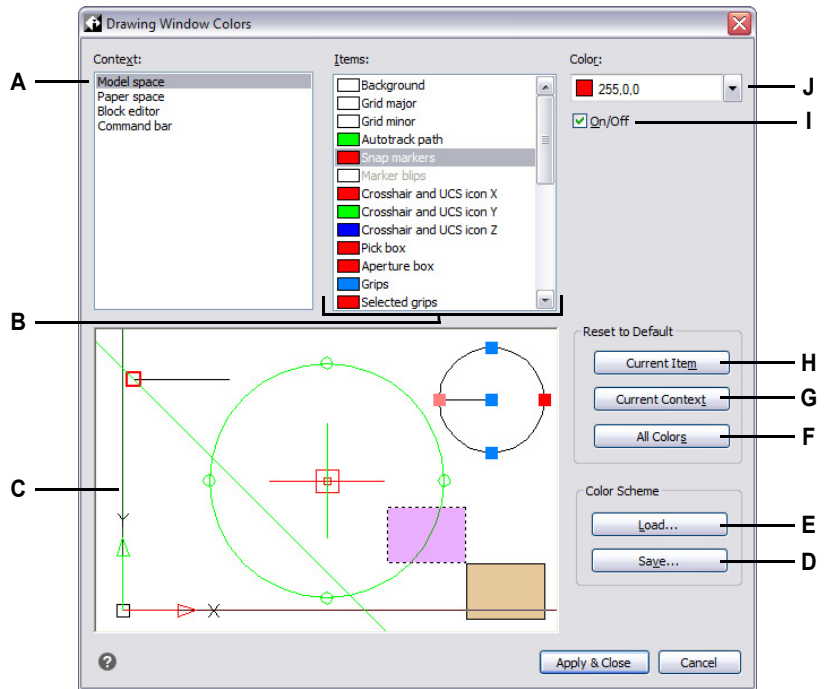
- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options.
 - Type *options* and then press Enter.
- 2 Click the Display tab.
- 3 Click Cursor Options to determine how the pointer displays:
 - **Pointer Defaults to Crosshairs** Click this check box to always display the pointer as the crosshairs (instead of the default arrow cursor with a selection pick box). Note that if this check box is not selected, all subsequent options are unavailable.
 - **Crosshairs Size** Enter a number for the percentage of the screen to be used by the crosshairs cursor, or use the up and down arrows to scroll to the number. Note that depending on your graphics device, a large percentage can negatively affect display performance.
 - **Gun Sight Crosshairs** Click this check box to display the pointer as the gun sight crosshairs instead of the regular crosshairs. You can also control this option by clicking Gun Sight Crosshairs On/Off in the status bar.
 - **Crosshairs Use XOR Invert Colors** Click this check box to turn on use of a logical XOR for the cursor color, which changes the color of the cursor as it moves over solid colors in the drawing so that the cursor is still visible. You can also control this option by clicking XOR Cursor On/Off in the status bar.
- 4 Click OK.

Setting colors of the main window

You can control the color of many aspects of the main drawing window, such as background color, crosshairs color, and more. You can also control the color of items that display in paper space, the block editor, and the command bar.


To set colors of the main drawing window

- 1 Do one of the following to choose Options (- On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
- On the menu, choose Tools > Options.
- Type *options* and then press Enter.
- 2 Click the Display tab.
- 3 Click Color Scheme.
- 4 In Context, select the area that contains items for which you want to specify colors: Model space, Paper space, Block editor, or Command bar.
- 5 In Items, select the item for which you want to specify a color.
- 6 In Colors, select a color or choose Select Color for more options.
- 7 Click On/Off to show or hide the item (available only if the selected item can be turned off).
- 8 When you have finished, click Apply & Close.



- A Select an area of the drawing window.
- B Select an item for which to change color.
- C Displays an example of the current color selections.
- D Click to save the current color settings in a color scheme file.
- E Click to select a color scheme file and load it.
- F Click to restore color defaults for all contexts and items.
- G Click to restore color defaults for the currently selected context.
- H Click to restore the color default for the currently selected item.
- I Click to show or hide the selected item (available only if the item can be turned off).
- J Select a color for the currently selected item. Choose Select Color for more color options.

To save and open color schemes


- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options.
 - Type *options* and then press Enter.
- 2 Click the Display tab.
- 3 Click Color Scheme.
- 4 Make any adjustments before you save the settings.

- 5 Click Save to save the current color selections as a color scheme file.
- 6 Click Load to select a color scheme (.xml or .clr file) and load it.
- 7 Click Apply & Close.

Color schemes are an easy way to reuse favorite color settings.

If you work on more than one computer, save a color scheme to a file and load the file on another computer.

To restore colors to their defaults

- 1 Do one of the following to choose Options (- On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
- On the menu, choose Tools > Options.
- Type *options* and then press Enter.
- 2 Click the Display tab.
- 3 Click Color Scheme.
- 4 Do one of the following:
 - **Reset a single item** — Select the item you want to revert to the default color, then click Current Item.
 - **Reset all items in a context** — Select the context, then click Current Context. All items in the current context will be reverted to their default colors.
 - **Reset all colors** — Click All Colors. All items in all context will be reverted to their default colors.
- 5 Click Apply & Close.


Setting mouse options

Mouse actions can be customized to best accommodate your work style. For example, you may prefer to repeat a recent command when right-clicking the mouse if you do not use shortcut menus.

A 3Dconnexion mouse is set up differently.

For more details, see “Using a specialized mouse for CAD” on page 953 in this chapter.


To set mouse options

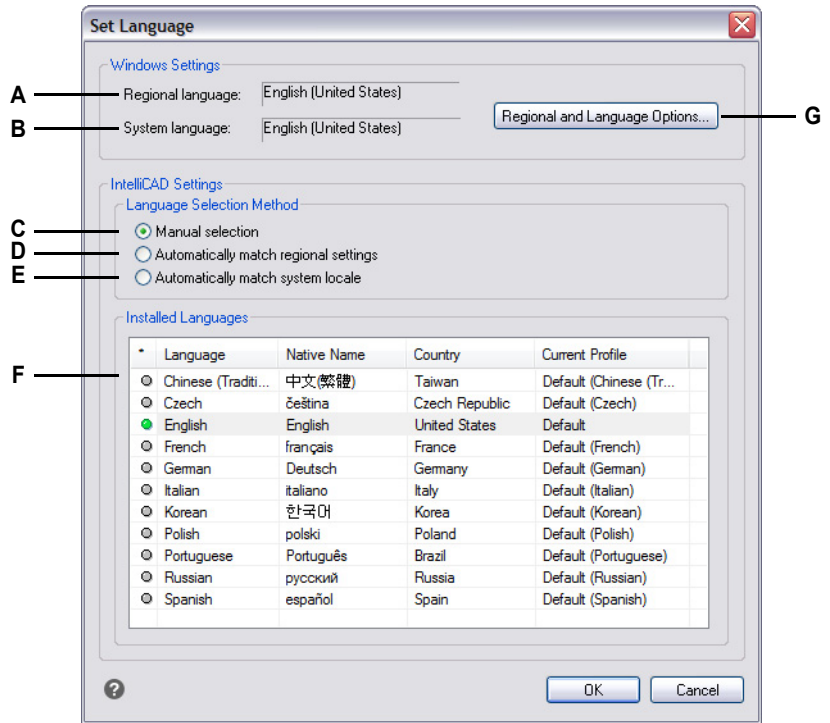
- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options.
 - Type *options* and then press Enter.
- 2 Click the Display tab.
- 3 Mark the Reverse Mouse Wheel Zoom Direction check box if you want to reverse the zoom direction of the mouse wheel, that is, spin the wheel forward to zoom out and spin it backward to zoom in. This can be especially helpful if you use the mouse with your left hand. When unmarked (the default), you spin the mouse wheel forward to zoom in and spin it backward to zoom out.
- 4 Mark the Use Mouse Wheel to Scroll Ribbon Tabs check box if you want the ability to scroll ribbon tabs using the mouse wheel when the mouse hovers over the ribbon.
- 5 To set the action to take when you right-click the mouse in a drawing, choose from the following options:
 - **Without Entities Selected** — To repeat the previously used command if you right-click when entities are not selected, choose Repeat Last Command. To display a shortcut menu if you right-click when entities are not selected, select Show Shortcut Menu.
 - **With Entities Selected** — To repeat the previously used command if you right-click when entities are selected, choose Repeat Last Command. To display a shortcut menu that displays options specific to the selected entities if you right-click, select Show Shortcut Menu. You can right-click anywhere in the drawing with entities selected and the shortcut menu for the selected entities will display.
- 6 When you have finished, click OK.

Setting the language

The list of available languages depends on which languages were installed on your computer. If the desired language doesn't appear in the list, repair or re-install the application with customized settings for the desired language.

To set the language

- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options.
 - Type *options* and then press Enter.
- 2 Click the Display tab.
- 3 In Language, click [...].
- 4 In Language Selection Method, choose one of the following options:
 - **Manual selection** — Uses the language selected in the Installed Languages list.
 - **Automatically match regional settings** — Matches the language set for the geographical region
 - **Automatically match system locale** — Matches the language set for the operating system.
- 5 Click OK.
- 6 When you have finished, click OK.




- A** Displays the language set for the geographical region.
- B** Displays the language set for the operating system.
- C** Select to choose the desired language from the list of installed languages.
- D** Select to automatically match the language set for the geographical region.
- E** Select to automatically match the language set for the operating system.
- F** Displays a list of languages installed for PTCAD. Click the desired language in the list; available only if the manual selection method is chosen.
- G** Click to open the operating system's dialog box for setting language options.

Customizing graphics device options for display performance

Depending on your computer and its available graphics devices, PTCAD automatically determines the best settings for displaying and rendering drawings. In some cases however, users may want to customize certain settings themselves. For example, switching between graphics devices can sometimes resolve performance issues when using specific graphics cards.

To set graphics device options

- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options.
 - Type *options* and then press Enter.
- 2 Click the Display tab.
- 3 Click Graphics Performance.

Use a shortcut.

In the status bar, right-click Switch Graphics Device and choose Settings to open the Graphics Performance dialog box. Or type graphicsconfig.

- 4 In Graphics Device, choose which graphics device to use:
 - **Advanced (OpenGL ES)** Select for high-end graphics cards. This graphics device is recommended for best 3D rendering quality as it supports advanced visual styles and high quality transparency. Note that it can decrease performance.
 - **Intermediate (OpenGL)** Select for mid-level graphics cards. This graphics device is recommended for best performance when working with 2D and 3D wireframe visual styles.
 - **Basic (Software)** Select for entry-level graphics cards that do not support OpenGL.

Use a shortcut.

In the status bar, click Switch Graphics Device to switch to the next graphics device in the list.

- 5 To turn on line smoothing (anti-aliasing) for entities displayed with two-dimensional visual styles, mark Line Smoothing for 2D Visual Styles. Turning on this option improves display quality but can decrease performance. This option is available only for an advanced or intermediate graphics device.
- 6 To turn on line smoothing (multi-sampling anti-aliasing) for entities displayed using three-dimensional visual styles, mark Line Smoothing for 3D Visual Styles.


Turning on this option improves display quality but can decrease performance. This option is available only for an advanced graphics device.

7 Click OK.

Setting how menus display

The display of menus can be customized, including whether prompt menus and right-click shortcut menus display, whether menus load automatically, and the number of drawing files that display on the File menu.

To set how menus display

- 1 Do one of the following to choose Options (- On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
- On the menu, choose Tools > Options.
- Type *options* and then press Enter.
- 2 Click the Display tab.
- 3 Mark the Display Grip Menus check box if you want grip menus to display when the mouse pauses over a grip that has an associated menu. For example, polyline grips have a menu.
- 4 Mark the Display Prompt Boxes check box if you want to show prompt boxes, which display the same options for commands that display on the status bar and the command bar. Turning prompt boxes off may save screen space and may minimize mouse clicks.
- 5 Mark the Display View Controls check box if you want to display a view and visual style control in the upper-left corner of the viewport that is currently active.
- 6 Mark the Display Drawing Window Controls check box if you want to display minimize, maximize, and restore window controls in the upper-right corner of the drawing area.
- 7 In Recent Drawing List Size, enter how many recently opened drawings are listed on the File menu.
- 8 In Prompt Boxes Transparency, enter the percentage of transparency to apply to prompt boxes when they display.
- 9 When you have finished, click OK.

Customizing display options for high resolution monitors or for switching between monitors

PTCAD is designed to run at the proper size on a variety of monitors with different display resolutions. However, if you switch between monitors that have different resolutions, or if you're using a monitor with 4K or higher resolution, you might need to set some options manually.

These custom options are system variables available at the command prompt, not in the Options dialog box.

To set display options for 4K monitors or for switching between monitors

- 1 Type any of the following system variable names and change their settings:
 - **3DORBITCTRSPHERESIZE** — Controls the size of the small orbit sphere that displays at the rotation point when using the Constrained Orbit (3DORBIT), Continuous Orbit (3DCORBIT), and Free Orbit (3DFORBIT) commands. Enter a number from 1 to 20.
 - **APERTURE** — Controls the entity snap selection area in pixels. Enter a number from 1 to 10.
 - **BLIPSIZE** — Controls the blip marker size in pixels. A blip is the plus sign (+) that appears temporarily when you specify a point on the screen. Enter a number from 1 to 255.
 - **GRIPSIZE** — Controls the size of the box (in pixels) used to display grips. Enter a number from 1 to 255.
 - **PDSIZE** — Controls the point display size. The default value (0) specifies a point size 5% of the screen size. A value greater than 0 specifies the absolute size, and a value less than 0 specifies a percentage size.
 - **PICKBOX** — Controls the target height for entity selection in pixels. Enter a number from 0 to 50.
 - **SNAPSIZE** — Controls the entity snap marker size in pixels. An entity snap marker is a temporary glyph that appears when using entity snapping and indicates the current snap location. Enter a number from 1 to 20.

Changing the options on the Profiles tab

PTCAD allows you to customize the settings that control your drawing environment, and then save and restore those settings in a profile. For example, if you prefer working with custom menus and toolbars, you can save these settings as your own profile.

Profiles can be helpful if you have multiple users with different preferences, or if you are a single user who works on various projects that require unique settings. You can even export your profile and bring it with you when you work on a different computer.

Understanding the settings saved in profiles

Profiles save many settings that control the drawing environment. Once you start using a profile, it automatically tracks and stores changes that you make to your drawing environment.

Some settings are saved immediately, but some require that you exit PTCAD and then start PTCAD again. This is because profiles save settings from your computer's registry and some settings are only saved to the registry when you exit PTCAD.

Settings saved in profiles


Setting	Location	When saved
Toolbar settings	Tools > Customize, Toolbars tab	Exit and restart of PTCAD
Menu settings	Tools > Customize, Menus tab	Immediately
Keyboard settings	Tools > Customize, Keyboard tab	Immediately
Alias settings	Tools > Customize, Aliases tab	Immediately
Window elements on/off status and their various settings	View > Display > Command Bar View > Display > Model and Layout Tabs View > Display > Prompt History Window View > Display > Scroll Bars View > Status Bar	Exit and restart of PTCAD
Tablet configurations	Tools > Tablet	Immediately
User paths	Tools > Options, Paths/Files tab	Immediately
System variables	Typed in command bar	Varies — some saved immediately and some upon exit and restart of PTCAD

Creating profiles

Create profiles if you want to save your custom drawing environment settings. This can be helpful if you have two or more drawing environments that you use regularly.

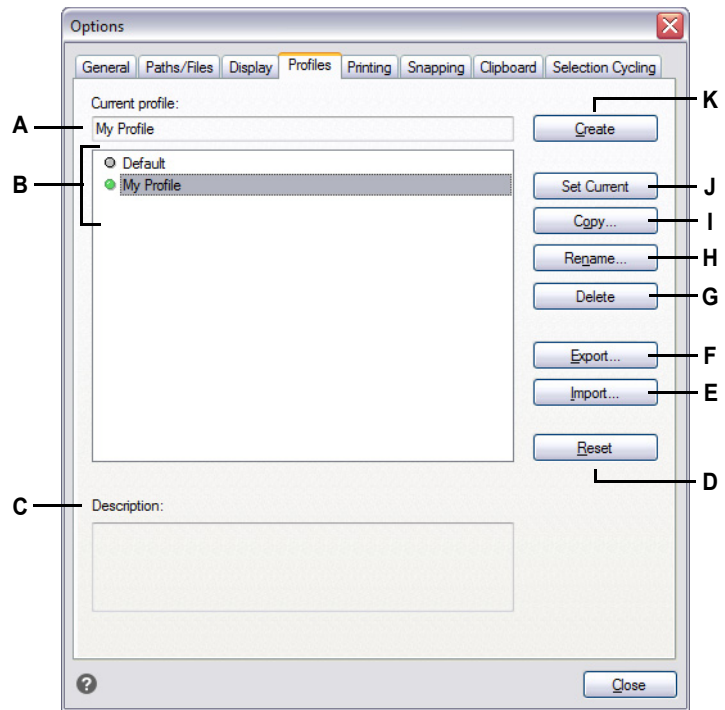
When you create a new profile, the current drawing environment settings are automatically saved with the new profile.

To create a profile

- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options.
 - Type *options* and then press Enter.
- 2 Click the Profiles tab.
- 3 Click Create.
- 4 Enter a name, a description (optional), and then click OK.
- 5 In the Options dialog box, click OK.
- 6 Make changes to your drawing environment.
PTCAD automatically saves the settings to the new profile.

Some cases require you to exit and restart PTCAD before settings are saved with the profile.

This is because profiles save settings from your computer's registry and some settings, such as toolbar settings, are only saved to the registry when you exit PTCAD.




- | | |
|--|---|
| A Displays the name of the currently loaded profile. | G Click to delete the selected profile. |
| B Select a profile to load it or modify it. | H Click to rename the selected profile. |
| C Displays a description of the profile. | I Click to make a copy of the selected profile. |
| D Click to restore the selected profile to the system default settings. | J Click to load the selected profile and make it the active profile. |
| E Click to open a profile stored in an .arg file. | K Click to create a new profile. |
| F Click to save the selected profile in an .arg file. | |

Loading a profile

While you work in PTCAD, you can load the custom settings of any profile. The current profile when you exit PTCAD is automatically loaded when you start PTCAD again.

To load a profile

- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options.
 - Type *options* and then press Enter.
- 2 Click the Profiles tab.
- 3 Select the desired profile.


- 4 Click Set Current.

Restoring the default profile settings

At any time you can return to the default drawing environment settings that were installed with PTCAD.

If the Default profile is unchanged, simply load it to restore the default settings. If the Default profile is deleted or changed, reset an existing profile (one that you no longer need) to replace its contents with the default settings.


To restore default settings using an unchanged Default profile

- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options.
 - Type *options* and then press Enter.
- 2 Click the Profiles tab.
- 3 Select the Default profile.
- 4 Click Set Current.

To restore default settings without using the Default profile

Resetting a profile erases all of the profile's custom settings.


Do this only if you are certain you no longer need the selected profile.

- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options.
 - Type *options* and then press Enter.
- 2 Click the Profiles tab.
- 3 Select a profile that you no longer need; all of its custom settings will be erased. If necessary, create or copy a profile to use for restoring the default settings.
- 4 Click Reset.


Managing profiles

Once you start using profiles, you may need to rename, copy, or delete them. Copying a profile is a quick way to create a new profile based on an existing profile.


To rename a profile

- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options.
 - Type *options* and then press Enter.
- 2 Click the Profiles tab.
- 3 Select the profile you want to rename.
- 4 Click Rename.
- 5 Make any necessary changes to the name or description, and then click OK.

To copy a profile

- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options.
 - Type *options* and then press Enter.
- 2 Click the Profiles tab.
- 3 Select the profile you want to copy.
- 4 Click Copy.
- 5 Enter a new name, a description (optional), and then click OK.

To delete a profile


- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options.
 - Type *options* and then press Enter.
- 2 Click the Profiles tab.
- 3 Select the profile you want to delete.
- 4 Click Delete.

Working with profiles on multiple computers


If you use multiple computers and you like to work with your own drawing environment settings, save time by bringing your profile with you.

On your computer, export your profile to an .arg file. Bring the file with you to the other computer using a disk, E-mail, network, or some other method. When you start working at another computer, simply open and load your profile instead of recreating your preferred drawing environment.

To export a profile to a file

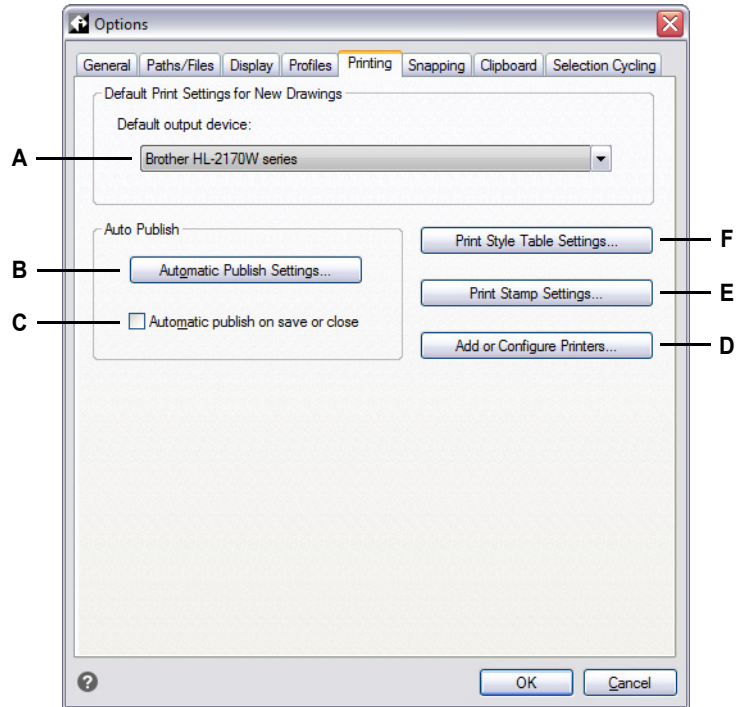
- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options.
 - Type *options* and then press Enter.
- 2 Click the Profiles tab.
- 3 Select the profile to export.
- 4 Click Export.
- 5 Specify a location and name for the exported file, and then click Save.

To open a profile from a file

- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options.
 - Type *options* and then press Enter.
- 2 Click the Profiles tab.
- 3 Click Import.
- 4 Locate and select the profile (.arg file), and then click Open.
- 5 Make any necessary changes to the name or description, and then click OK.
- 6 (Optional) To load the imported profile, select it, and then click Set Current.

Changing the options on the Printing tab

In the Options dialog box, on the Printing tab, you can determine several print settings, including the default printer, headers, footers, printer configuration files (PC3 files), and print styles that change the appearance of your printed drawing without modifying the actual entities in your drawing.




- | | |
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| <p>A Select a default printer to assign to new drawings.</p> <p>B Click to specify settings that are used when automatically publishing drawings.</p> <p>C Choose if you want drawings to be automatically published when they are saved or closed.</p> | <p>D Click to specify settings for printer configuration files (PC3 files).</p> <p>E Click to specify a header or footer for printed drawings.</p> <p>F Click to specify print style table settings for new drawings created without a template or legacy drawings.</p> |
|--|--|

Setting the default printer

A default printer, or output device, is assigned to all new drawings. Select the device you use most often when printing drawings. Later, if needed, you can assign a different printer for any existing drawing.


To set the default printer

- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options then click the Printing tab or choose File > Printer Options.
 - Type *options*, press Enter, then click the Printing tab.
- 2 In Default Output Device, select a printer to assign to new drawings.
- 3 When you have finished, click OK.

Setting automatic publishing options

A default printer, or output device, is assigned to all new drawings. Select the device you use most often when printing drawings. Later, if needed, you can assign a different printer for any existing drawing.

To set automatic publishing options

- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options then click the Printing tab or choose File > Printer Options.
 - Type *options*, press Enter, then click the Printing tab.
- 2 Click Automatic Publish Settings to specify settings that are used when automatically publishing drawings. For more details about these settings, see “Publishing a sheet list automatically” on page 684.
- 3 To turn on automatic publishing when you save or close drawings, mark Automatic Publish on Save or Close.
- 4 When you have finished, click OK.

You can also set up automatic publishing options using the Auto Publish command.


For more details, see “Publishing a sheet list automatically” on page 684.

Setting default print styles

Default print style settings affect only specific drawings: new drawings created without a template and older drawings when opened (older drawings that were created before print styles were available, for example, before AutoCAD 2000). Drawings that are currently opened are not affected.

For more details about print styles and print style tables, see “Using print styles” on page 648.

To set default print style settings for new drawings


- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options then click the Printing tab or choose File > Printer Options.
 - Type *options*, press Enter, then click the Printing tab.
- 2 Click Print Style Settings.
- 3 Select to use color-dependent or named print style tables for new drawings created without a template.
- 4 Select a default print style table to use with new drawings.
- 5 For named print style tables, select both the print style to assign to layer zero and to new entities. For color-dependent tables, the print style is BYCOLOR and is not selectable.
- 6 Click OK.
- 7 When you have finished, click OK.

Specifying a header and footer

You can include information such as a date and time stamp, your name and company name, or other information that you want to appear at the top or bottom of drawings when you print them.

Header and footer settings are set globally for all drawings, although they don't have to be included with each drawing you print. When printing, in the Print dialog box mark or unmark Print Stamp On accordingly.

To specify a header and footer for all drawings

- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options then click the Printing tab or choose File > Printer Options.
 - Type *options*, press Enter, then click the Printing tab.
- 2 Click Print Stamp Settings.
- 3 Type the content for the header and footer, or select an optional automatic field from the lists.

To align text to the left, middle, or right in a header or footer, separate the text with commas.
- 4 Click Advanced.
- 5 Customize settings for any of the following:
 - **Orientation** Select Horizontal to place the header and footer on the top and bottom of the drawing. Select Vertical to rotate the header and footer 90 degrees on the left and right of the drawing.
 - **Print Upside Down** Mark to rotate the print stamp upside down.
 - **X Offset** Enter the distance to offset the header and footer from the edge of the printable area in the x-direction.
 - **Y Offset** Enter the distance to offset the header and footer from the edge of the printable area in the y-direction.
 - **Offset Relative To** Select whether to measure the offset from the edge of the paper or the printable area.
 - **Font** Select the font for the header and footer text.
 - **Height** Select the height for the header and footer text.
 - **Color** Select the color for the header and footer text in the Color dialog box that opens.

- **Units** Select Inches or Millimeters as the unit of measure for the print stamp X Offset, Y Offset, and Height.
- **Add Print Event into Log File** Select to include print stamp information in the print log file.

6 Click OK.


7 When you have finished, click OK.

Setting up printer configuration files

Printer configuration files (PC3 files) store the printer information you use for specific drawings or layouts. You can add, modify, and delete PC3 files.

For more details about printer configuration files, see “Using printer configuration files” on page 643.


To set up printer configuration files

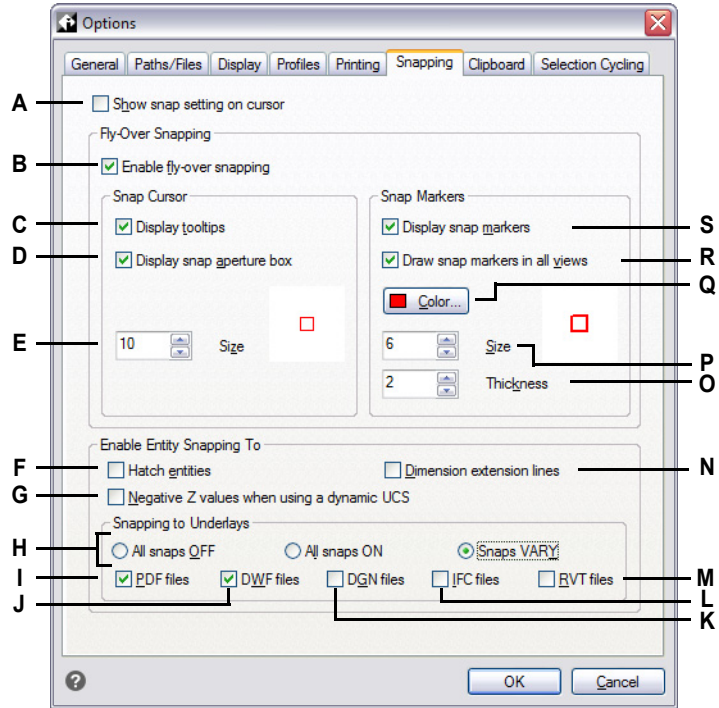
- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options then click the Printing tab or choose File > Printer Options.
 - Type *options*, press Enter, then click the Printing tab.
- 2 Click Add or Configure Printers.
- 3 To create a new PC3 file, do the following:
 - Click Add.
 - In the Add Printer Configuration File dialog box, select the desired printer for the new PC3 file.
 - Click Continue.
 - Select the options you want for the PC3 file. If you don't select any custom options, a PC3 file will not be created.
 - Click OK.
- 4 To modify a PC3 file, do the following:
 - Select the desired file in the list.
 - Click Modify.
 - Select the options you want for the PC3 file.
 - Click OK.
- 5 To delete a PC3 file, select the desired file in the list and click Delete.
- 6 Click OK.
- 7 When you have finished, click OK.

Changing the options on the Snapping tab

In the Options dialog box, on the Snapping tab, you can control how entity snaps work, including fly-over snapping. Fly-over snapping is a visual aid to help you see and use entity snaps more efficiently.

To change the options on the Snapping tab

- 1 Do one of the following to choose Options (- On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
- On the menu, choose Tools > Options.
- Type *options* and then press Enter.
- 2 Click the Snapping tab.
- 3 Select the options you want.
- 4 When you have finished, click OK.




- A Select to display an extra icon on the cursor to show which entity snap is active. (Available even if fly-over snapping is turned off.)
- B Select to turn on fly-over snapping.
- C Select to turn on fly-over snap tooltips, which indicate the type of snap that was used to select the marked location.
- D Select to turn on the fly-over snap aperture box. Entities found within the aperture box are available for selection, making it easier to find and select entity snap points.
- E Type or scroll to the tolerance size for the fly-over snap aperture box. Higher numbers increase the distance from the cursor in which entities are found.
- F Click to turn on entity snapping to hatches.
- G Click to turn on entity snapping to entities that have a negative z value when using a dynamic UCS.
- H Select whether there is no snapping to files attached to drawings as underlays, snapping is turned on for all underlays, or snapping is turned on but varies according to the file type.
- I Click to turn on entity snapping to .pdf files that are attached to drawings. (Available only if snapping to underlays is turned on to VARY.)
- J Click to turn on entity snapping to .dwf files that are attached to drawings. (Available only if snapping to underlays is turned on to VARY.)
- K Click to turn on entity snapping to .dgn files that are attached to drawings. (Available only if snapping to underlays is turned on to VARY.)
- L Click to turn on entity snapping to .ifc files that are attached to drawings. (Available only if snapping to underlays is turned on to VARY.)
- M Click to turn on entity snapping to .rvt and .rfa files that are attached to drawings. (Available only if snapping to underlays is turned on to VARY.)
- N Turns on entity snapping to dimension extension lines.
- O Type or scroll to the thickness of the fly-over snap marker.
- P Type or scroll to the size of the fly-over snap marker.
- Q Click to choose the color of the fly-over snap marker.
- R Select to turn on the display of fly-over snap markers in all views when you are using more than one viewport.
- S Select to turn on fly-over snap markers, which mark snap points on entities.

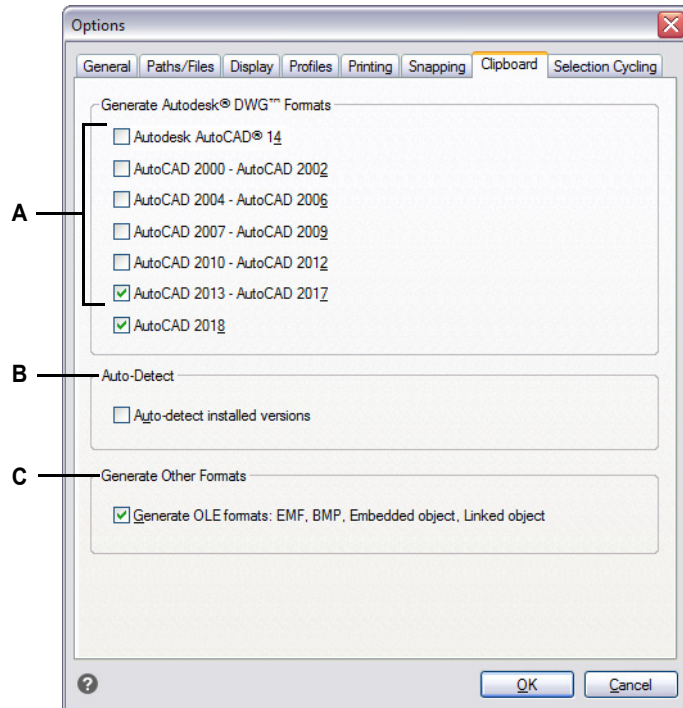
Changing the options on the Clipboard tab

You can control what formats are supported when copying content to the clipboard from PTCAD.

Copying all supported formats to the clipboard impacts performance—it is best to select only the necessary formats.

To change the options on the Clipboard tab

- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options.
 - Type *options* and then press Enter.
- 2 Click the Clipboard tab.
- 3 Select the options you want.
- 4 When you have finished, click OK.




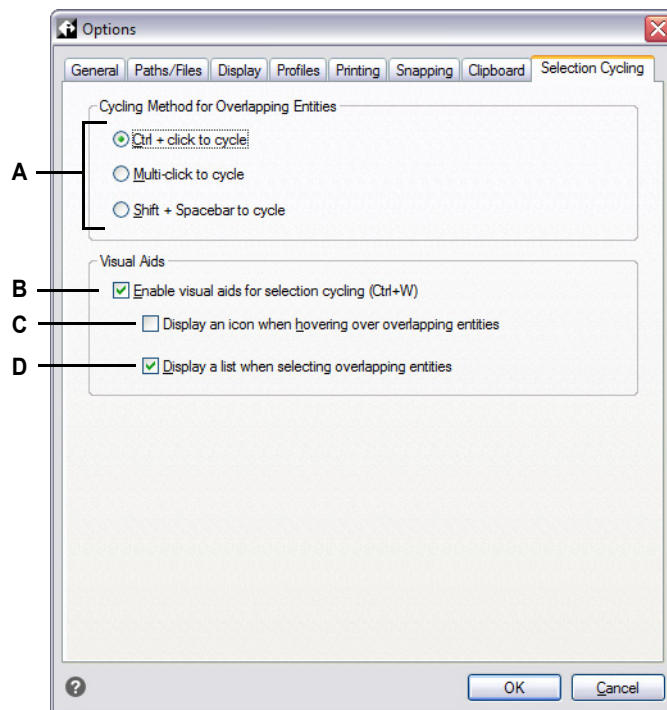
- A** Select which DWG format versions to copy to the clipboard.
- B** Select to copy the formats for installed versions automatically.
- C** Select to copy EMF, BMP, embedded object, and linked object formats to the clipboard.

Changing the options on the Selection Cycling tab

Selection cycling allows you to cycle through overlapping entities under the cursor. Use the Selection Cycling tab to determine how selection cycle works.

To change the options on the Selection Cycling tab

- 1 Do one of the following to choose Options ():
 - On the ribbon, choose the Application button then choose Options, or choose Tools > Options (in Manage).
 - On the menu, choose Tools > Options.
 - Type *options* and then press Enter.
- 2 Click the Selection Cycling tab.
- 3 Select the options you want.
- 4 When you have finished, click OK.



- | | |
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| <p>A Choose to cycle through overlapping entities by pressing Ctrl with a mouse click, multiple mouse clicks, or pressing Shift + Space.</p> <p>B Select to turn on visual aids when selecting entities that overlap.</p> | <p>C Select to display an icon at the cursor when you hover over overlapping entities.</p> <p>D Select to display a Selection Cycling list when you select overlapping entities.</p> |
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Customizing tool palettes

Tool palettes contain commands and blocks that are organized for easy access while working in drawings. Each tool palette is customizable according to your needs: add new palettes, delete existing palettes, and change the order in which they display on the pane. You can also control the transparency applied to the display of any palette, which is helpful if you want to view parts of a drawing that would otherwise be hidden by opaque tool palettes.

For details about displaying and positioning tool palettes, see “Displaying and hiding the Tool Palettes pane” on page 26.

Creating tool palettes and adding commands and blocks

Each tool palette that you create can contain any combination of tools that either run a command or insert a block.

To create a new tool palette

- 1 Right-click the name of an existing tool palette.
- 2 Choose New Palette.
- 3 Type the name of the new palette, then press Enter.

To add a command to a tool palette

- 1 On a tool palette, right-click an existing command tool and choose Copy.
- 2 Right-click the tool area of the desired palette and choose Paste.
- 3 Right-click the pasted command and choose Properties to change its settings:
 - **Name** Enter the name of the command. This name displays in the tool palette.
 - **Description** Type a one-sentence description of the tool.
 - **Command Name** Enter the command carried out when the tool is selected, which can contain a macro (one or more commands and their options), such as: `^C^C^CZOOM;E;MSNAPSHOT`
For details about special characters used in macros, see “Using macros in .cui files” on page 931 in this chapter.
 - **General** Select the color, layer, linetype, linetype scale, lineweight, text style, and dimension style to apply when using the command. Choose Use Current to use the current property.
- 4 Click OK.

Use a shortcut.

*Right-click a tool palette and choose **Customize Commands** or type quickcui. Then drag and drop commands from the list of commands to the tool palette.*

*Or, drag an entity (for example, a line) from the drawing to the tool palette, which adds the corresponding entity-creation command (for example, the **Line** command) to the tool palette.*

To add a block to a tool palette

- 1 Open a drawing that contains blocks, or if necessary create and insert a block. For details, see “Working with blocks” on page 546.
- 2 Save the drawing. For details, see “Saving your drawing” on page 108.
- 3 Select the block in the drawing and drag-and-drop it to the desired tool palette.
- 4 Right-click the new tool and choose **Properties** to customize it.

Modifying tool palettes

The Tool Palettes pane can contain multiple palettes. You can rename, reorder, and delete tool palettes, and you can also change the transparency in which they display.

To change the name of a tool palette

- 1 Right click the name of the tool palette you want to rename.
- 2 Choose **Rename Palette**.
- 3 Type a new name, then press **Enter**.

To change the order in which tool palettes display

- 1 Right click the name of the tool palette.
- 2 Choose **Move Up** or **Move Down**.

To delete a tool palette

- 1 Right-click the name of the tool palette you want to delete.
- 2 Choose **Delete Palette**.
- 3 Click **Yes** to confirm the deletion or **No** to cancel.

To change the transparency of tool palettes

- 1 Right-click the Tool Palette pane title bar.
- 2 Choose Transparency.
- 3 Select the desired options:
 - **General** Move the slider to specify the transparency in which panes display in the program. The lower the number, the more transparent the pane is.
 - **Rollover** Move the slider to specify the transparency in which panes display when the mouse moves over them. The lower the number, the more transparent the pane is. The transparency for mouse rollover cannot be less than the general transparency.
 - **Preview** Click to see a preview in the program of the mouse rollover transparency. You might need to move the Transparency dialog box to see a pane.
 - **Apply settings to all palettes** Mark this check box to apply selected transparency options to the display of all panes in the program, including any custom panes.
 - **Disable transparency for all palettes** Mark this check box to turn off transparency options for all panes in the program, including any custom panes.
- 4 Click OK.

Modifying the tools on a tool palette

Individual tools on a tool palette can be modified in several ways: reorder the display of tools, change the image that displays for any tool, and settings such as name, command or block that is assigned, and more. You can also delete tools from any palette.

To change the name of a tool

- 1 Right-click the tool you want to rename, then choose Rename.
- 2 Type a new name, then press Enter.

To reorder tools on a tool palette

- 1 Click and drag a tool to move it up or down.

To add a separator between tools

- 1 Right-click the tool below where you want to add a separator.
- 2 Choose Insert Separator.

To delete a tool or separator from a tool palette

- 1 Right-click the tool or separator you want to delete, then choose Delete.

To specify an image to display for a tool

- 1 Right-click a tool, then choose Image.
- 2 Do one of the following to select the image you want to display on the tool palette for the tool:
 - Select the desired image in the Tool Image dialog box. This adds the image by resource ID.
 - Click Browse to load a .dll external resource file that contains bitmap resources, a .bmp file, or a .png file. The loaded images must be 32x32 pixels.
- 3 When you have finished, click OK.

To change the settings of a tool on a tool palette

- 1 Right-click the tool and choose Properties.
- 2 Customize the name and description of the tool:
 - **Name** Enter the name of the tool. This name displays in the tool palette.
 - **Description** Type a one-sentence description of the tool.
- 3 If the tool is a command, you can customize these options:
 - **Command Name** Enter the command carried out when the tool is selected, which can contain a macro (one or more commands and their options), such as: `^C^C^CZOOM;E;MSNAPSHOT`
 - **General** Select the color, layer, linetype, linetype scale, lineweight, text style, and dimension style to apply when using the command. Choose Use Current to use the current property.

If the tool is a block, you can customize these options:

- **Name** Enter the name of the block to insert when using this tool.
 - **Source File** Enter the filename and path of the drawing that contains the source block.
 - **Scale** Enter the x-, y-, and z-scale of the block.
 - **Rotation** Enter the rotation of the block.
 - **Prompt Rotation** Choose Yes to prompt for the block rotation value when inserting the block using this tool, or choose No.
 - **Explode** Choose Yes to allow blocks inserted with this tool to be exploded, or choose No.
 - **General** Select the color, layer, linetype, linetype scale, lineweight, text style, and dimension style to apply when inserting the block. Choose Use Current to use the current property.
- 4 Click OK.

Working with groups of tool palettes

The Tool Palettes pane displays one group of palettes at a time. If you don't create palette groups, all palettes display on the pane. If you organize palettes into groups, you can easily switch the palettes that display on the Tool Palettes pane, simply make a group current to view its palettes on the Tool Palettes pane.

Palettes can be organized into groups, which can be used to quickly switch between groups of palettes.

Working with tool palette files

Tool palettes can be exchanged between computers by importing and exporting them as .xtp files. Tool palette groups can be imported and exported as .xpg files.

To export a tool palette to a file

- 1 Do one of the following to choose Customize:
 - Type *customize* and then press Enter.
 - Right-click anywhere on a tool palette and choose Customize Palettes.
- 2 Right-click the palette you want to save.
- 3 Choose Export.
- 4 Enter a name for the tool palette file (.xtp file), then click Save.

To export all tool palettes to a file

- 1 Do one of the following to choose Customize:
 - Type *customize* and then press Enter.
 - Right-click anywhere on a tool palette and choose Customize Palettes.
- 2 Right-click a palette group.
- 3 Choose Export All.
- 4 Enter a name for the tool palette group file (.xpg file), then click Save.

To import a tool palette file

- 1 Do one of the following to choose Customize:
 - Type *customize* and then press Enter.
 - Right-click anywhere on a tool palette and choose Customize Palettes.
- 2 Right-click a palette or a palette group.
- 3 Choose Import.
- 4 Select the tool palette file (.xtp file) or tool palette group file (.xpg file) to load, then click Open.

Customizing menus

You can customize a current menu and save your changes as a file. Menu files can also be loaded. Loadable menu file formats can be any of the following: PTCAD (*.icm files), Customizations (.cui files), and AutoCAD (*.mnu, *.mns files).

Understanding menu compatibility

CUI format menu files are created by newer versions of AutoCAD. MNU format files are menu files created by all AutoCAD releases, and MNS format files are included in AutoCAD Releases 13, 14, and 2000. PTCAD reads all file formats, even when menu macros include AutoLISP code. This feature allows you to continue using your existing AutoCAD menus.

PTCAD supports all sections of CUI format files required for customization.

For MNU and MNS format file compatibility, see the following table.

PTCAD support of specific sections in AutoCAD MNU and MNS format files

Menu section	Definition	PTCAD support
***POP0	Cursor menu	Supported
***POP n	Pull-down menus	Supported
***AUX n	Auxiliary menus	Not supported
***BUTTON n	Button menus	Supported
***ICON	Icon menus	Not supported
***SCREEN	Screen menus	Not supported
***TABLET n	Tablet menus	Supported

To see how PTCAD reads AutoCAD menu source files

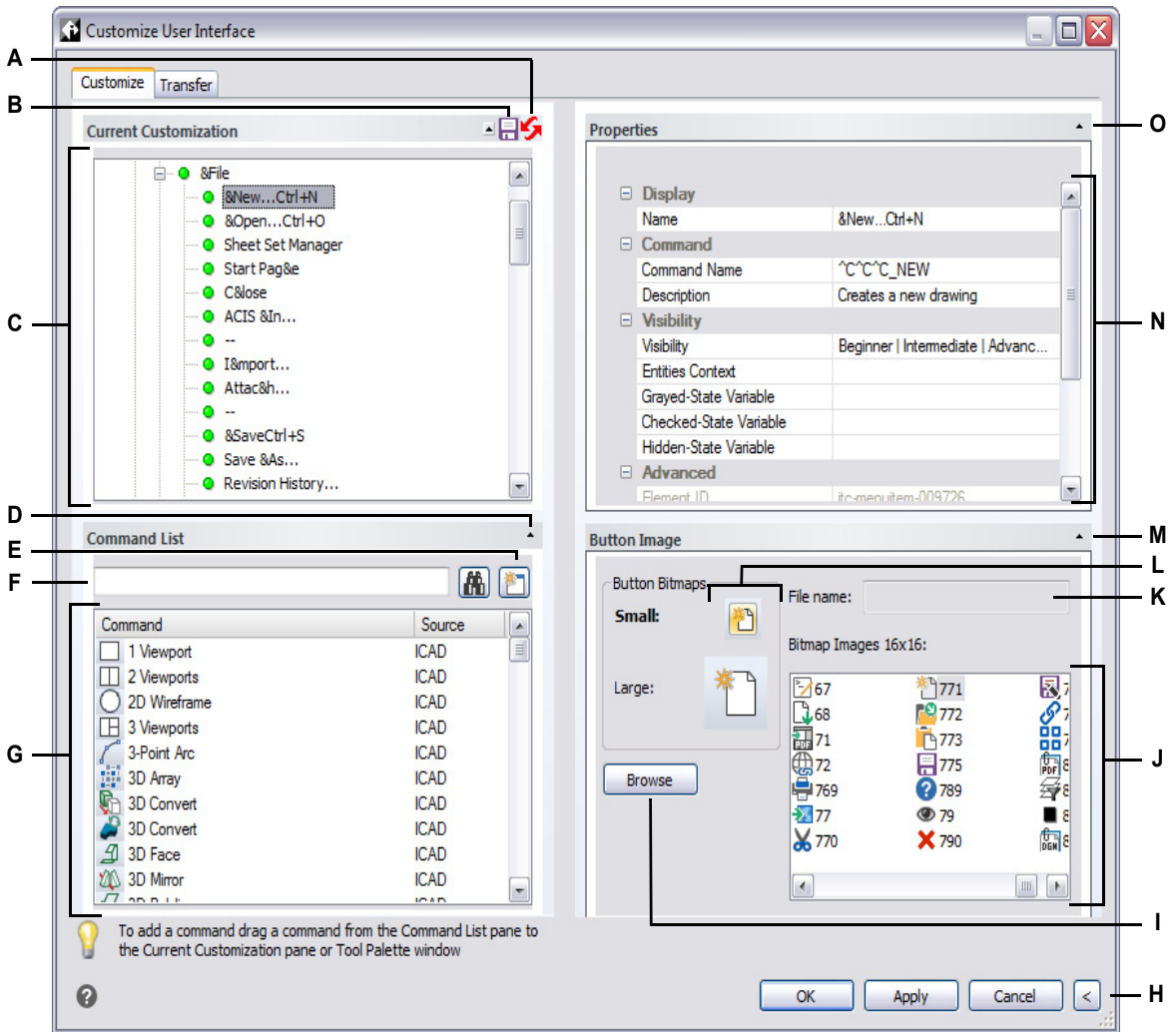
- 1 Type *menu* and then press Enter.
- 2 Under Files Of Type, select AutoCAD Menu File (mnu).
- 3 In the Open Menu dialog box, go to the AutoCAD Support folder and select the Acad.mnu file (or Acad.mns for Releases 13, 14, and 2000).
- 4 To load the AutoCAD menu file into PTCAD, click Open.
The PTCAD menu bar now looks identical to the AutoCAD menu bar.
- 5 To see how it works, choose a few commands from the menu bar, such as File > Open or Draw > Line.
- 6 To restore PTCAD to its default user interface, choose Tools > Options, click the Profiles tab, and then click Reset.

Creating new menus and adding commands

You can create a new menu by inserting a menu item at the top level of the Menus list. Then you can add commands to the new menu item. You can also add submenus and modify existing menu names and commands by adding, deleting, and rearranging them.


Bullet colors indicate whether a command is available at the current experience level.

A green bullet in front of a menu item or command indicates that the menu item or command is available; a red bullet in front of a menu item or command indicates that the menu item or command is not available for you to use at the experience level you have set. To change the experience level for an item, use the `property` in the `Properties` area. To change your experience level for the user interface, choose `Tools > Options`, then the `General` tab.




- A Click to restore the default customization or restore from a backup.
- B Click to save the customization.
- C Select a menu, submenu, or a menu's command.
- D Click to show or hide the Command List area.
- E Click to create a new command.
- F Type a command name for which to filter the command list.
- G Displays the list of commands and which source menu group they belong to. Select a command to change its button images in Button Image or its properties in Properties. Drag and drop a command to the Current Customization. Right-click for more options.
- H Click to hide the Properties and Button Image areas.
- I Click to select an image file or .dll that contains the image to assign to the currently selected button. If you load an image file or .dll file, you can click Default to reset the images that display in Bitmap Images to the images that were installed with PTCAD.
- J Select a button image for the selected small or large button.
- K Displays the full path and filename of the .dll, .bmp, or .png file that was loaded when you clicked Browse.
- L Select which button you want to customize: small or large.
- M Click to show or hide the Button Image area.
- N Displays properties assigned to the selected menu item in Current Customization or the selected command in Command List. Enter a new value or click [...] to select more options for each property.
- O Click to show or hide the Properties area.

To create a new menu

- 1 Do one of the following to choose Customize UI (- On the ribbon, choose Tools > Customize UI (in Manage).
- On the menu, choose Tools > Customize UI.
- Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization area, select either a workspace or a customization file (for example, ICAD), then expand Menus in the list. The menus that display on the menu bar in the user interface are shown in the list with a plus sign (+) next to each name.
- 4 Right-click the menu name located below the location of the new menu.
- 5 Choose Insert Menu Item.
- 6 Type a name for the new pull-down menu, and then press Enter.
- 7 Add a command to the menu (see next). To see the new menu, you must add a command to it.
- 8 Click OK.

To add a command and customize a menu

- 1 Do one of the following to choose Customize UI (- On the ribbon, choose Tools > Customize UI (in Manage).
- On the menu, choose Tools > Customize UI.
- Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization area, select either a workspace or a customization file (for example, ICAD), then expand Menus in the list.
- 4 Expand a menu name to show the menu sub-items and commands that display on each menu.
- 5 From the Command List, drag a command to where you want it located in the Current Customization list.
- 6 To further customize a menu, right-click an item located below where you want to add a new item, and choose one of the following:
 - **Insert Menu Item** Inserts a command above your selection. Enter the name as you want it to appear on the menu in the user interfaces, then press Enter. In Properties, enter the command name for the menu item in Command Name.
 - **Insert Menu Sub-Item** Inserts a sub-menu above your selection. Enter the name of the sub-menu as you want it to appear in the user interface, then press Enter.

- **Insert Spacer** Inserts a line above your selection that visually separates groups of commands on the menu.
- **Insert Context Menu** Inserts a menu that appears when you right-click the mouse in the user interface.
- **Insert Context Menu Sub-Item** Inserts a sub-menu to the selected context menu. Enter the name of the sub-menu as you want it to appear in the user interface, then press Enter.

7 Click OK.

Setting custom images for commands on a menu

You can customize the image that appears for each command on a menu.

Because you can configure the menu to display either large or small images and to display either in color or monochrome, create four different images for each command. Create custom images using the following dimensions:


- Small images: 16 x 16 pixels.
- Large images: 32 x 32 pixels.

For best results, images should be 32-bit ARGB format with an alpha transparent background.

Images that do not match these dimensions are stretched or reduced by the program to fit the specified size.

The resulting images may not appear as originally intended.

To customize the images for commands on a menu


- 1 Do one of the following to choose Customize UI (

If High-Contrast Grayscale Buttons is marked on the Display tab in the Options dialog box, the available button images are black and white. Black and white icons


are used for high contrast which is used by color blind users or others with visual impairments. See Section 508 compliance.

- 6 Do one of the following:
 - In Bitmap Images, select the desired image from the display of available images.
 - Click Browse to load a .dll external resource file that contains bitmap resources, a .bmp file, or a .png file. The loaded images must be 16x16 pixels for small icons and 32x32 pixels for large icons.
- 7 When you have finished, click OK.

Setting properties of menu items

You can specify properties for each item on a menu, including the name as it appears on the menu, its , and more.

To customize the properties of menu items

- 1 Do one of the following to choose Customize UI (- On the ribbon, choose Tools > Customize UI (in Manage).
- On the menu, choose Tools > Customize UI.
- Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization area, select either a workspace or a customization file (for example, ICAD), then expand Menus in the list.
- 4 Expand a menu name to show the menu sub-items and commands that display on each menu, then select a menu item.

The properties available for customization vary according to the menu item you select. *The Properties area of the Customize User Interface dialog box displays only those properties that are appropriate for the menu item you select. For example, settings are available for commands, not menu names.*

- 5 In Name, enter the name that you want to display in the user interface.
You can specify an access key by including an ampersand (&) immediately preceding the letter you want to use as the access key. Be sure not to assign the same access key to more than one menu or command within a menu. For example, if you add a command named Quick Line to the Insert menu, including an ampersand immediately preceding the letter Q causes that letter to appear underlined in

the menu. You can then select that command by displaying the menu and pressing the Alt+Q keys.

- 6 In Command Name, enter the name of the command to carry out when the menu item is selected in the user interface.

Command Name may contain a macro (one or more commands and their options), such as:

```
^C^C^CZOOM;E;MSNAPSHOT
```

For details about special characters used in macros, see “Using macros in .cui files” on page 931 in this chapter.

- 7 In Description, type a one-sentence description of the item. The string appears on the status line for menu items.
- 8 In Visibility, click [...] to make your selections for the menu item:
 - **Experience Level** Select the user levels when the item will be visible: Beginner, Intermediate, or Advanced. Items appear only at the experience level you specify. For example, if you choose Advanced, the item does not appear if the user selects the Beginner or Intermediate level using the Tools > Options command; but if you select Intermediate without also selecting Advanced, the item will appear only when you set the experience level to Intermediate. Selecting all three options sets the item to appear at all user levels.
 - **MDI Window** Select the number of drawing windows that must be open for this item to appear. If you want the item to appear only when at least one window is open, select At Least One Open. If you want the item to appear when no windows are open, select No Windows Open. Checking both options means that the item always appears.

- **ActiveX In-Place Activation** Select the type of ActiveX editing with which the item is to appear. If you want the item to appear during embedded server editing, select Server, Embedded. If you want the item to appear during in-place server editing, select Server, In-place. If you want the ribbon item to appear during container editing, select Container.
 - **Document Type** Select whether the item displays when different types of documents (or drawings) are open.
 - **Make Item Hidden** Mark to hide the item in the user interface. The next time you open the Customize User Interface dialog box, this item is marked with a red circle.
- 9 In Entities Context, click [...] to define whether the menu item displays in an entity's context menu. Choose from the following:
- Customize the context menu on a per-entity basis by selecting or deselecting entities. Checking the box next to an entity causes the selected menu item to appear on the context menu for that particular entity.
 - If you do not want the menu item to appear in the context menu when only one entity is selected, select Exclude if Single Entity is Selected.
 - If you do not want the menu item to appear in the context menu when more than one entity is selected, select Exclude if Multiple Entities are Selected.
- 10 If of the menu item can be controlled by a system variable, specify any of the following:
- **Grayed-State Variable** Enter the system variable name and value that controls whether this command displays in the user interface as grayed out. When the specified system variable is off, the button is grayed-out or appears off. For example, the Zoom In command's grayed state variable is VIEWMODE = 1 (perspective view = on).
 - **Checked-State Variable** Enter the system variable name and value that controls whether this command displays in the user interface as pressed in. When the specified system variable is on, the command's button appears pressed in. For example, the Set Center Snap command's pressed state variable is OSMODE = 4 (center snap = on).
 - **Hidden-State Variable** Enter the system variable name that controls whether this command is hidden in the user interface. When the specified system variable is on, the item is hidden. For example, some commands that require an additional license have a hidden-state variable.


Enter the system variable name, an ampersand, then the value, with no spaces. For example: "OSMODE&4".

- 11 When you have finished, click OK.

You can reset the user interface to be the same as when the program was first installed.
Use the Options command, click the Profiles tab, then click Reset.

Renaming menu items

To rename a menu item

- 1 Do one of the following to choose Customize UI (


Copying and pasting menu items

To copy and paste a menu item

- 1 Do one of the following to choose Customize UI (

Deleting menu items

To delete a menu item

- 1 Do one of the following to choose Customize UI (- On the ribbon, choose Tools > Customize UI (in Manage).
- On the menu, choose Tools > Customize UI.
- Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization list, right-click the menu item you want to delete.
- 4 Choose Delete.
- 5 In the Confirmation dialog box, click Yes to delete the menu item.
- 6 Click OK.

Some menu items have sub-items below them.

Deleting a menu item that has sub-items below it in the Menu Tree also deletes all of those sub-items.

Customizing the ribbon

For versions of PTCAD that include a ribbon, several areas contain commands:

- Application button — Located in the upper left corner and contains file-related commands, such as New, Open, Import, Export, and more.
- Quick Access toolbar — Located to the right of the Application button and contains common commands.
- Ribbon — Located below the Application button and runs horizontally at the top of the drawing window. The main ribbon consist of tabs that contain related commands grouped together, for example, on the tabs named Home, Edit, Draw, etc. Each tab has panels that contain sub-categories of commands within a tab, for example Draw, Modify, and Layers on the Home tab.
- Contextual ribbon states — Located temporarily when working with certain entities: multiline text, raster images, and external references.

All of these areas can be customized with the exception of contextual ribbon states, which are built-in to PTCAD.

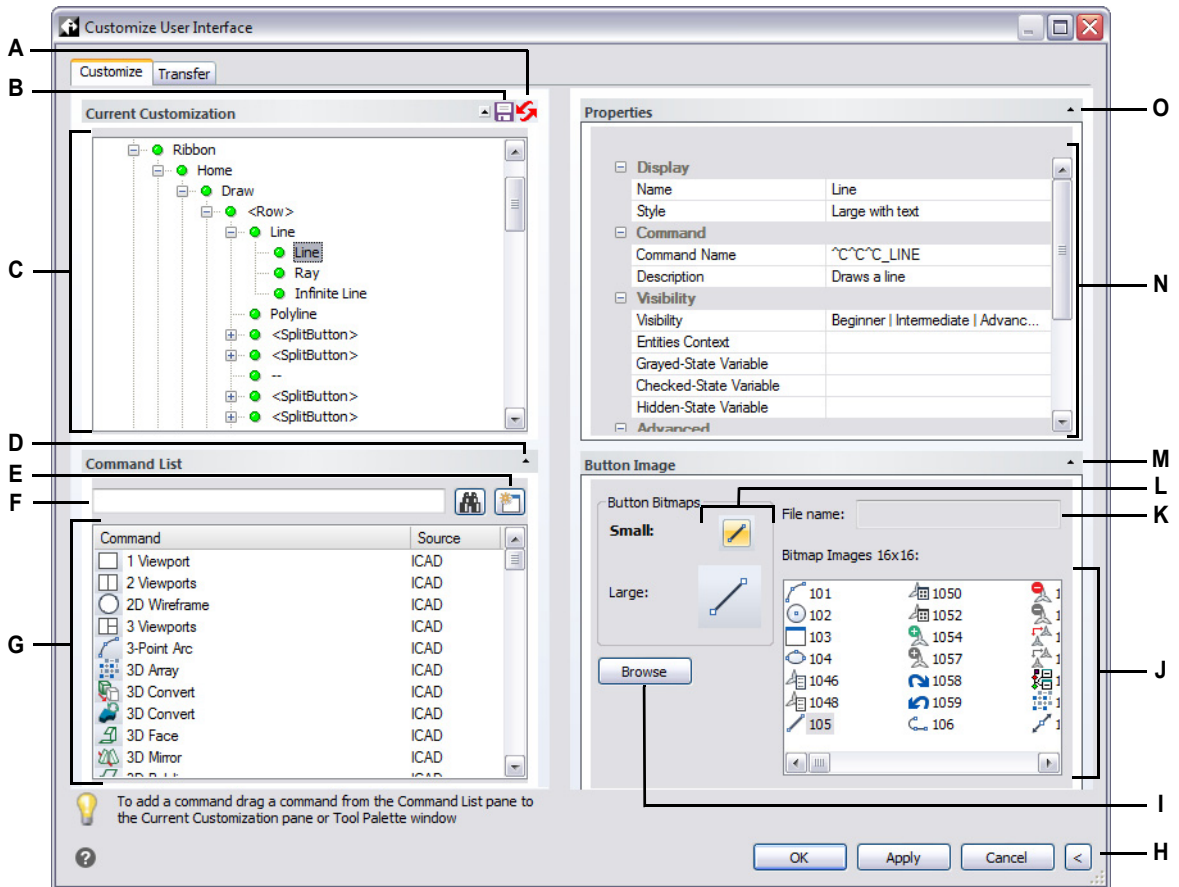
Ribbon settings are saved automatically in a .cui file. The CUI format is the newest user interface format and also supports all legacy user interface elements such as menus, toolbars, tablet, etc. The CUI format is the only menu format the supports the ribbon interface, so if you are creating a menu file that will support the ribbon, use the CUI format.

Creating and customizing ribbon tabs and child panels

You can create a new ribbon tab by inserting a tab at the top level of the Ribbon list. Then you can add panels, or groups, to the new ribbon tab and add commands. You can also modify existing names and commands by adding, deleting, and rearranging them.


Bullet colors indicate whether a command is available at the current experience level.

A green bullet in front of a ribbon item or command indicates that the item is available; a red bullet indicates that the item is not available for you to use at the experience level you have set. To change the experience level for an item, use the property in the Properties area. To change your experience level for the user interface, use the Options command on the Tools menu.



- A** Click to restore the default customization or restore from a backup.
- B** Click to save the customization.
- C** Select a ribbon tab, panel, or a ribbon's command.
- D** Click to show or hide the Command List area.
- E** Click to create a new command.
- F** Type a command name for which to filter the command list.
- G** Displays the list of commands and which source menu group they belong to. Select a command to change its button images in Button Image or its properties in Properties. Drag and drop a command to the Current Customization. Right-click for more options.
- H** Click to hide the Properties and Button Image areas.
- I** Click to select an image file or .dll that contains the image to assign to the currently selected button. If you load an image file or .dll file, you can click Default to reset the images that display in Bitmap Images to the images that were installed with PTCAD.
- J** Select a button image for the selected small or large button.
- K** Displays the full path and filename of the .dll, .bmp, or .png file that was loaded when you clicked Browse.
- L** Select which button you want to customize: small or large.
- M** Click to show or hide the Button Image area.
- N** Displays properties assigned to the selected ribbon item in Current Customization or the selected command in Command List. Enter a new value or click [...] to select more options for each property.
- O** Click to show or hide the Properties area.

To create a new ribbon tab

- 1 Do one of the following to choose Customize UI (**To customize a ribbon tab**

- 1 Do one of the following to choose Customize UI (

- 6 To further customize a ribbon tab, right-click an item located below where you want to add a new item, and choose one of the following:
- **Insert Tab** Inserts a new tab for the ribbon. Enter the name of the tab, then press Enter. The tab is inserted in the list above your selection.
 - **Insert Panel and Insert Child Panel** Inserts an area on the ribbon tab used to group related commands. Enter the name of the panel which displays below the group of commands on the tab, then press Enter. The panel is inserted in the list above your selection, or for child panels, nested in the list under your selection in the list.
 - **Insert Row and Insert Child Row** Inserts a row to a panel. Each panel can have one or more rows, which is a group of buttons aligned horizontally. The row is inserted in the list above your selection, or for child rows, nested in the list below your selection. Note that a set of panel rows can be broken into two parts (visible and collapsed) using the <PanelBreaker> element (right-click and choose Insert Panel Breaker).
 - **Insert Command Button and Insert Child Command Button** Inserts a command button that runs a command when clicked in the user interface. The command button is inserted in the list above your selection, or for child command buttons, nested in the list below your selection.
 - **Insert Menu Button and Insert Child Menu Button** Inserts a menu name that when clicked on the ribbon displays additional command buttons. The menu is inserted in the list above your selection, or for child menu buttons, nested in the list below your selection.
 - **Insert Row Panel and Insert Child Row Panel** Inserts a group of rows that display on a panel. The row panel is inserted in the list above your selection, or for child row panels, nested in the list below your selection.
 - **Insert Control and Insert Child Control** Inserts a standard control for selecting layers, colors, linetypes, lineweights, dimension styles, text styles, and print styles. The control is inserted in the list above your selection, or for child controls, nested in the list below your selection.
 - **Insert Separator and Insert Child Separator** Inserts a line that appears vertically on the ribbon to separate groups of commands. The separator is inserted in the list above your selection, or for child separators, nested in the list below your selection.
 - **Insert Split Button and Insert Child Split Button** Inserts a button that when clicked on the ribbon displays additional command buttons. For example, you might want the Array and 3D Array commands grouped together on a split button. The command button that displays by default on the ribbon is the last one used. The split button is inserted in the list above your selection, or for child split buttons, nested in the list below your selection.
- 7 Click OK.

You can reset the user interface to be the same as when the program was first installed.
Use the Options command, click the Profiles tab, then click Reset.


Creating and customizing quick access toolbars

You can create a new ribbon tab by inserting a tab at the top level of the Ribbon list. Then you can add panels, or groups, to the new ribbon tab and add commands. You can also modify existing names and commands by adding, deleting, and rearranging them.

To create a new quick access toolbar

- 1 Do one of the following to choose Customize UI (

To customize a quick access toolbar


- 1 Do one of the following to choose Customize UI (- On the ribbon, choose Tools > Customize UI (in Manage).
- On the menu, choose Tools > Customize UI.
- Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization area, select either a workspace or a customization file (for example, ICAD), then expand Quick Access Toolbars in the list.
- 4 Expand a quick access toolbar to show the commands that it displays.
- 5 Do one of the following to add a command to the quick access toolbar:
 - From the Command List, drag a command to where you want it located in the Current Customization list.
 - Right-click a command in the Quick Access Toolbar list and choose Insert Command Button. Type the name, and then press Enter. Then in Properties, enter the command name for the item in Command Name.
- 6 Click OK.

You can reset the user interface to be the same as when the program was first installed.
 Use the *Options* command, click the *Profiles* tab, then click *Reset*.


Creating and customizing application buttons

You can create a new application button and modify an existing one. You can also modify existing names and commands by adding, deleting, and rearranging them.

To create a new application button

- 1 Do one of the following to choose Customize UI (- On the ribbon, choose Tools > Customize UI (in Manage).
- On the menu, choose Tools > Customize UI.
- Type *cui* and then press Enter.
- 2 In the Current Customization area, select either a workspace or a customization file (for example, ICAD), then expand Application Buttons in the list.
- 3 Right-click an application button in the list, then choose Insert Application Button. The new application button will be inserted above your selection in the list.
- 4 Type the application button name, and then press Enter
- 5 Add a command to the application button (see next). To see the new application button, you must add a command to it.
- 6 Click OK.

To customize an application button

- 1 Do one of the following to choose Customize UI ():
 - On the ribbon, choose Tools > Customize UI (in Manage).
 - On the menu, choose Tools > Customize UI.
 - Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization area, select either a workspace or a customization file (for example, ICAD), then expand Application Buttons in the list.
- 4 Expand an application button to show the commands that it displays.
- 5 From the Command List, drag a command to where you want it located in the Current Customization list.
- 6 To further customize an application button, right-click an item located below where you want to add a new item, and choose one of the following:
 - **Insert Command Button** Inserts an item that runs a command when clicked in the user interface. The command is inserted in the list above your selection.
 - **Insert Menu Button** Inserts a menu item that opens a sub-menu when clicked in the user interface. The menu is inserted in the list above your selection.
 - **Insert Separator** Inserts a line in the user interface that visually separates groups of commands.
 - **Insert Child Command Button** Inserts an item that runs a command when clicked in the user interface. The command is nested under your selection in the list. This option displays for menu buttons (not command buttons).
 - **Insert Child Separator** Inserts a line in the user interface that visually separates groups of commands. The separator is nested under your selection in the list. This option displays for menu buttons (not command buttons).
- 7 Click OK.

Setting custom images for commands on the ribbon

You can customize the image that appears for each command on the ribbon.

Because you can configure the ribbon to display either large or small images and to display either in color or monochrome, create four different images for each command. Create custom images using the following dimensions:


- Small images: 16 x 16 pixels.
- Large images: 32 x 32 pixels.

For best results, images should be 32-bit ARGB format with an alpha transparent background.

Images that do not match these dimensions are stretched or reduced by the program to fit the specified size.

The resulting images may not appear as originally intended.

To customize the images for commands on the ribbon


- 1 Do one of the following to choose Customize UI (- On the ribbon, choose Tools > Customize UI (in Manage).
- On the menu, choose Tools > Customize UI.
- Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization area, select either a workspace or a customization file (for example, ICAD), then expand Ribbon in the list.
- 4 Expand a ribbon tab name and panel names to show the commands that display on each ribbon tab, then select a command.
- 5 In the Button Image area, in Button Bitmaps, click the button image you want to change: Small or Large.

If High-Contrast Grayscale Buttons is marked on the Display tab in the Options dialog box, the available button images are black and white. Black and white icons are used for high contrast which is used by color blind users or others with visual impairments. See Section 508 compliance.
- 6 Do one of the following:
 - In Bitmap Images, select the desired image from the display of available images.
 - Click Browse to load a .dll external resource file that contains bitmap resources, a .bmp file, or a .png file. The loaded images must be 16x16 pixels for small icons and 32x32 pixels for large icons.
- 7 When you have finished, click OK.

Setting properties of ribbon items

You can specify properties for each item on the ribbon, including the name as it appears on the ribbon, its corresponding command, and more.

To customize the properties of ribbon items

- 1 Do one of the following to choose Customize UI ():
 - On the ribbon, choose Tools > Customize UI (in Manage).
 - On the menu, choose Tools > Customize UI.
 - Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization area, select either a workspace or a customization file (for example, ICAD), then expand Ribbon in the list.
- 4 Expand a ribbon tab name and panel name to show the commands that display on each ribbon tab, then select a ribbon item.

The properties available for customization vary according to the ribbon item you select.

The Properties area of the Customize User Interface dialog box displays only those properties that are appropriate for the ribbon item you select. For example, settings are available for commands, not ribbon panel names.

- 5 In Name, enter the name that you want to display in the ribbon.
- 6 In Command Name, enter the name of the command to carry out when the ribbon item is selected in the user interface.

Command Name may contain a macro (one or more commands and their options), such as:

```
^C^C^CZOOM;E;MSNAPSHOT
```

For details about special characters used in macros, see “Using macros in .cui files” on page 931 in this chapter.
- 7 In Description, type a one-sentence description of the item. The string appears on the status line for ribbon items.
- 8 In Visibility, click [...] to make your selections for the ribbon item:
 - **Experience Level** Select the user levels when the item will be visible: Beginner, Intermediate, or Advanced. Items appear only at the experience level you specify. For example, if you choose Advanced, the item does not appear if the user selects the Beginner or Intermediate level using the Tools > Options

command; but if you select Intermediate without also selecting Advanced, the item will appear only when you set the experience level to Intermediate. Selecting all three options sets the item to appear at all user levels.


- **MDI Window** Select the number of drawing windows that must be open for this item to appear. If you want the item to appear only when at least one window is open, select *At Least One Open*. If you want the item to appear when no windows are open, select *No Windows Open*. Checking both options means that the item always appears.
- **ActiveX In-Place Activation** Select the type of ActiveX editing with which the item is to appear. If you want the item to appear during embedded server editing, select *Server, Embedded*. If you want the item to appear during in-place server editing, select *Server, In-place*. If you want the item to appear during container editing, select *Container*.
- **Document Type** Select whether the item displays when different types of documents (or drawings) are open.
- **Make Item Hidden** Mark to hide the item in the user interface. The next time you open the *Customize User Interface* dialog box, this item is marked with a red circle.

- 9 In Entities Content, click [...] to define whether the ribbon item displays in an entity's context menu. Choose from the following:
- Customize the context menu on a per-entity basis by selecting or deselecting entities. Checking the box next to an entity causes the selected ribbon item to appear on the context menu for that particular entity.
 - If you do not want the ribbon item to appear in the context menu when only one entity is selected, select Exclude if Single Entity is Selected.
 - If you do not want the ribbon item to appear in the context menu when more than one entity is selected, select Exclude if Multiple Entities are Selected.
- 10 If the ribbon item can be controlled by a system variable, specify any of the following:
- **Grayed-State Variable** Enter the system variable name and value that controls whether this command displays in the user interface as grayed out. When the specified system variable is off, the button is grayed-out or appears off. For example, the Zoom In command's grayed state variable is VIEWMODE = 1 (perspective view = on).
 - **Checked-State Variable** Enter the system variable name and value that controls whether this command displays in the user interface as pressed in. When the specified system variable is on, the command's button appears pressed in. For example, the Set Center Snap command's pressed state variable is OSMODE = 4 (center snap = on).
 - **Hidden-State Variable** Enter the system variable name that controls whether this command is hidden in the user interface. When the specified system variable is on, the item is hidden. For example, some commands that require an additional license have a hidden-state variable.
- Enter the system variable name, an ampersand, then the value, with no spaces. For example: "OSMODE&4".
- 11 When you have finished, click OK.

You can reset the user interface to be the same as when the program was first installed.
Use the Options command, click the Profiles tab, then click Reset.

Renaming ribbon items

To rename a ribbon item

- 1 Do one of the following to choose Customize UI (


Copying and pasting ribbon items

To copy and paste a ribbon item

- 1 Do one of the following to choose Customize UI (

Deleting ribbon items

To delete a ribbon item

- 1 Do one of the following to choose Customize UI (

Some ribbon items have sub-items below them.

Deleting a ribbon item that has sub-items below it also deletes all of those sub-items.

Customizing toolbars

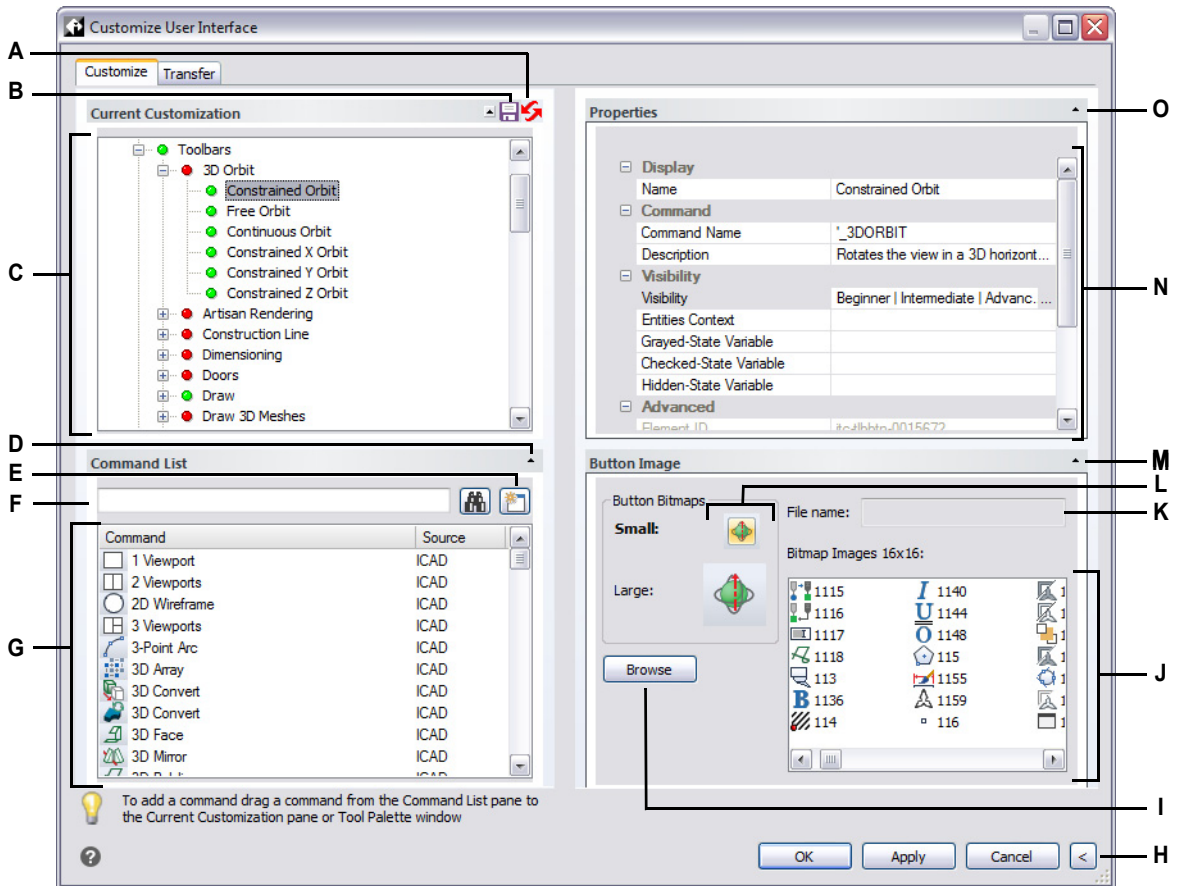
PTCAD provides toolbars so that you can access frequently used commands.

When a toolbar is turned on, it is always available, or on top, and in the same location so it is easy to find and use. However, toolbars occupy drawing display space and reduce the area available for working in drawings.

You can customize these toolbars by adding or removing tools or by rearranging the organization of tools. You can also create custom toolbars.


Creating and customizing toolbars

You can create a new toolbar using the Customize UI command, then you can add tools to the new toolbar. You can also add, delete, or modify tools on any existing toolbar.




- A** Click to restore the default customization or restore from a backup.
- B** Click to save the customization.
- C** Select a toolbar or a toolbar command.
- D** Click to show or hide the Command List area.
- E** Click to create a new command.
- F** Type a command name for which to filter the command list.
- G** Displays the list of commands and which source menu group they belong to. Select a command to change its button images in Button Image or its properties in Properties. Drag and drop a command to the Current Customization. Right-click for more options.
- H** Click to hide the Properties and Button Image areas.
- I** Click to select an image file or .dll that contains the image to assign to the currently selected button. If you load an image file or .dll file, you can click Default to reset the images that display in Bitmap Images to the images that were installed with PTCAD.
- J** Select a button image for the selected small or large button.
- K** Displays the full path and filename of the .dll, .bmp, or .png file that was loaded when you clicked Browse.
- L** Select which button you want to customize: small or large.
- M** Click to show or hide the Button Image area.
- N** Displays properties assigned to the selected toolbar item in Current Customization or the selected command in Command List. Enter a new value or click [...] to select more options for each property.
- O** Click to show or hide the Properties area.

To create a new toolbar

- 1 Do one of the following to choose Customize UI ():
 - On the ribbon, choose Tools > Customize UI (in Manage).
 - On the menu, choose Tools > Customize UI.
 - Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization area, select either a workspace or a customization file (for example, ICAD), then expand Toolbars in the list. The toolbars are shown in the list with a plus sign (+) next to each name.
- 4 Right-click a toolbar name located below the location of the new toolbar.
- 5 Choose Insert Toolbar.
- 6 Type a name for the new toolbar, and then press Enter.
- 7 Add a command to the toolbar (see next). To see the new toolbar in the user interface, you must add a command to it.
- 8 Click OK.

To add a command to a toolbar

- 1 Do one of the following to choose Customize UI ():
 - On the ribbon, choose Tools > Customize UI (in Manage).
 - On the menu, choose Tools > Customize UI.
 - Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization area, select either a workspace or a customization file (for example, ICAD), then expand Toolbars in the list.
- 4 Expand a toolbar name to show the commands that display on each toolbar.
- 5 From the Command List, drag a command to where you want it located in the Current Customization list.
- 6 To further customize a toolbar, right-click an item located below where you want to add a new item, and choose one of the following:
 - **Insert Button** Inserts a toolbar button that runs a command when clicked in the user interface. The toolbar button is inserted in the list above your selection.
 - **Insert Sub-Item** Inserts a toolbar button if you right-clicked a toolbar name in the list. If you right-clicked an item in the list located on a toolbar, inserts a toolbar button nested in the list under your selection, and the selection becomes the parent command on a flyout.

Toolbar flyouts are a way to organize related tools and conserve space on

toolbars. A flyout is indicated by a small triangle in the lower right corner of a tool. When you click a flyout tool, a set of additional tools displays. The flyout tool you select then becomes the default tool on the toolbar.

- **Insert Spacer** Inserts a line that appears vertically on the toolbar to separate groups of commands. The separator is inserted in the list above your selection.

7 Click OK.

You can reset the user interface to be the same as when the program was first installed.
Use the Options command, click the Profiles tab, then click Reset.

Setting custom images for commands on a toolbar

You can customize the image that appears for each command on a toolbar.

Because you can configure toolbars to display either large or small tools and to display tools either in color or monochrome, create four different images for each tool. Create custom images using the following dimensions:


- Small images: 16 x 16 pixels.
- Large images: 32 x 32 pixels.

For best results, images should be 32-bit ARGB format with an alpha transparent background.

Images that do not match these dimensions are stretched or reduced by the program to fit the specified size.

The resulting tools may not appear as originally intended.

To customize the images for commands on a toolbar

- 1 Do one of the following to choose Customize UI (

If High-Contrast Grayscale Buttons is marked on the Display tab in the Options dialog box, the available button images are black and white. Black and white icons


are used for high contrast which is used by color blind users or others with visual impairments. See Section 508 compliance.

- 6 Do one of the following:
 - In Bitmap Images, select the desired image from the display of available images.
 - Click Browse to load a .dll external resource file that contains bitmap resources, a .bmp file, or a .png file. The loaded images must be 16x16 pixels for small icons and 32x32 pixels for large icons.
- 7 When you have finished, click OK.

Setting properties of toolbar items

You can specify properties for each item on a toolbar, including the name as it appears on a tooltip, a tool's , and more.

To customize the properties of toolbar items

- 1 Do one of the following to choose Customize UI (- On the ribbon, choose Tools > Customize UI (in Manage).
- On the menu, choose Tools > Customize UI.
- Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization area, select either a workspace or a customization file (for example, ICAD), then expand Toolbars in the list.
- 4 Expand a toolbar name to show the commands that display on each toolbar, then select a toolbar item.

The properties available for customization vary according to the toolbar item you select.

The Properties area of the Customize User Interface dialog box displays only those properties that are appropriate for the toolbar item you select. For example, settings are available for commands, not toolbar names.

- 5 In Name, enter the name that you want to display when you hover over a toolbar.
- 6 In Command Name, enter the name of the command to carry out when the toolbar item is selected in the user interface.

Command Name may contain a macro (one or more commands and their options), such as:

```
^C^C^CZOOM;E;MSNAPSHOT
```

For details about special characters used in macros, see “Using macros in .cui files” on page 931 in this chapter.

- 7 In Description, type a one-sentence description of the item. The string appears on the status line for toolbar items.
- 8 In , click [...] to make your selections for the toolbar item:
 - **Experience Level** Select the user levels when the item will be visible: Beginner, Intermediate, and/or Advanced. Items appear only at the experience level you specify. For example, if you choose Advanced, the item does not appear if the user selects the Beginner or Intermediate level using the Tools > Options command; but if you select Intermediate without also selecting Advanced, the item will appear only when you set the experience level to Intermediate. Selecting all three options sets the item to appear at all user levels.
 - **MDI Window** Select the number of drawing windows that must be open for this item to appear. If you want the item to appear only when at least one window is open, select At Least One Open. If you want the item to appear when no windows are open, select No Windows Open. Checking both options means that the item always appears.

- **ActiveX In-Place Activation** Select the type of ActiveX editing with which the item is to appear. If you want the item to appear during embedded server editing, select Server, Embedded. If you want the item to appear during in-place server editing, select Server, In-place. If you want the item to appear during container editing, select Container.
 - **Document Type** Select whether the item displays when different types of documents (or drawings) are open.
 - **Make Item Hidden** Mark to hide the item in the user interface. The next time you open the Customize User Interface dialog box, this item is marked with a red circle.
- 9 In Entities Content, click [...] to define whether the toolbar item displays in an entity's context menu. Choose from the following:
- Customize the context menu on a per-entity basis by selecting or deselecting entities. Checking the box next to an entity causes the selected toolbar item to appear on the context menu for that particular entity.
 - If you do not want the toolbar item to appear in the context menu when only one entity is selected, select Exclude if Single Entity is Selected.
 - If you do not want the toolbar item to appear in the context menu when more than one entity is selected, select Exclude if Multiple Entities are Selected.
- 10 If of the toolbar item can be controlled by a system variable, specify any of the following:
- **Grayed-State Variable** Enter the system variable name and value that controls whether this command displays in the user interface as grayed out. When the specified system variable is off, the button is grayed-out or appears off. For example, the Zoom In command's grayed state variable is VIEWMODE = 1 (perspective view = on).
 - **Checked-State Variable** Enter the system variable name and value that controls whether this command displays in the user interface as pressed in. When the specified system variable is on, the command's button appears pressed in. For example, the Set Center Snap command's pressed state variable is OSMODE = 4 (center snap = on).
 - **Hidden-State Variable** Enter the system variable name that controls whether this command is hidden in the user interface. When the specified system variable is on, the item is hidden. For example, some commands that require an additional license have a hidden-state variable.


Enter the system variable name, an ampersand, then the value, with no spaces. For example: "OSMODE&4".

- 11 When you have finished, click OK.

You can reset the user interface to be the same as when the program was first installed.
Use the Options command, click the Profiles tab, then click Reset.

Renaming toolbar items

To rename a toolbar item

- 1 Do one of the following to choose Customize UI (

You can also rename toolbars using the Toolbars command.

Choose View > Toolbars to open the Select Toolbars dialog box. Select the toolbar you want to rename, then click Rename and type the new name.


Copying and pasting toolbar items

To copy and paste a toolbar item

- 1 Do one of the following to choose Customize UI (

Deleting toolbar items

To delete a toolbar item

- 1 Do one of the following to choose Customize UI (- On the ribbon, choose Tools > Customize UI (in Manage).
- On the menu, choose Tools > Customize UI.
- Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization list, right-click the toolbar item you want to delete.
- 4 Choose Delete.
- 5 In the Confirmation dialog box, click Yes to delete the toolbar item.
- 6 Click OK.

Customizing the keyboard, aliases, double-click actions, and image tile menus

Customizing the keyboard

PTCAD provides keyboard shortcuts so you can access frequently used commands. You can customize these shortcuts and add new shortcuts using the Custom-ize UI command.

Creating and customizing keyboard shortcuts

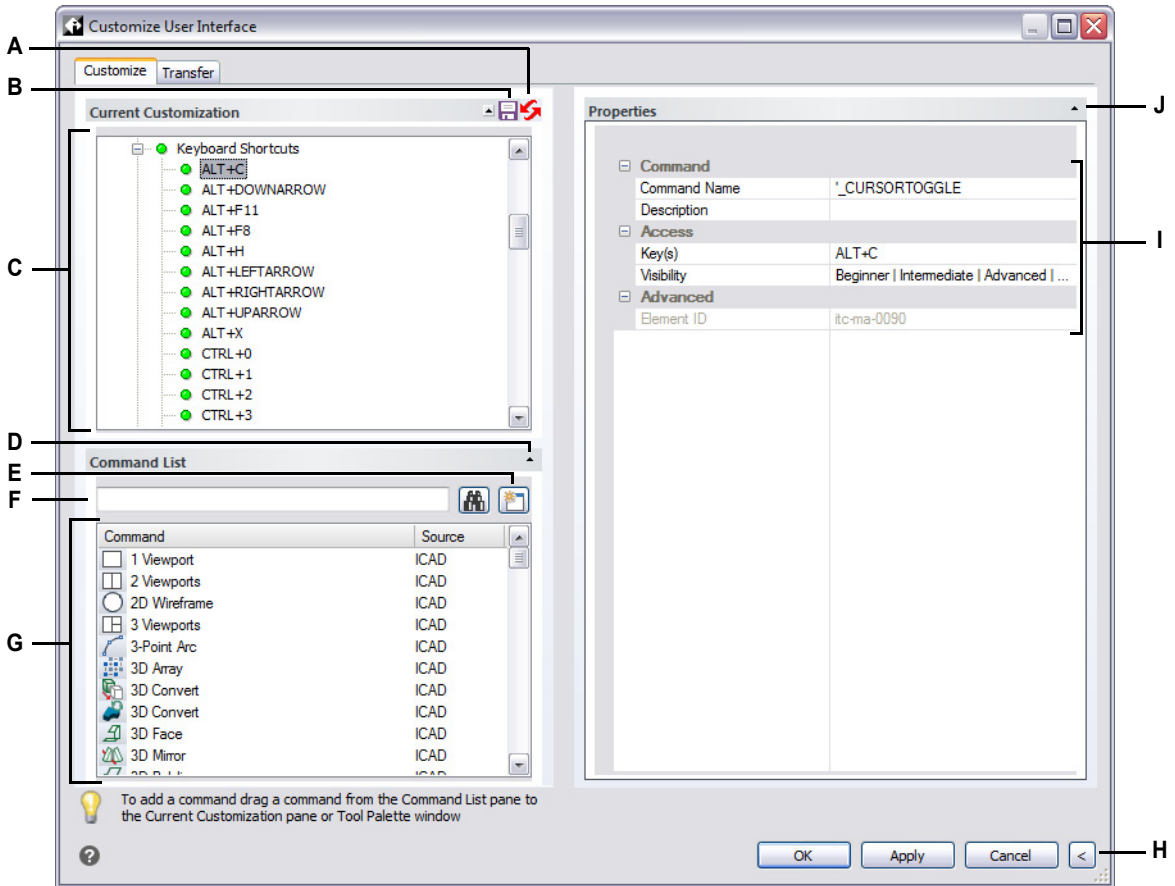
You can assign macros to special keys and certain combinations of keys to create a keyboard shortcut. A macro consists of one or more commands that are displayed on the status bar as follows:

- A single command, such as QSAVE.
- A command with options, such as ARC;\A;\.
- More than one command, such as ^C^C^CZOOM;E;QSAVE;QPRINT.

Keyboard shortcuts are more powerful than aliases. An alias lets you abbreviate a single command name; a keyboard shortcut contains one or more macros. To activate a macro, you press the shortcut key; you do not need to press Enter as you do with an alias. A shortcut consists of the following keys:


- The function and the cursor control keys, as well as alphanumeric keys pressed in combination with the Ctrl, Alt, and Shift keys. The Shift key must be used in conjunction with the Ctrl, Alt, and/or function keys.
- Alphanumeric keys are those labeled A through Z and 1 through 0.

- The cursor keys are the up, down, left, and right arrows and the Page Up, Page Down, Home, End, Insert, and Delete keys.
- The function keys are those labeled F1 through F12.



- A** Click to restore the default customization or restore from a backup.
- B** Click to save the customization.
- C** Select a keyboard shortcut.
- D** Click to show or hide the Command List area.
- E** Click to create a new command.
- F** Type a command name for which to filter the command list.
- G** Displays the list of commands and which source menu group they belong to. Select a command to change its button images in Button Image or its properties in Properties. Right-click for more options.
- H** Click to hide the Properties and Button Image areas.
- I** Displays properties assigned to the selected keyboard shortcut in Current Customization or the selected command in Command List. Enter a new value or click [...] if available to select more options for each property.
- J** Click to show or hide the Properties area.

To create a keyboard shortcut


- 1 Do one of the following to choose Customize UI (- On the ribbon, choose Tools > Customize UI (in Manage).
- On the menu, choose Tools > Customize UI.
- Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization area, select either a workspace or a customization file (for example, ICAD), then expand Keyboard Shortcuts in the list.
- 4 Right-click a shortcut name located above where you want to add the new keyboard shortcut in the list.
- 5 Choose Insert Accelerator.
- 6 In the Keyboard Shortcut Keys dialog box, press the keyboard combination you want to add, for example, press Alt+A.
- 7 Click OK.
- 8 In Command Name, enter the name of the command to carry out when the keyboard combination is pressed in the user interface.
 Command Name may contain a macro (one or more commands and their options), such as:
`^C^C^CZOOM;E;MSNAPSHOT`
 For details about special characters used in macros, see “Using macros in .cui files” on page 931 in this chapter.
- 9 In Description, type a one-sentence description of the item.
- 10 To save your changes and close the dialog box, click OK.

You can reset the user interface to be the same as when the program was first installed.
Use the Options command, click the Profiles tab, then click Reset.

Setting properties of keyboard shortcuts

You can specify properties for each keyboard shortcut, including the associated command, description, and key combination.

To customize the properties of keyboard shortcuts


- 1 Do one of the following to choose Customize UI (- On the ribbon, choose Tools > Customize UI (in Manage).
- On the menu, choose Tools > Customize UI.
- Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization area, select either a workspace or a customization file (for example, ICAD), then expand Keyboard Shortcuts in the list.

- 4 Select a keyboard shortcut.
- 5 In Command Name, enter the name of the command to carry out when the keyboard shortcut is pressed in the user interface.
Command Name may contain a macro (one or more commands and their options), such as:
`^C^C^CZOOM;E;MSNAPSHOT`
For details about special characters used in macros, see “Using macros in .cui files” on page 931 in this chapter.
- 6 In Description, type a one-sentence description of the item.
- 7 To specify a new key combination, do the following:
 - In Key(s), press [...].
 - Press a new key combination.
 - Click OK.
- 8 When you have finished, click OK.

You can reset the user interface to be the same as when the program was first installed.
Use the Options command, click the Profiles tab, then click Reset.


Copying and pasting keyboard shortcuts

To copy and paste a keyboard shortcut

- 1 Do one of the following to choose Customize UI (- On the ribbon, choose Tools > Customize UI (in Manage).
- On the menu, choose Tools > Customize UI.
- Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization list, right-click the keyboard shortcut you want to copy.
- 4 Choose Copy.
- 5 Right-click the keyboard shortcut located above where you want to locate the copy.
- 6 Choose Paste.
- 7 Click OK.

Deleting keyboard shortcuts

To delete a keyboard shortcut item

- 1 Do one of the following to choose Customize UI ():
 - On the ribbon, choose Tools > Customize UI (in Manage).
 - On the menu, choose Tools > Customize UI.
 - Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization list, right-click the keyboard shortcut you want to delete.
- 4 Choose Delete.
- 5 In the Confirmation dialog box, click Yes to delete the keyboard shortcut.
- 6 Click OK.

Customizing aliases

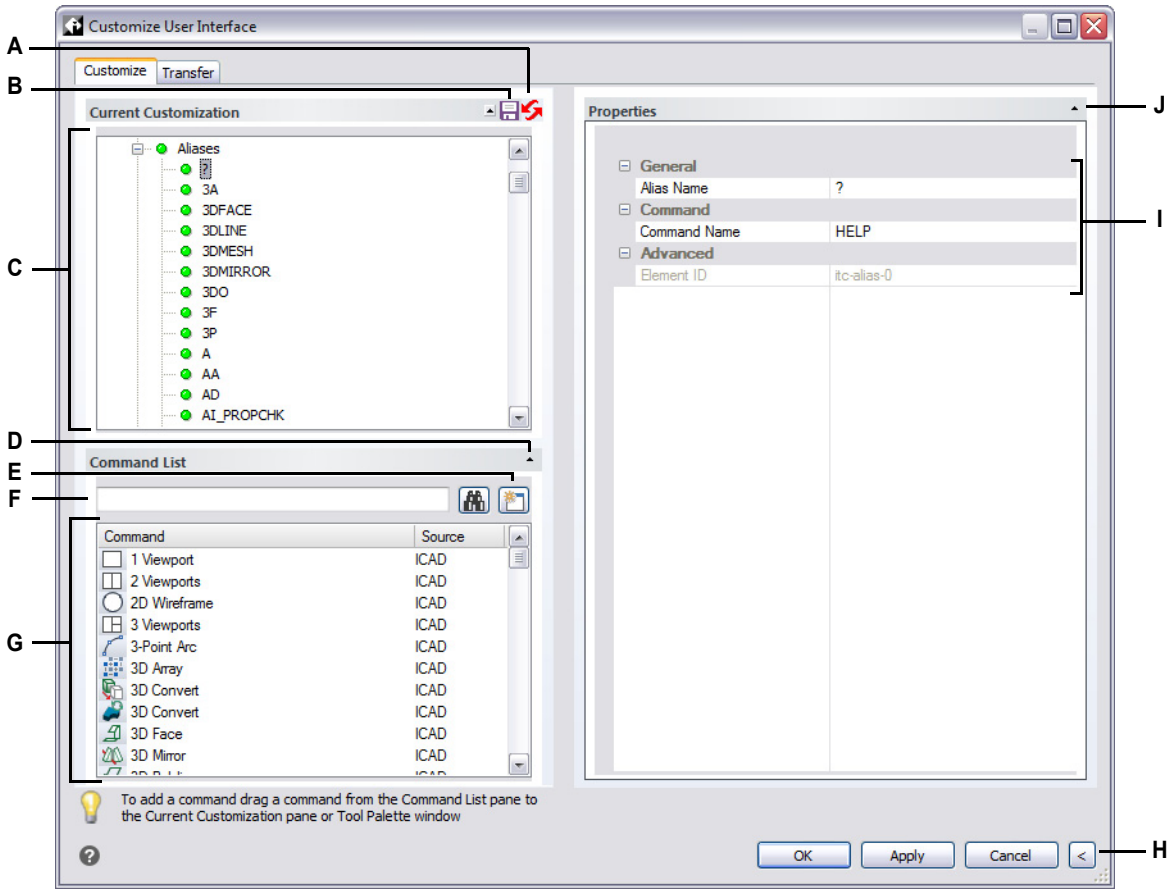
PTCAD provides aliases for many commands. You can use aliases to issue frequently used commands by entering one or two letters rather than the entire command name.

The program also uses aliases to maintain command-name compatibility with AutoCAD. You can use the same aliases and keyboard shortcuts used by AutoCAD. In addition, PTCAD has enhanced several AutoCAD commands. For example, PTCAD added two useful options to the *rectangle* command: you can draw a rectangle as a square, and you can rotate a rectangle at an angle.

You can customize aliases and add new aliases using the Customize UI command, as described in this section. Another method is to use the Customize Aliases command; for more information about it, click [?] after typing *aliasedit*.

Creating aliases


To create a new command alias, you first define the alias and then assign it one of the available PTCAD commands.



- A** Click to restore the default customization or restore from a backup.
- B** Click to save the customization.
- C** Select an alias.
- D** Click to show or hide the Command List area.
- E** Click to create a new command.
- F** Type a command name for which to filter the command list.

- G** Displays the list of commands and which source menu group they belong to. Select a command to change its button images in Button Image or its properties in Properties. Right-click for more options.
- H** Click to hide the Properties and Button Image areas.
- I** Displays properties assigned to the selected alias in Current Customization or the selected command in Command List. Enter a new value or click [...] if available to select more options for each property.
- J** Click to show or hide the Properties area.

To create a new alias


- 1 Do one of the following to choose Customize UI (- On the ribbon, choose Tools > Customize UI (in Manage).
- On the menu, choose Tools > Customize UI.
- Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization area, select either a workspace or a customization file (for example, ICAD), then expand Aliases in the list.
- 4 Right-click an alias name located below where you want to add the new alias in the list.
- 5 Choose Insert Alias.
- 6 Type the new alias, then press Enter.
- 7 In Command Name, enter the name of the command to carry out when the alias is entered in the user interface.
 Command Name may contain a macro (one or more commands and their options), such as:
`^C^C^CZOOM;E;MSNAPSHOT`
 For details about special characters used in macros, see “Using macros in .cui files” on page 931 in this chapter.
- 8 In Description, type a one-sentence description of the item.
- 9 To save your changes and close the dialog box, click OK.

You can reset the user interface to be the same as when the program was first installed.
Use the Options command, click the Profiles tab, then click Reset.

Setting properties of aliases

You can specify properties for each alias, including the associated command and description.

To customize the properties of aliases

- 1 Do one of the following to choose Customize UI (- On the ribbon, choose Tools > Customize UI (in Manage).
- On the menu, choose Tools > Customize UI.
- Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization area, select either a workspace or a customization file (for example, ICAD), then expand Aliases in the list.

- 4 Select an alias.
- 5 In Name, enter a new name for the alias.
- 6 In Command Name, enter the name of the command to carry out when the alias name is entered in the user interface.

Command Name may contain a macro (one or more commands and their options), such as:

```
^C^C^CZOOM;E;MSNAPSHOT
```


For details about special characters used in macros, see “Using macros in .cui files” on page 931 in this chapter.

- 7 In Description, type a one-sentence description of the item.
- 8 When you have finished, click OK.

You can reset the user interface to be the same as when the program was first installed.
Use the Options command, click the Profiles tab, then click Reset.


Copying and pasting an alias item

To copy and paste an alias item

- 1 Do one of the following to choose Customize UI (- On the ribbon, choose Tools > Customize UI (in Manage).
- On the menu, choose Tools > Customize UI.
- Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization list, right-click the alias you want to copy.
- 4 Choose Copy.
- 5 Right-click the alias located above where you want to locate the copy.
- 6 Choose Paste.
- 7 Click OK.

Deleting an alias item

To delete an existing alias item

- 1 Do one of the following to choose Customize UI (- On the ribbon, choose Tools > Customize UI (in Manage).
- On the menu, choose Tools > Customize UI.
- Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization list, right-click the alias you want to delete.
- 4 Choose Delete.
- 5 In the Confirmation dialog box, click Yes to delete the alias.
- 6 Click OK.

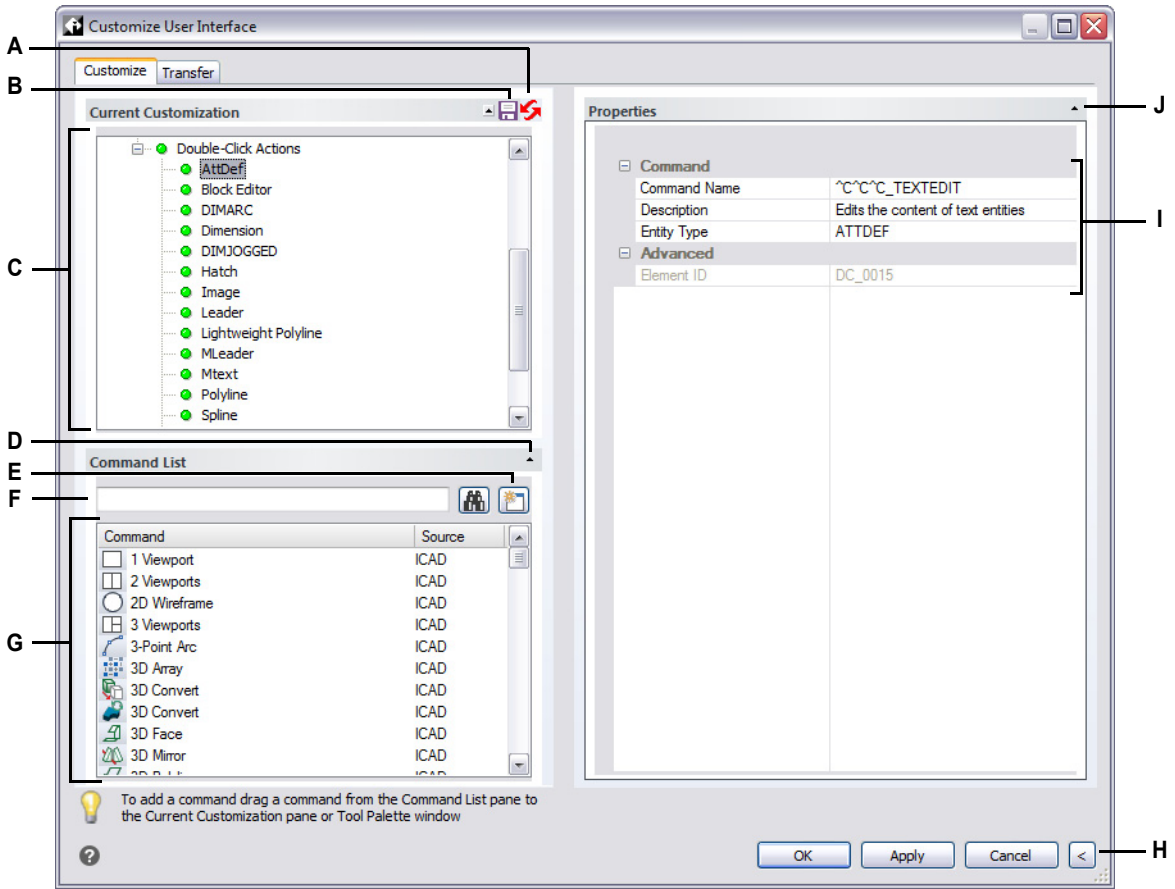
Customizing double-click actions

Double-click actions allow you to start a command when double-clicking a specific entity. For example, because of a double-click action, you can double-click a block to start the Block Editor command.

If an entity does not have a double-click action assigned to it, double-clicking the entity opens the Quick Properties pane.


Creating double-click actions

To create a new double-click action, you first name it and then assign it an entity type and one of the available PTCAD commands.



- A** Click to restore the default customization or restore from a backup.
- B** Click to save the customization.
- C** Select a double-click action.
- D** Click to show or hide the Command List area.
- E** Click to create a new command.
- F** Type a command name for which to filter the command list.
- G** Displays the list of commands and which source menu group they belong to. Select a command to change its button images in Button Image or its properties in Properties. Right-click for more options.
- H** Click to hide the Properties and Button Image areas.
- I** Displays properties assigned to the selected double-click action in Current Customization or the selected command in Command List. Enter a new value or click [...] if available to select more options for each property.
- J** Click to show or hide the Properties area.

To create a new double-click action


- 1 Do one of the following to choose Customize UI ():
 - On the ribbon, choose Tools > Customize UI (in Manage).
 - On the menu, choose Tools > Customize UI.
 - Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization area, select either a workspace or a customization file (for example, ICAD), then expand Double-Click Actions in the list.
- 4 Right-click a double-click action located above where you want to add the new double-click action in the list.
- 5 Choose New.
- 6 Type the name of the new double-click action, then press Enter.
- 7 In Command Name, enter the name of the command to carry out when double-clicking in the drawing. For example, MTEDIT starts the Multiline Text Editor. Command Name may contain a macro (one or more commands and their options), such as:
`^C^C^CZOOM;E;MSNAPSHOT`
 For details about special characters used in macros, see “Using macros in .cui files” on page 931 in this chapter.
- 8 In Description, optionally type a one-sentence description of the double-click action.
- 9 In Entity Type, type the name of the entity type that when double-clicked, starts the command. For example, if you enter MTEXT, the action starts the command when you double-click multiline text.
- 10 To save your changes and close the dialog box, click OK.

You can reset the user interface to be the same as when the program was first installed.
Use the Options command, click the Profiles tab, then click Reset.

Setting properties of double-click actions

You can specify properties for each double-click action, including the associated command and description.


To customize the properties of double-click actions

- 1 Do one of the following to choose Customize UI ():
 - On the ribbon, choose Tools > Customize UI (in Manage).
 - On the menu, choose Tools > Customize UI.
 - Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization area, select either a workspace or a customization file (for example, ICAD), then expand Double-Click Actions in the list.
- 4 Select a double-click action.
- 5 In Name, enter a new name for the double-click action.
- 6 In Command Name, enter the name of the command to carry out when double-clicking in the drawing. For example, MTEDIT starts the Multiline Text Editor. Command Name may contain a macro (one or more commands and their options), such as:
`^C^CZOOM;E;MSNAPSHOT`
 For details about special characters used in macros, see “Using macros in .cui files” on page 931 in this chapter.
- 7 In Description, optionally type a one-sentence description of the double-click action.
- 8 In DXF name, type the name of the entity type that when double-clicked, starts the command. For example, if you enter MTEXT, the action starts the command when you double-click multiline text.
- 9 When you have finished, click OK.

You can reset the user interface to be the same as when the program was first installed.
Use the Options command, click the Profiles tab, then click Reset.


Copying and pasting a double-click action

To copy and paste a double-click action

- 1 Do one of the following to choose Customize UI (- On the ribbon, choose Tools > Customize UI (in Manage).
- On the menu, choose Tools > Customize UI.
- Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization list, right-click the double-click action you want to copy.
- 4 Choose Copy.
- 5 Right-click the double-click action located above where you want to locate the copy.
- 6 Choose Paste.
- 7 Click OK.

Deleting a double-click action

To delete a double-click action

- 1 Do one of the following to choose Customize UI (- On the ribbon, choose Tools > Customize UI (in Manage).
- On the menu, choose Tools > Customize UI.
- Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization list, right-click the double-click action you want to delete.
- 4 Choose Delete.
- 5 In the Confirmation dialog box, click Yes to delete the double-click action.
- 6 Click OK,


Customizing image tile menus

Image tile menus are used to create custom dialog boxes that contain lists and images of frequently-used blocks or commands. A legacy user interface, image tile menus are used most commonly by third-party programs to display the contents of block libraries and to make it easy to insert blocks in drawings. They can be added to any existing menu, toolbar, or ribbon tab (if supported by your program version).


Image tile menus display images from slide (.sld) files that are created with the Make Snapshot command. For details about creating .sld files, see “Creating snapshots” on page 780.

To create a new user interface for inserting blocks or running commands, it is recommended to use tool palettes instead. For more details, see “Starting commands using tool palettes” on page 35. For more details, see Starting commands using tool palettes.

To create an image tile menu

- 1 Do one of the following to choose Customize UI ():
 - On the ribbon, choose Tools > Customize UI (in Manage).
 - On the menu, choose Tools > Customize UI.
 - Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization area, select a customization file (for example, ICAD), then right-click Image Tile Menus in the list.
- 4 Choose New Image Tile Menu.
- 5 Type the name of the new image tile menu, then press Enter.
- 6 Do the following for each command you want to add to the image tile menu:
 - Drag a command from the Command List to the new image tile menu.
 - In the Properties area for the command, in Slide Label, enter the name of the slide (.sld) file, without the file extension, that you want to associate with the command. For details about creating .sld files, see “Creating snapshots” on page 780.
 - Place the .sld file in the program’s installation folder.
- 7 Click OK.

To add an image tile menu to the user interface



- 1 Do one of the following to choose Customize UI ():
 - On the ribbon, choose Tools > Customize UI (in Manage).
 - On the menu, choose Tools > Customize UI.
 - Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization area, select the image tile menu you want to add to the user interface.
- 4 In the Properties area, copy the alias that was assigned automatically to the image tile menu, for example IMG3001.
- 5 In the Current Customization area, select a customization file (for example, ICAD), then right-click where you want to add the image tile menu, and create a new item. For example, to add the image tile menu to a menu, right-click the command located below where you want to add the image tile menu, then choose Insert Menu Item.
- 6 Enter the name as you want it to appear in the user interface, then press Enter.
- 7 In the Properties area, enter the following for the Command Name, where IMG3001 is the alias of the image tile menu:
`$I=IMG3001 $I=*`
- 8 Click OK.

Customizing the Quick Properties pane

The Quick Properties pane opens and closes dynamically, allowing you to access entity-specific properties when you need them without crowding available screen space as with the Properties pane. It displays automatically when double-clicking an entity, if the entity does not have a double-click action assigned to it, and it optionally displays when using any selection method and a command is not active. For details about using the Quick Properties pane and changing its behavior, see “Modifying entity properties by using the Quick Properties pane” on page 408.

The Quick Properties pane generally doesn’t display all properties of an entity, and the available properties is easily customizable for each entity type.

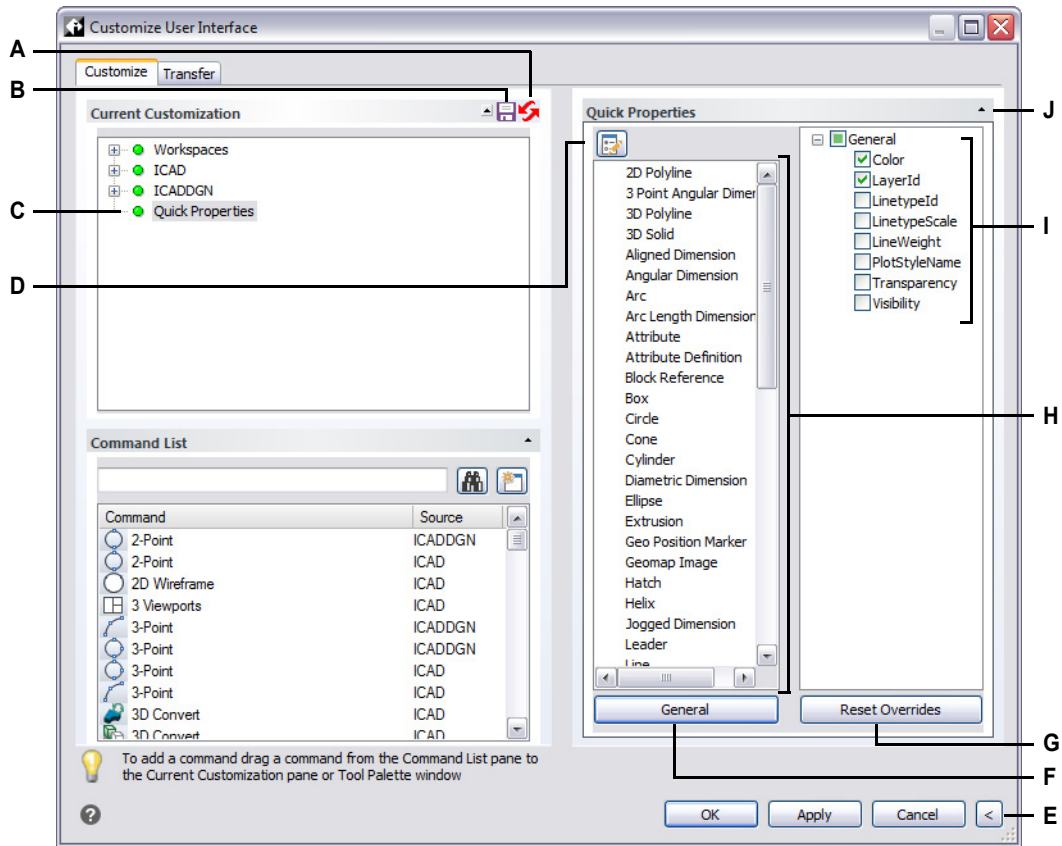
To customize the contents of the Quick Properties pane

- 1 Do one of the following to choose Customize UI (- On the ribbon, choose Tools > Customize UI (in Manage).
- On the menu, choose Tools > Customize UI.
- Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization area, select Quick Properties.
- 4 To specify the entity types for which the Quick Properties pane displays, do the following in the Quick Properties area:
 - Click(). The Edit Entity Type List dialog box displays.
 - Mark the entity types for which you want the Quick Properties pane to display. Your selections using the Drawing Settings command affect whether the Quick Properties pane displays for unmarked entity types. For more details, see “Modifying entity properties by using the Quick Properties pane” on page 408.
 - Click OK.
- 5 To specify the properties that display in the Quick Properties pane, do the following for each entity type:
 - Select an entity type.
 - Mark the properties you want included in the Quick Properties pane.

You can change the properties available to choose from in the list.

Click General to temporarily display only the general properties for the selected entity type, then optionally click Reset Overrides to restore the display of hidden properties. Right-click the list of properties and choose Show Advanced Properties to show all available properties.

- 6 Click OK.



- A** Click to restore the default customization or restore from a backup.
- B** Click to save the current customization.
- C** Select to customize the contents of the Quick Properties pane.
- D** Click to specify the entity types for which the Quick Properties pane displays.
- E** Click to hide the Quick Properties area.
- F** Click to display only the general properties for the selected entity type.
- G** Click to restore the display of hidden properties. Available only if General was previously clicked.
- H** Select the entity type for which you want to customize quick properties.
- I** Select each property you want included in the Quick Properties pane for the selected entity type.
- J** Click to show or hide the Quick Properties area.

Working with user interface customization files

Many users customize areas of the PTCAD main window such as the menus and toolbars using the Customize UI (CUI) command. These settings are saved in a `.cui/.cuix` file, which is an XML format file that contains settings for the following:

- Ribbon
- Workspaces
- Menus
- Toolbars
- Keyboard accelerators
- Aliases
- Double-click actions

You can save and load `.cui/.cuix` files, restore them, copy settings between `.cui` files, and even manually edit a `.cui/.cuix` file.

PTCAD works with other legacy file types too that are used to customize the user interface. You can also load AutoCAD menus and toolbars (`*.mnu`, `*.mns`), PTCAD menus and toolbars (`*.icm`), PTCAD aliases (`*.ica`), AutoCAD aliases (`*.pgp`), and PTCAD keyboard shortcuts (`*.ick`).

Tool palettes are part of the user interface and they can be customized too, but they are not saved in customization files. To customize tool palettes, see “Customizing tool palettes” on page 864 in this chapter.


Saving customization files

PTCAD automatically saves any customizations you make to the following:

- Ribbon
- Workspaces
- Menus
- Toolbars
- Keyboard accelerators
- Aliases
- Double-click actions

Your changes are saved automatically in a `.cui/.cuix` file. If desired, you can save the settings to a new `.cui/.cuix` file.

To save customization settings to a new customization file

- 1 Do one of the following to choose Customize UI (- On the ribbon, choose Tools > Customize UI (in Manage).
- On the menu, choose Tools > Customize UI.
- Type *cui* and then press Enter.
- 2 Click the Transfer tab.
- 3 On the right, click Save As.
- 4 Enter a name for the .cui/.cuix file.
- 5 Click Save.
- 6 Click OK.


Loading customization files

You can replace the current customization file with a different one or you can extend the current customization file with additional items like custom menus. The program loads full customizations (*.cui, *.cuix), AutoCAD menus and toolbars (*.mnu, *.mns), PTCAD menus and toolbars (*.icm), PTCAD aliases (*.ica), AutoCAD aliases (*.pgp), and PTCAD keyboard shortcuts (*.ick).



Loading a menu, alias, or keyboard shortcut file replaces only those items in the user interface.

You can also restore customization settings by loading the default customization file from when the program was installed or by loading a backup customization file.

To load a customization file

- 1 Do one of the following to choose Customize UI (- On the ribbon, choose Tools > Customize UI (in Manage).
- On the menu, choose Tools > Customize UI.
- Type *cui* and then press Enter.
- 2 Click the Transfer tab.
- 3 On the left, click Open.
- 4 From the Files Of Type list, choose the file type you want to load.
- 5 Select the desired file.
- 6 Click Open.
- 7 Click OK.



To restore the default customization file

- 1 Do one of the following to choose Customize UI (- On the ribbon, choose Tools > Customize UI (in Manage).
- On the menu, choose Tools > Customize UI.
- Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization area, click Restore From (.
- 4 Choose Default Customization.
- 5 Click Yes to restore the default customization file or No to cancel.
- 6 Click OK.

You can also reset user interface changes from the command prompt.

To reverse all user interface changes made with the Customize User Interface command and reset the user interface customization to the default, type `resetui` to start the Reset User Interface command.


To restore a backup customization file

- 1 Do one of the following to choose Customize UI (- On the ribbon, choose Tools > Customize UI (in Manage).
- On the menu, choose Tools > Customize UI.
- Type *cui* and then press Enter.
- 2 Click the Customize tab.
- 3 In the Current Customization area, click Restore From (.
- 4 Choose Daily, Weekly, Monthly, or Annual Backup.
- 5 Click Yes to restore the backup customization file or No to cancel.
- 6 Click OK.

Creating a new .cui file and copying settings

Use the Customize UI command to quickly create a new .cui/.cuix file and copy settings from one customization file to another. You choose which settings to copy, from a single command to all settings in a file.

To create a new .cui/.cuix file and copy settings

- 1 Do one of the following to choose Customize UI (- On the ribbon, choose Tools > Customize UI (in Manage).
- On the menu, choose Tools > Customize UI.
- Type *cui* and then press Enter.
- 2 Click the Transfer tab.
- 3 On the right, click New.
- 4 On the left, click Open and select the customization file that contains the settings you want to copy.
- 5 Drag and drop settings between the two files.
- 6 Click OK when done.

Customizing a .cui file using an editor

A .cui file is an XML file format, which can be edited using any ASCII or Unicode text editor. However, it is best to edit .cui files using an XML editor such as Visual Studio or Notepad++. If you are developing with PTCAD, you already have Visual Studio.

Use caution when manually editing a .cui file and use the following tips about the format of the file:

- Content of .cui/.cuix files is case-sensitive. For example, the following is correct; no errors will be shown but ID attribute will be ignored:

```
<ToolTip id="123">Help</ToolTip>
```

The following is also correct; ID will be equal to 123:

```
<ToolTip ID="123">Help</ToolTip>
```

- Content of .cui files contains special symbols, similar to an .xml file: & (ampersand), ' (quote), and more. Replace these special symbols with corresponding escape sequence or use XML CDATA structure instead. For example, the following is not correct:

```
<ToolTip id="123">&Help</ToolTip>
```

Correct:

```
<ToolTip ID="123">&amp;Help</ToolTip>
```

Not correct:

```
<Command>'_HELP</Command>
```

Correct:


```
<Command><! [CDATA['_HELP']]></Command>
```

- Properties of several entity types can be described in both node-style and attribute-style. For example, the following two toolbar buttons are the same:

```
<ToolbarButton UID="unique-id" IsSeparator="true"
="CUI_LEV_ALL,CUI_MDI_OPN,CUI_OLE_SEM"
Entity="RCM_ENT_ACS,RCM_ENT_MUL"/>
```

```
<ToolbarButton UID="unique-id">
<IsSeparator>true</IsSeparator>
<>CUI_LEV_ALL,CUI_MDI_OPN,CUI_OLE_SEM</>
<Entity>RCM_ENT_ACS,RCM_ENT_MUL</Entity>
</ToolbarButton>
```

To customize a .cui/.cuix file using an editor

- 1 Do one of the following to choose Customize UI (

There are many editors, but if you are developing with PTCAD, you already have Visual Studio, which is a good XML editor for .cui files.

Understanding versioning of a .cui file

Each .cui file has a node that controls versioning:

```
<FileVersion MajorVersion="1" MinorVersion="2" UserVersion="0"
IncrementalVersion="91"/>
```

- **IncrementalVersion** Must be incremented with any change in the Icad.cui file.
- **UserVersion** Incremented automatically when a user customizes PTCAD. In the default Icad.cui file, this must be always equal to 0 and never edited manually.
- **MinorVersion** Must be incremented with a change or creation of a versioned entity (PopupMenu, Toolbar, MenuMacro->Macro, ...).
- **MajorVersion** Must be incremented with the following:
 - Change of Icad.cui file schema (for example, added/deleted sections or added new type of entities).
 - Deletion of a versioned entity.
 - Change of UID attribute of an existing versioned entity or change of Name attribute in case of a Workspace entity.

If MajorVersion is incremented, the user's .cui file will be replaced with an updated .cui file.

The .cui file also contains versioned entities: Workspace, MenuMacro->Macro, PopMenu, Toolbar, AppButton, QuickAccessToolbar, RibbonPanelSource, and RibbonTabSource. Each versioned entity also has a node:

```
<ModifiedRev MinorVersion="2"/>
```

After a versioned entity is modified or created, FileVersion->MinorVersion must be incremented, and then the result must be assigned to ModifiedRev->MinorVersion of the modified entity.

Understanding the format of a .cui file

A .cui file can contain settings for the following areas of the PTCAD main window:

- Ribbon
- Workspaces
- Menus
- Toolbars
- Keyboard accelerators
- Command aliases

CUI file format

Item	Values (Default Value)	Required?	Description
FileVersion			
<i>MajorVersion</i>	4 byte integer	Yes	Major version of the .cui file. For details, see “Understanding versioning of a .cui file” on page 921 in this chapter.
<i>MinorVersion</i>	4 byte integer	Yes	Minor version of the .cui file. For details, see “Understanding versioning of a .cui file” on page 921 in this chapter.
<i>IncrementalVersion</i>	4 byte integer	Yes	Incremental version of the .cui file. For details, see “Understanding versioning of a .cui file” on page 921 in this chapter.
<i>UserVersion</i>	4 byte integer	Yes	User version of the .cui file. For details, see “Understanding versioning of a .cui file” on page 921 in this chapter.
AppButton			
			The application button.
<i>UID</i>	Text	Yes	ID of an AppButton. Unique among all UIDs.
<i>LargeImage</i>	Text("0")	No	Image resource ID of AppButton’s icon or path to local image file.
<i>LargeImageHighContrast</i>	Text("0")	No	Image resource ID of AppButton’s high contrast icon or path to local file.
<i>Name</i>	Text("")	No	Name or tooltip of an AppButton.
AppButtonItem			
			Regular command button or separator line.
<i>UID</i>	Text	Yes	ID of an AppButtonItem. Unique among all UIDs.
<i>MenuMacroID</i>	Text	Yes/No	ID of referenced MenuMacro. Not required in case of separator button. Required in all other cases.
<i>IsSeparator</i>	true/false(false)	No	Determines whether an item is a separator line or regular command button.
AppButtonPopItem			
			Sub-menu that contains a set of AppButtonItem items.
<i>UID</i>	Text	Yes	ID of an AppButtonPopItem. Unique among all UIDs.
<i>LargeImage</i>	Text("0")	No	Image resource ID of AppButtonPopItem’s icon or path to local image file.
<i>LargeImageHighContrast</i>	Text("0")	No	Image resource ID of AppButtonPopItem’s high contrast icon or path to local image file.

CUI file format

Item	Values (Default Value)	Required?	Description
<i>Name</i>	Text("")	No	Name or tooltip of an AppButtonPopItem.
QuickAccessToolbar			The Quick Access Toolbar.
<i>UID</i>	Text	Yes	ID of a Quick Access toolbar. Unique among all UIDs.
QuickAccessToolbarStandardItem			Regular command button.
<i>UID</i>	Text	Yes	ID of a QuickAccessToolbarStandardItem. Unique among all UIDs.
<i>MenuMacroID</i>	Text	Yes	ID of referenced MenuMacro.
Workspace			Consists of the following sections: WSPopRoot, WSToolbarRoot, and WSRibbonRoot.
<i>DefaultWorkspace</i>	true/false (false)	No	Whether a workspace is selected by default.
<i>Name</i>	Text	Yes	Name of workspace.
WSPop			In-workspace declaration of a pop-menu.
<i>Display</i>	0/1 (0)	No	Whether to display (1) or not to display (0) pop-menu in the MenuBar.
<i>pUID</i>	Text	Yes	UID of a referenced pop-menu. Unique among all UIDs.
WSToolbar			In-workspace declaration of a toolbar.
<i>Display</i>	0/1 (0)	No	Whether to display (1) or not to display (0) a toolbar.
<i>pUID</i>	Text	Yes	UID of a referenced toolbar. Unique among all UIDs.
<i>ToolbarOrient</i>	top/left/bottom/right (float)	No	Toolbar orientation.
<i>rows</i>	4 byte integer (1)	No	Number of button rows for a toolbar.
<i>xval</i>	4 byte integer (0)	No	X-coordinate of a toolbar (relative in case of docked toolbar).
<i>yval</i>	4 byte integer (0)	No	Y-coordinate of a toolbar (relative in case of docked toolbar).
WSRibbonTabSourceReference			In-workspace declaration of a ribbon tab. Consists of a set of WSRibbonPanelSourceReference items.
<i>Show</i>	true/false (false)	No	Whether to show or not to show a ribbon tab.
<i>TabId</i>	Text	Yes	UID of a referenced ribbon tab.
WSRibbonPanelSourceReference			In-workspace declaration of a ribbon panel.
<i>Show</i>	true/false (false)	No	Whether to show or not to show a ribbon panel.
<i>PanelId</i>	Text	Yes	UID of a referenced ribbon panel.
RibbonPanelSource			Ribbon panel definition. Consists of a set of RibbonRow items.
<i>UID</i>	Text	Yes	ID of a RibbonPanelSource. Unique among all UIDs.
<i>Name</i>	Text	Yes	Name of a RibbonPanelSource.
RibbonSplitButton			Ribbon button with sub-items. Consists of RibbonCommandButton items.
<i>UID</i>	Text	Yes	ID of a RibbonSplitButton. Unique among all UIDs.

CUI file format

Item	Values (Default Value)	Required?	Description
<i>ButtonStyle</i>	LargeWithText/ SmallWithoutText/ SmallWithText (SmallWithoutText)	No	Style of a button.
RibbonCommandButton			Regular command button.
<i>UID</i>	Text	Yes	ID of a RibbonCommandButton. Unique among all UIDs.
<i>MenuMacroID</i>	Text	Yes	ID of referenced MenuMacro.
<i>ButtonStyle</i>	LargeWithText/ SmallWithoutText/ SmallWithText (SmallWithoutText)	No	Style of a button.
RibbonSeparator			Ribbon Separator element.
<i>UID</i>	Text	Yes	ID of a RibbonSeparator. Unique among all UIDs.
<i>SeparatorStyle</i>	Line/Spacer (Spacer)	No	Separator style can be either a line or a spacer.
RibbonControl			Ribbon control. Can be of one of these predefined types: color, linetype, linewidth, layer, print style, text style, or dimension style.
<i>UID</i>	rbnctrl-layer/ rbnctrl-color/ rbnctrl-linetype/ rbnctrl-lineweight/ rbnctrl-printstyle/ rbnctrl-textstyle/ rbnctrl-dimstyle	Yes	Unique among all UIDs. Any other UID is not allowed.
RibbonTabSource			Ribbon tab definition. Consists of a set of RibbonPanelSourceReference items.
<i>UID</i>	Text	Yes	ID of a RibbonTabSource. Unique among all UIDs.
<i>Name</i>	Text	Yes	Name of a RibbonTabSource.
MenuAccelerator			
<i>UID</i>	Text	Yes	ID of a MenuAccelerator. Unique among all UIDs.
MenuAccelerator.Accelerator			
<i>KeyCombo.AccelChar</i>	Text ("")	Yes	Virtual key code. Use letters, digits ("A", "B", "C", ... , "1", "2", "3", ... , "0"), functional key codes ("F1", "F2", "F3", ..., "F16"), Num-pad key codes ("NUM1", "NUM2", "NUM3", ..., "NUM0", "NUM*", "NUM+", "NUM-", "NUMDEL", "NUM/"), some other key codes ("BACKSPACE", "PAUSE", "PAGEUP", "PAGEDOWN", "LEFTARROW", "UPARROW", "RIGHTARROW", "DOWNARROW", "INS", "DEL", "END", "HOME")
<i>KeyCombo.accKeys</i>	Text ("")	Yes if previous is letter or digit	Accelerator modifier. Following combinations are allowed: "CTRL", "ALT", "SHIFT", "CTRL+ALT", "ALT+CTRL", "SHIFT+CTRL", "CTRL+SHIFT", "ALT+SHIFT", "SHIFT+ALT", "CTRL+ALT+SHIFT", "CTRL+SHIFT+ALT", "ALT+SHIFT+CTRL", "ALT+CTRL+SHIFT", "SHIFT+CTRL+ALT", "SHIFT+ALT+CTRL"
<i>MenuItem.MacroRef.MenuMacroID</i>	Text ("")	No	ID of a macro to be bound to this accelerator

CUI file format

Item	Values (Default Value)	Required?	Description
MenuAlias			
<i>UID</i>	Text	Yes	ID of a MenuAlias. Unique among all UIDs.
<i>Name</i>	Text	Yes	Name of an alias.
<i>Command</i>	Text	Yes	Command that is to be called for the alias.
MenuGroup			
<i>Name</i>	Text ("ICAD")	No	Name of a menu group.
MenuMacro			
<i>UID</i>	Text	Yes	ID of a MenuMacro. Unique among all UIDs.
<i>Macro.Name.ID</i>	4 byte integer (0)	No	Text resource id of macro name. Used to extract localized text.
<i>Macro.Name</i>	Text ("")	No	Default macro name.
<i>Macro.ToolTip.ID</i>	4 byte integer	No	Text resource id of macro tooltip. Used to extract localized text.
<i>Macro.ToolTip</i>	Text	Yes	Default macro tooltip.
<i>Macro.HelpString.ID</i>	4 byte integer (0)	No	Text resource id of macro help string. Used to extract localized text.
<i>Macro.HelpString</i>	Text	Yes	Default macro help string.
<i>Macro.Command</i>	Text	Yes	Command of a macro.
<i>Macro.SmallImage</i>	4 byte integer (0)	No	Image resource id of small icon or path to local image file.
<i>Macro.LargeImage</i>	4 byte integer (0)	No	Image resource id of large icon or path to local image file.
<i>Macro.SmallImageHighContrast</i>	4 byte integer (0)	No	Image resource id of small high contrast icon or path to local image file.
<i>Macro.LargeImageHighContrast</i>	4 byte integer (0)	No	Image resource id of large high contrast icon or path to local image file.
<i>Macro.AcadToolbarConversion</i>	Text ("")	No	Used to extract MenuMacro icon.

CUI file format

Item	Values (Default Value)	Required?	Description																																														
<i>Macro.</i>	Text (CUI_DEFAULT)	No	<p>Determines the of MenuMacro, constructed as a comma-separated concatenation of the following values, calculated with a logical OR applied to all values:</p> <table border="1"> <thead> <tr> <th data-bbox="743 334 896 360">Value</th> <th data-bbox="901 334 1313 360">Description</th> </tr> </thead> <tbody> <tr> <td>CUI_DOC_DWG</td> <td>.dwg document</td> </tr> <tr> <td>CUI_DOC_DGN</td> <td>.dgn document</td> </tr> <tr> <td>CUI_DOC_ALL</td> <td>All types of documents</td> </tr> <tr> <td>CUI_DOC_STP</td> <td>Start Page document</td> </tr> <tr> <td>CUI_LEV_BEG</td> <td>User level: Beginner</td> </tr> <tr> <td>CUI_LEV_INT</td> <td>User level: Intermediate</td> </tr> <tr> <td>CUI_LEV_EXP</td> <td>User level: Expert</td> </tr> <tr> <td>CUI_LEV_ALL</td> <td>User level: Any</td> </tr> <tr> <td>CUI_MDI_OPN</td> <td>MDI Window: At least one open</td> </tr> <tr> <td>CUI_MDI_CLS</td> <td>MDI Window: No MDI windows</td> </tr> <tr> <td>CUI_MDI_ALL</td> <td>MDI Window: Either MDI state</td> </tr> <tr> <td>CUI_OLE_SEM</td> <td>OLE: Server, embedded</td> </tr> <tr> <td>CUI_OLE_SIP</td> <td>OLE: Server, in-place</td> </tr> <tr> <td>CUI_OLE_CLI</td> <td>OLE: Client</td> </tr> <tr> <td>CUI_OLE_ALL</td> <td>OLE: Any OLE state</td> </tr> <tr> <td>CUI_VIS_ALL</td> <td>CUI_LEV_ALL,CUI_MDI_ALL,CUI_OLE_ALL</td> </tr> <tr> <td>CUI_SPL_RCM</td> <td>Context (right-click) menu only</td> </tr> <tr> <td>CUI_SPL_HIDE</td> <td>Hide this menu or button</td> </tr> <tr> <td>CUI_SPL_OSNAP</td> <td>Temporary osnap mode (shift right click when command is active)</td> </tr> <tr> <td>CUI_SPL_CTRL</td> <td>Button is a control]</td> </tr> <tr> <td>CUI_SPL_NOENT</td> <td>Ignore all entity flags]</td> </tr> <tr> <td>CUI_DEFAULT</td> <td>Default :</td> </tr> </tbody> </table> <p>CUI_LEV_ALL,CUI_MDI_OPN,CUI_MDI_CLS, CUI_OLE_CLI</p>	Value	Description	CUI_DOC_DWG	.dwg document	CUI_DOC_DGN	.dgn document	CUI_DOC_ALL	All types of documents	CUI_DOC_STP	Start Page document	CUI_LEV_BEG	User level: Beginner	CUI_LEV_INT	User level: Intermediate	CUI_LEV_EXP	User level: Expert	CUI_LEV_ALL	User level: Any	CUI_MDI_OPN	MDI Window: At least one open	CUI_MDI_CLS	MDI Window: No MDI windows	CUI_MDI_ALL	MDI Window: Either MDI state	CUI_OLE_SEM	OLE: Server, embedded	CUI_OLE_SIP	OLE: Server, in-place	CUI_OLE_CLI	OLE: Client	CUI_OLE_ALL	OLE: Any OLE state	CUI_VIS_ALL	CUI_LEV_ALL,CUI_MDI_ALL,CUI_OLE_ALL	CUI_SPL_RCM	Context (right-click) menu only	CUI_SPL_HIDE	Hide this menu or button	CUI_SPL_OSNAP	Temporary osnap mode (shift right click when command is active)	CUI_SPL_CTRL	Button is a control]	CUI_SPL_NOENT	Ignore all entity flags]	CUI_DEFAULT	Default :
Value	Description																																																
CUI_DOC_DWG	.dwg document																																																
CUI_DOC_DGN	.dgn document																																																
CUI_DOC_ALL	All types of documents																																																
CUI_DOC_STP	Start Page document																																																
CUI_LEV_BEG	User level: Beginner																																																
CUI_LEV_INT	User level: Intermediate																																																
CUI_LEV_EXP	User level: Expert																																																
CUI_LEV_ALL	User level: Any																																																
CUI_MDI_OPN	MDI Window: At least one open																																																
CUI_MDI_CLS	MDI Window: No MDI windows																																																
CUI_MDI_ALL	MDI Window: Either MDI state																																																
CUI_OLE_SEM	OLE: Server, embedded																																																
CUI_OLE_SIP	OLE: Server, in-place																																																
CUI_OLE_CLI	OLE: Client																																																
CUI_OLE_ALL	OLE: Any OLE state																																																
CUI_VIS_ALL	CUI_LEV_ALL,CUI_MDI_ALL,CUI_OLE_ALL																																																
CUI_SPL_RCM	Context (right-click) menu only																																																
CUI_SPL_HIDE	Hide this menu or button																																																
CUI_SPL_OSNAP	Temporary osnap mode (shift right click when command is active)																																																
CUI_SPL_CTRL	Button is a control]																																																
CUI_SPL_NOENT	Ignore all entity flags]																																																
CUI_DEFAULT	Default :																																																

CUI file format

Item	Values (Default Value)	Required?	Description																																																																														
<i>Macro.Entity</i>	Text (RCM_ENT_ALL)	No	Determines the entity of Menu.Macro, constructed as a comma-separated concatenation of the following values, calculated with a logical OR applied to all values:																																																																														
<p>Note that this item controls of right-click menus, which are designed to display only when certain entities are in the right-click selection set. This item is only used when Macro. includes CUI.SPL.RCM.</p>																																																																																	
			<table border="1"> <thead> <tr> <th>Entity Value</th> <th>Description</th> </tr> </thead> <tbody> <tr><td>RCM_ENT_PNT</td><td>Point</td></tr> <tr><td>RCM_ENT_LIN</td><td>Line</td></tr> <tr><td>RCM_ENT_RAY</td><td>Ray</td></tr> <tr><td>RCM_ENT_XLN</td><td>XLine</td></tr> <tr><td>RCM_ENT_ARC</td><td>Arc</td></tr> <tr><td>RCM_ENT_CIR</td><td>Circle</td></tr> <tr><td>RCM_ENT_ELL</td><td>Ellipse</td></tr> <tr><td>RCM_ENT_SHP</td><td>Shape</td></tr> <tr><td>RCM_ENT_TRC</td><td>Trace</td></tr> <tr><td>RCM_ENT_SLD</td><td>Solid</td></tr> <tr><td>RCM_ENT_3DF</td><td>3D Face</td></tr> <tr><td>RCM_ENT_3DS</td><td>3D Solid</td></tr> <tr><td>RCM_ENT_2DP</td><td>2D Polyline</td></tr> <tr><td>RCM_ENT_3DP</td><td>3D Polyline</td></tr> <tr><td>RCM_ENT_PFM</td><td>Polyface Mesh</td></tr> <tr><td>RCM_ENT_3DM</td><td>3D Mesh</td></tr> <tr><td>RCM_ENT_PLN</td><td>All types of polylines: RCM_ENT_2DP,RCM_ENT_3DP, RCM_ENT_PFM,RCM_ENT_3DM</td></tr> <tr><td>RCM_ENT_TXT</td><td>Text</td></tr> <tr><td>RCM_ENT_INS</td><td>Insert</td></tr> <tr><td>RCM_ENT_ATD</td><td>AttDef</td></tr> <tr><td>RCM_ENT_DIM</td><td>Dimension</td></tr> <tr><td>RCM_ENT_LDR</td><td>Leader</td></tr> <tr><td>RCM_ENT_TOL</td><td>Tolerance</td></tr> <tr><td>RCM_ENT_SPL</td><td>Spline</td></tr> <tr><td>RCM_ENT_MTX</td><td>Mtext</td></tr> <tr><td>RCM_ENT_MLN</td><td>Mline</td></tr> <tr><td>RCM_ENT_GRP</td><td>Group</td></tr> <tr><td>RCM_ENT_IMG</td><td>Image</td></tr> <tr><td>RCM_ENT_VPT</td><td>Viewport</td></tr> <tr><td>RCM_ENT_HAT</td><td>Hatch</td></tr> <tr><td>RCM_ENT_RGN</td><td>Region</td></tr> <tr><td>RCM_ENT_BDY</td><td>Body</td></tr> <tr><td>RCM_ENT_XRF</td><td>Xreferences</td></tr> <tr><td>RCM_ENT_GMI</td><td>Geographic Map Image</td></tr> <tr><td>RCM_ENT_ACS</td><td>All ACIS entities: RCM_ENT_3DS, RCM_ENT_RGN,RCM_ENT_BDY</td></tr> <tr><td>RCM_ENT_ALL</td><td>Any entity</td></tr> <tr><td>RCM_ENT_SGL</td><td>Hide this item if a single entity is selected</td></tr> <tr><td>RCM_ENT_MUL</td><td>Hide this item if multiple entities are selected</td></tr> </tbody> </table>	Entity Value	Description	RCM_ENT_PNT	Point	RCM_ENT_LIN	Line	RCM_ENT_RAY	Ray	RCM_ENT_XLN	XLine	RCM_ENT_ARC	Arc	RCM_ENT_CIR	Circle	RCM_ENT_ELL	Ellipse	RCM_ENT_SHP	Shape	RCM_ENT_TRC	Trace	RCM_ENT_SLD	Solid	RCM_ENT_3DF	3D Face	RCM_ENT_3DS	3D Solid	RCM_ENT_2DP	2D Polyline	RCM_ENT_3DP	3D Polyline	RCM_ENT_PFM	Polyface Mesh	RCM_ENT_3DM	3D Mesh	RCM_ENT_PLN	All types of polylines: RCM_ENT_2DP,RCM_ENT_3DP, RCM_ENT_PFM,RCM_ENT_3DM	RCM_ENT_TXT	Text	RCM_ENT_INS	Insert	RCM_ENT_ATD	AttDef	RCM_ENT_DIM	Dimension	RCM_ENT_LDR	Leader	RCM_ENT_TOL	Tolerance	RCM_ENT_SPL	Spline	RCM_ENT_MTX	Mtext	RCM_ENT_MLN	Mline	RCM_ENT_GRP	Group	RCM_ENT_IMG	Image	RCM_ENT_VPT	Viewport	RCM_ENT_HAT	Hatch	RCM_ENT_RGN	Region	RCM_ENT_BDY	Body	RCM_ENT_XRF	Xreferences	RCM_ENT_GMI	Geographic Map Image	RCM_ENT_ACS	All ACIS entities: RCM_ENT_3DS, RCM_ENT_RGN,RCM_ENT_BDY	RCM_ENT_ALL	Any entity	RCM_ENT_SGL	Hide this item if a single entity is selected	RCM_ENT_MUL	Hide this item if multiple entities are selected
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CUI file format

Item	Values (Default Value)	Required?	Description
<i>Macro.Entity</i> (continued)	Text (RCM_ENT_ALL)	No	Continued: Entity Value Description RCM_ENT_MLR Multileader RCM_ENT_REF Refedit RCM_ENT_BIM IFC or RVT Underlay RCM_ENT_BED BEdit RCM_ENT_BEL Building Elevation Line RCM_ENT_BSL Building Section Line RCM_ENT_2DS 2D Section or Elevation RCM_ENT_TBL Table RCM_ENT_ARR All array entities: RCM_ENT_ARR_POL, RCM_ENT_ARR_PAT, RCM_ENT_ARR_REC) RCM_ENT_ARR_POL ARRAY Polar Array RCM_ENT_ARR_PAT ARRAY Path Array RCM_ENT_ARR_REC ARRAY Rectangular Array
<i>Macro.CheckedExpression</i>	Text ("")	No	Controls whether the button or menu item is in checked state based on value of the system variable.
<i>Macro.GreyedOutExpression</i>	Text ("")	No	Controls whether the button or menu item is grayed-out (disabled) based on the value of the system variable.
<i>Macro.HideExpression</i>	Text ("")	No	Controls whether the button or menu item is visible based on value of the system variable.
DoubleClickAction			
<i>UID</i>	Text	Yes	ID of a DoubleClickAction. Unique among all UIDs.
<i>DxfName</i>	Text	Yes	DXF name of an entity type to be bound to this double-click action.
<i>MenuItem.MacroRef.MenuMacroID</i>	Text ("")	No	ID of a macro to be bound to this double-click action.
PopupMenu			
<i>UID</i>	Text	Yes	ID of a PopMenu. Unique among all UIDs.
<i>Alias</i>	Text	Yes	Language independent tear-off-name, usually "POPNN" where NN is a number.
<i>Name.ID</i>	4 byte integer (0)	No	Text resource id of popup menu name. Used to extract localized menu name.
<i>Name</i>	Text	Yes	Default popup menu name.
PopMenuItem			
<i>IsSeparator</i>	true/false (false)	No	Defines whether the menu item is a separator.
<i>NameRef.ID</i>	4 byte integer (0)	No	Text resource id of menu item name. Used to extract localized menu item name.
<i>NameRef</i>	Text ("")	No	Default menu item name.
<i>MenuItem.MacroRef.MenuMacroID</i>	Text ("")	No	Id of a macro to be bound to this menu item.
PopupMenuRef			

CUI file format

Item	Values (Default Value)	Required?	Description
<i>pUID</i>	Text	Yes	ID of a popup menu to be bound to this menu item as a sub-menu. Unique among all UIDs.
<i>ToolTip.ID</i>	4 byte integer (0)	No	Text resource id of menu item tooltip. Used to extract localized text.
<i>ToolTip</i>	Text	Yes	Default menu item tooltip.
<i>HelpString.ID</i>	4 byte integer (0)	No	Text resource id of menu item help string. Used to extract localized text.
<i>HelpString</i>	Text	Yes	Default menu item help string.
	Text (CUI_DEFAULT)	No	Determines the of PopMenuRef, constructed as a comma-separated concatenation of the values, calculated with a logical OR applied to all values. For details about available values, see "Value Description" on page 926 in this chapter.
<i>Entity</i>	Text (RCM_ENT_ALL)	No	Determines the entity of PopMenuRef, constructed as a comma-separated concatenation of the values and calculated with a logical OR applied to all values. For details about available values, see "Entity Value Description" on page 927 in this chapter.
<i>CheckedExpression</i>	Text ("")	No	Controls whether the button or menu item is in checked state based on value of the system variable.
<i>GreyedOutExpression</i>	Text ("")	No	Controls whether the button or menu item is grayed-out (disabled) based on the value of the system variable.
<i>HideExpression</i>	Text ("")	No	Controls whether the button or menu item is visible based on value of the system variable.
Toolbar			
<i>UID</i>	Text	Yes	ID of a Toolbar. Unique among all UIDs.
<i>ToolbarOrient</i>	float/top/left/right/ bottom (float)	No	Specifies toolbar orientation.
<i>ToolbarVisible</i>	hide/show (hide)	No	Specifies whether to show or hide toolbar.
<i>rows</i>	4 byte integer (1)	No	Number of button rows for a toolbar.
<i>xval</i>	4 byte integer (0)	No	X coordinate of a floating toolbar and zero-based position index of a docked toolbar.
<i>yval</i>	4 byte integer (0)	No	Y coordinate of a floating toolbar and zero-based position index of a docking row for a docked toolbar.
<i>Name</i>	Text	Yes	Name of a toolbar.
<i>UseOwnIcon</i>	true/false (false)	No	Specifies whether to use single icon when toolbar is used as flyout.
ToolbarButton			
<i>UID</i>	Text	Yes	ID of a ToolbarButton. Unique among all UIDs.
<i>IsSeparator</i>	true/false (false)	No	Defines whether the button is a separator.
<i>NameRef.ID</i>	4 byte integer (0)	No	Text resource id of button name. Used to extract localized button tooltip.

CUI file format

Item	Values (Default Value)	Required?	Description
<i>NameRef</i>	Text ("")	No	Default button name and tooltip.
<i>MenuItem.MacroRef.MenuMacroID</i>	Text ("")	No	ID of a macro to be bound to this button.
ToolbarFlyout			
<i>pTargetId</i>	8 byte integer	Yes	ID of a popup menu to be bound to this item as a fly-out.
<i>ToolTip.ID</i>	4 byte integer (0)	No	Text resource id of tooltip. Used to extract localized text.
<i>ToolTip</i>		Yes	Default fly-out tooltip.
<i>HelpString.ID</i>	4 byte integer (0)	No	Text resource id of help string. Used to extract localized text.
<i>HelpString</i>	Text	Yes	Default fly-out help string.
<i>SmallImage</i>	4 byte integer (0)	No	Image resource id of small icon, or path to local image file.
<i>LargeImage</i>	4 byte integer (0)	No	Image resource id of large icon.
<i>SmallImageHighContrast</i>	4 byte integer (0)	No	Image resource id of small high contrast icon.
<i>LargeImageHighContrast</i>	4 byte integer (0)	No	Image resource id of large high contrast icon.
<i>AcadToolbarConversion</i>	Text ("")	No	Used to extract fly-out icon.
	Text (CUI_DEFAULT)	No	Determines the of the fly-out, constructed as a comma-separated concatenation of values, calculated with a logical OR applied to all values. For details about available values, see "Value Description" on page 926 in this chapter.
<i>Entity</i>	Text (RCM_ENT_ALL)	No	Determines the entity of the fly-out, constructed as a comma-separated concatenation of the values and calculated with a logical OR applied to all values. For details about available values, see "Entity Value Description" on page 927 in this chapter.
<i>CheckedExpression</i>	Text ("")	No	Controls whether the button or menu item is in the checked state based on the value of the system variable.
<i>GreyedOutExpression</i>	Text ("")	No	Controls whether the button or menu item is grayed-out (disabled) based on the value of the system variable.
<i>HideExpression</i>	Text ("")	No	Controls whether the button or menu item is visible based on value of the system variable.
ToolbarControl			
<i>UID</i>	tibctrl-color/ tibctrl-linetype/ tibctrl-lineweight/ tibctrl-printstyle/ tibctrl-textstyle/ tibctrl- dimensionstyle	Yes	ID of an AppButton. Unique among all UIDs. Any other UID is not allowed.
<i>HelpString.ID</i>	4 byte integer (0)	No	Text resource id of help string. Used to extract localized text.
<i>HelpString</i>	Text	Yes	Default control help string.
	Text (CUI_DEFAULT)	No	Determines the of the toolbar, constructed as a comma-separated concatenation of the values, calculated with a logical OR applied to all values. For details about available values, see "Value Description" on page 926 in this chapter.

Using macros in .cui files

Macros in .cui files can run commands with custom options for tool palettes, menus, the ribbon (for versions that have a ribbon), toolbars, keyboard shortcuts, and aliases. For example:

```
^C^C^CZOOM;E;QSAVE;QPRINT
```

This example zooms to extents, saves the file, and then prints the file.

You can add macros to .cui files manually using a text editor and also in the Properties area of the Customize dialog box. For example, to add a macro to a menu item, see “Setting properties of menu items” on page 874 in this chapter.

The following table describes special characters that can be included in macros that are used in .cui files.

Character	Description
;	Equivalent to pressing the Enter key. For example, adding ; or ^M advances to the next prompt: _DRAWORDER; BACK _DRAWORDER^M_BACK
^M	
^C	Equivalent to pressing the Esc key. Adding three ^C (Ctrl+C) characters before a command cancels any active commands or dialog boxes. For example: ^C^C^C_LINE Note that ^C is ignored if used outside of a command; it doesn't clear the selection.
^	Equivalent to pressing the Ctrl key. Adding a carrot before a character runs the equivalent shortcut, for example, to toggle entity snapping on or off: ^B
^^	Suppresses the display of all prompts and input for the subsequent command.
^Q	Suppresses the display of all prompts and input for all subsequent commands.
blank space	Equivalent to entering a space, for example, a blank space between commands is the same as pressing the Spacebar or the Enter key.
.	Accesses the original or default command if it was undefined using the Undefine command. For example: .LINE
_	Uses the default English translation for subsequent commands and options. This allows the macro to run on all languages. For example: _ARC
-	Runs the command line version of the command instead of the dialog box version. This is useful when running commands in scripts. For example: -BOUNDARY
'	Runs the command transparently, i.e., within a running command. For example: _LINE;_'COLOR

Character	Description
*	Repeats the subsequent command or entire macro until another command is run or the user cancels it. For example: *CIRCLE
=*	Displays the current top-level user interface item.
\$	Starts a DIESEL routine. For example: \$M=\$(getvar,cmdnames)
@	Inputs the coordinates of the previously specified point. For example: _CIRCLE:@_DIAMETER3
^B	Turns snaps on or off by toggling the SNAPMODE system variable. For example: ^B_CIRCLE;
^F	Forces a cancel. Can be used outside of a command to clear a selection.
^E	Sets the isometric plane to the next option (Top, Right, or Left) by setting the SNAPISOPAIR system variable. Equivalent to F5 and Ctrl+E.
^G	Turns the reference grid on or off by toggling the GRIDMODE system variable. Equivalent to F7 and Ctrl+G.
^O	Turns orthogonal mode on or off by toggling the ORTHOMODE system variable. Equivalent to F8 and Ctrl+L.
^P	Turns menu echo mode on or off by setting the MENU ECHO system variable.
^T	Turns tablet mode on or off by toggling the TABMODE system variable.
^V	Activates the next viewport which affects the CVPORT system variable.

Creating customized toolbar files manually

With PTCAD, the easiest way to share toolbars is to create a new .cui file using the Customize UI command and share that file with other PTCAD users. For more details, see “Creating a new .cui file and copying settings” on page 919 in this chapter.

You can also manually create toolbars that you can share by creating or modifying an .mnu file manually.

To manually create a toolbar that you can share

- 1 Open any ASCII or Unicode text editor.
- 2 Use the following toolbar syntax to type the toolbar definitions:

```
***MENUGROUP=group_name

***TOOLBARS
**toolbar_name
ID_toolbar_name [_Toolbar ("toolbar_name", orient, visible, xval, yval,
rows)]
ID_button_name [_Button ("button_name", id_small, id_large)]command

***HELPSTRINGS
ID_button_name [help_string]
```

Toolbar syntax and explanations

Item	Explanation
<i>group_name</i>	Menu group name.
<i>toolbar_name</i>	Toolbar name.
<i>orient</i>	Orientation. Select Floating, Top, Bottom, Left, and Right (not case sensitive).
<i>visible</i>	. Select Show or Hide (not case sensitive).
<i>xval</i>	x coordinate, designated in pixels from left edge of screen. Suggested value: 200.
<i>yval</i>	y coordinate, designated in pixels from top edge of screen. Suggested value: 200.
<i>rows</i>	Number of rows
<i>button_name</i>	Tool name. Appears as a ToolTip when user places cursor over button.
<i>id_small</i>	Name of small (16 x 16 pixels) icon bitmap (BMP) file. This file must be located in the PTCAD folder specified in Tools > Options > Paths.
<i>id_large</i>	Name of large (32 x 32 pixels) icon bitmap (BMP) file. This file must be located in the PTCAD folder specified in Tools > Options > Paths.
<i>command</i>	Command string (Example: ^C^C_LINE).
<i>help_string</i>	Help string. Appears in the status bar when cursor passes over the button.

- 3 Save the file to the PTCAD folder with a *.mnu extension.

Example Toolbar File Contents

```


***MENUGROUP=example

***TOOLBARS
**NewDraw
ID_ NewDraw [_Toolbar("NewDraw ", _Bottom, _Show, 200, 200, 1)]
ID_Line_0 [_Button("Line", Iline.bmp, IL_line.bmp)]^C^C_line
ID_Hatch [_Button("Hatch", Ihatch.bmp, IL_hatch.bmp)]^C^C_hatch
ID_Dtext [_Button("Dtext", Idtext.bmp, IL_dtext.bmp)]^C^C_dtext
ID_0 [_Button("Circle Rad", Icirad.bmp, IL_cirad.bmp)]^C^Ccircle;
ID_Erase [_Button("Erase", Ierase.bmp, IL_erase.bmp)]^C^Cerase;

***HELPSTRINGS
ID_Line_0 [Creates straight line segments]
ID_Hatch [Fills an enclosed area with a nonassociative hatch pattern]
ID_Dtext [Displays text on screen as it is entered]
ID_0 [Allows user to draw a circle with a radius value]
ID_Erase [Removes objects from a drawing]

```

To open the file on another computer

- 1 Copy the toolbar (.mnu) file and all related custom bitmap (*.bmp) files to the PTCAD folder on the other computer.
- 2 Open PTCAD.
- 3 Do one of the following to choose Customize UI (- On the ribbon, choose Tools > Customize UI (in Manage).
- On the menu, choose Tools > Customize UI.
- Type *cui* and press Enter.
- 4 Click the Transfer tab.
- 5 On the left, click Open.
- 6 Select the new toolbar file, then click Open.
- 7 Click OK.

Customizing entities

PTCAD provides ways to customize entities beyond common formats, dimensions, and layers. Further customization includes using custom audio notes and using custom shape files.

Working with audio notes

In PTCAD you can record and attach audio notes, which are sound clips attached to entities and available for playback at any time. For example, audio notes in a factory floor plan can describe maintenance activities for specific areas of the factory and employees can play the audio notes to hear instructions.

When you include audio notes in your drawing, they are saved in the drawing file — not in a separate audio file. Note that this can increase the file size of your drawing. Audio notes created from an existing *.wav file are also saved in the drawing file and the separate *.wav file remains unchanged.


You can play audio notes in PTCAD only.

Attaching audio notes

Audio notes can be created from existing *.wav files. You can also record an audio note using any microphone that works with your computer. Many computers have a built-in microphone. If you do not have a microphone and you record an audio note, the audio note will contain no sound.

An audio note icon displays on entities that have attached audio notes. The icon is for display only and cannot be selected.

To record and attach an audio note to entities

- 1 Do one of the following to choose Audio Note ():
 - On the ribbon, choose Tools > Audio Note (in Manage).
 - On the menu, choose Tools > Audio Note.
 - On the Tools toolbar, click the Audio Note tool.
 - Type *audionote* and then press Enter.
- 2 Select the desired entities, and then press Enter.
- 3 Choose Record.
- 4 Click Record to begin the audio note, and then speak into the microphone.
- 5 Click Stop to end the audio note.
- 6 To test the audio note, click Test, and then use Pause and Stop in the Test Audio Note area.
- 7 If necessary, you can record the audio note again. This overwrites the previous audio note for the entities you selected in Step 2.
- 8 Click Attach.

You can modify an audio note after you attach it to an entity.

Select the entity and re-record the audio note using the previous steps.

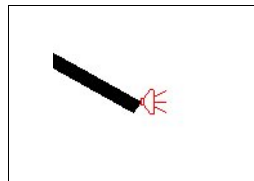
To attach an audio note to entities using a .wav file

- 1 Do one of the following to choose Audio Note (🔊):
 - On the ribbon, choose Tools > Audio Note (in Manage).
 - On the menu, choose Tools > Audio Note.
 - On the Tools toolbar, click the Audio Note tool.
 - Type *audionote* and then press Enter.
- 2 Select the desired entities, and then press Enter.
- 3 Choose Attach.
- 4 Select the desired *.wav file, and then click Open.

Selecting audio notes

Audio notes themselves cannot be selected. Instead, you select the entity that corresponds to the desired audio note.

An audio note icon displays on entities that have attached audio notes. The icon is for display only and cannot be selected.




An audio note icon displays on an entity, signifying that an audio note is attached to the entity.

To select audio notes

- 1 Do one of the following to choose Audio Note (🔊):
 - On the ribbon, choose Tools > Audio Note (in Manage).
 - On the menu, choose Tools > Audio Note.
 - On the Tools toolbar, click the Audio Note tool.
 - Type *audionote* and then press Enter.
- 2 Select the entity that is attached to the desired audio note, and then press Enter. You cannot select the audio note icon that displays on the entity.
- 3 Choose an option to continue working with audio notes.

Playing audio notes

To play audio notes


- 1 Do one of the following to choose Audio Note ():
 - On the ribbon, choose Tools > Audio Note (in Manage).
 - On the menu, choose Tools > Audio Note.
 - On the Tools toolbar, click the Audio Note tool.
 - Type *audionote* and then press Enter.
- 2 Select the entity that has the audio note you want to play, and then press Enter.
- 3 Choose Playback.
- 4 In the Playback dialog box, do the following:
 - Click Play to begin playback.
 - To pause the audio note temporarily, click Pause and then click Resume to continue.
 - Click Stop to end playback.
- 5 Click OK.

Deleting audio notes

You can remove audio notes from selected entities. In some cases, you may want to remove extra audio notes to reduce the drawing file size.

When you remove an audio note from an entity, the audio note is removed permanently and cannot be recovered. However, if the audio note was created using an existing *.wav file, that *.wav file is not removed.

To delete audio notes

- 1 Do one of the following to choose Audio Note ():
 - On the ribbon, choose Tools > Audio Note (in Manage).
 - On the menu, choose Tools > Audio Note.
 - On the Tools toolbar, click the Audio Note tool.
 - Type *audionote* and then press Enter.
- 2 Select entities that have the audio notes you want to delete, and then press Enter.
- 3 Choose Delete.

Using shape files

Shapes are entities that you define for use as drawing symbols and text fonts. You can specify the scale and rotation to use for each shape as you add it.

To use shape files, you first load the compiled shape file that defines the shape. Then you use insert shapes from the file into your drawing.

To load a shape file

- 1 Type *load* and then press Enter.
- 2 In the Load Shape File dialog box, select a shape file.
- 3 Click OK to close the Load Shape File dialog box.

To use a shape file

- 1 Type *shape* and then press Enter.
- 2 Type a shape name and press Enter.
- 3 Specify an insertion point.
- 4 Specify a height.
- 5 Specify a rotation angle.

Creating and replaying scripts

PTCAD can record anything you type on the keyboard and any points you select in a drawing. You can save all of these actions to a script file (with the *.scr extension) and then repeat them by replaying the script. You can use scripts for successfully repeating commands, showing snapshots in a slide show, or batch printing. You can also load and run script files created for use with AutoCAD.

A script is a form of text file that contains one line of text or other data for each action. PTCAD includes a Script Recorder that records both command line entries and screen picks you make with your mouse. For example, when you type a command and press Enter, it is recorded on a line in the script file. When you select a point in a drawing, the coordinate of that point is recorded on a line in the script file.

You can also create script files outside PTCAD using a text editor (such as Microsoft® Notepad or Microsoft® WordPad) or a word-processing program (such as Microsoft® Word) that saves the file in ASCII format. The file type and extension must be .scr.


Recording scripts

After you activate the Script Recorder, every keyboard entry you make and any points you select in a drawing are recorded until you stop the Script Recorder. You can play back your script at any time.

CAUTION


The Script Recorder does not record your use of toolbars, menus, or dialog boxes. Using these elements while recording a script causes unpredictable results.

To record a script

- 1 Do one of the following to choose Record Script (- On the ribbon, choose Tools > Record Script (in Applications).
- On the menu, choose Tools > Record Actions > Record Script.
- On the Tools toolbar, click the Record Script tool.
- Type *recscript* and then press Enter.
- 2 Specify the name of the script file you want to create.
- 3 Click Save.
- 4 Type commands on the keyboard.

The Script Recorder records all keyboard entries and all points you select in the drawing, saving everything to the script file.

To stop recording


- 1 Do one of the following to choose Stop Recording ():
 - On the ribbon, choose Tools > Stop Recording (in Applications).
 - On the menu, choose Tools > Record Actions > Stop Recording.
 - On the Tools toolbar, click the Stop Recording tool.
 - Type *stopscript* and then press Enter.

Replaying scripts

There are a number of ways you can replay scripts:

- Choose the Run Script command in PTCAD.
- Drag and drop the script file from Windows Explorer or the desktop onto the PTCAD drawing window.
- Run PTCAD with the script at startup using the Command Prompt.

To play a script in PTCAD

- 1 Do one of the following to choose ():
 - On the ribbon, choose Tools > Run Script (in Applications).
 - On the menu, choose Tools > Record Actions > Run Script.
 - On the Tools toolbar, click the Run Script tool.
 - Type *script* and then press Enter.
- 2 In the Run Script dialog box, specify the name of the script file you want to run.
- 3 Click Open.

The program immediately runs the script, performing all the actions originally recorded.


To start PTCAD and play a script using the Command Prompt

- 1 Open the Command Prompt window.
- 2 Type the following, where filename.scr is the name of the script, including the path to the .scr file:

```
icad.exe /b D:\temp\filename.scr
```


Modifying scripts

To append to a script

- 1 Do one of the following to choose Record Script (- On the ribbon, choose Tools > Record Script (in Applications).
- On the menu, choose Tools > Record Actions > Record Script.
- On the Tools toolbar, click the Record Script tool.
- Type *rescript* and then press Enter.
- 2 In the Record Script dialog box, select the Append To Script check box.
- 3 Select the existing script file to append.
- 4 Click Save.
- 5 Click Yes to the prompt asking whether you want to replace the existing script.
- 6 Repeat the procedure to enter additional commands and steps.

Customizing script files using a text editor

The easiest way to create or customize a script file manually is to record and save a script in PTCAD that approximates the commands and actions you want, then customize the script as necessary using a text editor. In addition to PTCAD commands, it is also common to include standard LISP functions and commands in script files.

Be sure to save the script file in ASCII format with an .scr file extension.

Understanding script syntax

The following table describes the syntax to use in script files when calling commands and specifying actions. For example, script files can contain comments; any line that begins with a semicolon is considered a comment, and the program ignores these lines when replaying the script.

Script syntax

Character	Description
;	Semicolons indicate comments. Any text after a semicolon on the same line is ignored.
blank space	Spaces and new, empty lines in scripts are the equivalent of pressing the Enter key.
_	Underscores before a command or keyword run the English name of the subsequent command, not a localized version in a different language. For example, a script with the syntax <code>_CIRCLE</code> can be used in other language versions of the program without replacing every occurrence of the command name CIRCLE.

Script syntax

Character	Description
.	Periods before commands run the original, standard version of the subsequent command. This ensures that a redefined version of a command is not run.
-	Dashes before commands run the command line version of the command. For example, LAYER runs the dialog box version of the command, and -LAYER runs the command line version where all input can be entered via the script.
'	Apostrophes before commands run the command transparently, i.e., within another command. For example, with the LINE command running, change the current color without interrupting the command by typing 'COLOR.

The following is text from a sample script:

```
; Draw a circle (this is a comment)
_.CIRCLE 0,0,0 _DIAMETER 3
```

The following shows the same script but with @ representing the location of spaces for clarity:

```
; Draw a circle (this is a comment)
_.CIRCLE@0,0,0@_DIAMETER@3@
```

The spaces are critical, representing the Enter key and moving the command to a new option or exiting the command. The space after the number 3 (the circle diameter) represents the user pressing Enter to submit the diameter to complete the command. A new blank line could also have been used in the script file and produced the same result as using a space.

Using common commands and system variables

The following table describes commands and system variables that are commonly used in scripts.

Common commands and system variables used in scripts

Character	Description
'DELAY	Pauses the script for a specified number of milliseconds.
FILEDIA	Turns off dialog boxes, which forces the input to the command line where the script can provide filenames and other information.
'GRAPHSCR	Switches from the drawing area to the text window.
MENUECHO	Turns off command prompts.
OSMODE	Turns snap settings on or off to make sure selected points are at the correct location.
PICKBOX	Adjusts the size of the entity selection pickbox to help accurately select entities in the drawing.

Common commands and system variables used in scripts

Character	Description
RESUME	Continues a script after a break or interruption.
RSCRIPT	Repeats a script.
'TEXTSCR	Switches from the text window to the drawing area.

The following is a sample script that opens a drawing file without displaying a dialog box:

```
; Open a file
_.FILEDIA 0
_.OPEN C:\\Temp\\My File.dwg
; Reset FILEDIA
_.FILEDIA 1
```

Running PTCAD with command line options

Scripts can start PTCAD with or without custom options. These custom options can also be called when starting PTCAD from the operating system's Command Prompt window.

Command line options for customizing the start of PTCAD

Character	Description
?	Displays the list of available options.
Nologo	Prevents the splash screen from displaying during startup.
P <i>profilename</i>	Specifies an overriding profile name.
S <i>path1;...</i>	Specifies the full path to an overriding support path.
T <i>templatename</i>	Creates a new drawing using the specified template.
B <i>scriptname</i>	Specifies the full path to the script to run automatically after startup.
Ld <i>applicationname</i>	Loads a third-party application automatically after startup.

To start PTCAD from the Windows Command Prompt using command line options

- 1 Open the Windows Command Prompt.
- 2 Run PTCAD with any of the available options, for example, the following starts PTCAD and automatically runs a script:

```
icad.exe /b filename.scr
```

PTCAD also supports several standard operating system command line options, such as /RegServer and /UnregServer.

Running custom applications from a script

Scripts can include lines that run LISP routines and entire SDS and IRX applications.

To run a custom application from a script

- Add a line in your script, where filename is the name of your custom application:

```
; Runs a LISP routine
load "D:\\Temp\\filename.lsp"
; Runs an SDS application
xload "D:/temp/filename.dll"
; Runs an IRX application
arxload "D:/temp/filename.irx"
```

Programming PTCAD

Another way you can customize PTCAD is to add custom programs written in any of several programming languages that run within PTCAD, including the following: TX, .NET, LISP, SDS, DCL, VBA, and DIESEL.

In PTCAD, you can run many programs originally created for use with AutoCAD. Specifically, you can use programs written entirely in AutoLISP with no modification. In addition, you can run many ADS programs originally written for use with AutoCAD after first recompiling them using the PTCAD run-time libraries. Many AutoCAD third-party programs are compatible with PTCAD.

Use the online Help.

For information about programming for PTCAD, see the online Help for the PTCAD Developer's Reference.

Using TX/IRX applications

The TX/IRX interface is available for developers to create custom entities and applications.

To load a TX/IRX custom application

Advanced experience level

- 1 Do one of the following:
 - On the ribbon, choose Tools > Load Application (in Applications).
 - On the menu, choose Tools > Load Application.
 - Type *appload* and then press Enter.
 - Drag and drop the TX or IRX file into PTCAD.

- 2 In the Load Application Files dialog box, click Add File.
- 3 Select the TX or IRX file that you want to load, and then click Open.
- 4 Click OK.

To run a TX/IRX custom application

Advanced experience level

- 1 Do one of the following:
 - On the ribbon, choose Tools > Load Application (in Applications).
 - On the menu, choose Tools > Load Application.
 - Type *apload* and then press Enter.
- 2 In the Load Application Files dialog box, choose the routine you want to run (make sure that it is the only one selected), and then click Load.

TX/IRX custom applications can also be run automatically when the program starts.

You can add .tx and .irx files to a startup suite or use a script. For details, see “Loading custom applications automatically when PTCAD starts” on page 951 in this chapter.

For more information

- See the sample TX and IRX applications.
- Read the online documentation for TX and IRX, available in the Developer Reference of PTCAD Help.

Using .NET applications

PTCAD supports the .NET programming language in two ways:

- .NET Classic — More closely matches the .NET interface used by AutoCAD.
- .NET based on COM — Initial .NET interface supported by PTCAD.

To prepare a .NET custom application

- 1 In the .NET application, inherit the IICadPlugin interface that is defined in PTCAD as:

```
public interface IICadPlugin
{
    void PluginLoaded(object sender, EventArgs e);
    void PluginUnLoading(object sender, EventArgs e);
}
```

Where `PluginLoaded()` is called on plug-in load, the `sender` argument is a COM Interop model class `PTCAD.Application` for working with (storing it as a

class member), and `PluginUnloading()` is called right before the plug-in is unloaded.

To load a .NET Classic custom application

- 1 Do one of the following:
 - On the menu, choose Tools > Load Classic .NET Application.
 - Type *netload* and then press Enter.
- 2 In the Open an Assembly dialog box, select the .NET .dll file that you want to load, and then click Open.

.NET Classic custom applications can also be run automatically when the program starts. You can add .dll files for .NET to a startup suite or use a script. For details, see “Loading custom applications automatically when PTCAD starts” on page 951 in this chapter.

Using LISP routines

PTCAD supports the LISP programming language and is compatible with AutoLISP, the implementation of the LISP language in AutoCAD. This means that you can load and run any AutoLISP program written for use with AutoCAD.

To load a LISP routine

Advanced experience level

- 1 Do one of the following:
 - On the ribbon, choose Tools > Load Application (in Applications).
 - On the menu, choose Tools > Load Application.
 - Type *appload* and then press Enter.
 - Drag and drop the LISP file into PTCAD.
- 2 In the Load Application Files dialog box, click Add File.
- 3 Select the LISP file that you want to load, and then click Open.
- 4 Click OK.

Load LISP routines from the command bar.

In the command bar, type (load “d:/path/routine.lsp”), making sure to include the parentheses and the quotation marks, where d:/path is the drive and path where the LISP routine is located on your computer, and routine.lsp is the LISP routine file name.

To run a LISP routine

Advanced experience level

- 1 Do one of the following:
 - On the ribbon, choose Tools > Load Application (in Applications).
 - On the menu, choose Tools > Load Application.
 - Type *apload* and then press Enter.
- 2 In the Load Application Files dialog box, choose the routine you want to run (make sure that it is the only one selected), and then click Load.

Some LISP routines are created in such a way that you can run them by simply typing the name of the routine, or by typing a keyword, directly in the command bar. If nothing happens when you attempt to run the LISP routine from within the Load Application Files dialog box, turn on the display of the command bar or Prompt History window by choosing View > Display > Command Bar or View > Display > Prompt History Window, and look for an entry that is similar to the following:

```
Loading D:\path\routine.lsp
C:KEYWORD
```

where *D:\path\routine.lsp* is the complete drive, path, and file name of the LISP routine. You may need to scroll back several lines in the command bar or Prompt History window to find the lines indicating where the LISP routine was loaded. You can run the LISP routine by typing the name of the routine or keyword appearing after the C drive designation.

For example, if you loaded a LISP routine named *drawbox.lsp* and see the designation *C:DRAWBOX* in the command bar or Prompt History window, you can run the LISP routine by typing *drawbox* in the command bar.

LISP routines can also be run automatically when the program starts.

You can add .lsp and .mnl files to a startup suite or use a script. For details, see “Loading custom applications automatically when PTCAD starts” on page 951 in this chapter.

Using SDS applications

To write AutoCAD applications in C or C++, Autodesk® uses the ADS (AutoCAD Development System). This is an API (applications programming interface) that provides a library to access AutoCAD-specific functions and drawing data.

The equivalent in PTCAD is called SDS™, the Solutions Development System™. SDS is a C/C++ language interface compatible with the ADS interface in AutoCAD. Like scripts and AutoLISP, you can run your existing ADS applications in PTCAD. Simply recompile the source code using the SDS libraries provided on the PTCAD CD-ROM, or, if you use an AutoCAD program written by a third-party vendor, contact that vendor for the PTCAD version.

PTCAD provides the Sds.H file, which redefines ADS function names to their SDS equivalents. SDS supports the AutoCAD dialog control language (DCL), which is used by ADS to define the look of a dialog box. You can use all DCL files unmodified within SDS.

Understanding SDS compatibility

The primary difference between ADS and SDS is that all SDS functions have an sds_ prefix, and ADS functions have a variety of prefixes, such as ads_, acad_, and acrx_. The exception is dialog-related SDS functions, which have a dlg_ prefix. PTCAD accepts either prefix. Other differences include the additional SDS functions listed in the following table.

SDS functions with no ADS equivalent

SDS function name	Description
<i>sds_grclear</i>	Clears all graphics from the drawing window; similar to the LISP (<i>grclear</i>) function.
<i>sds_name_clear</i>	Clears the entity name or selection set.
<i>sds_name_equal</i>	Verifies whether two entity names or selection sets are equal.
<i>sds_name_nil</i>	Verifies whether the entity name or selection set has been cleared.
<i>sds_name_set</i>	Copies one entity name or selection set to another drawing.
<i>sds_pmtssget</i>	Similar to the ads_ssgget function, but allows you to display a prompt appropriate for the specific command, rather than the generic "Select object" prompt.
<i>sds_point_set</i>	Copies a point from one variable to another.
<i>sds_progresspercent</i>	Displays the percentage done in a progress bar.
<i>sds_progressstart</i>	Starts the progress bar.
<i>sds_progressstop</i>	Ends the progress bar.
<i>sds_readaliasfile</i>	Loads the PGP file into PTCAD.
<i>sds_sendmessage</i>	Sends a message to the PTCAD command line.
<i>sds_swapscreen</i>	Flips the off-screen device context to the display.

Some ADS functions are not supported in SDS, including: *ads_arxload*, *ads_arxloaded*, *ads_arxunload*, *ads_ssggetx*, *ads_ssGetKeywordCallbackPtr*, *ads_ssGetOtherCallbackPtr*, *adsw_acadMainWnd*, and *adsw_acadDocWnd*.

SDS custom applications can be run automatically when the program starts.

You can add .dll files for SDS to a startup suite or use a script. For details, see "Loading custom applications automatically when PTCAD starts" on page 951 in *this chapter*.

For more information

- Read the online documentation for SDS functions.
- See the \PTCAD\Api\Sds folder, which contains the SDS include, header, and library files.
- See the \PTCAD\Api\Dcl folder, which contains the core DCL files.

Using DDE applications

PTCAD supports the DDE API, which allows you to execute PTCAD com-mands at the command line from an external application. PTCAD includes a sam-ple command-line application that reads a script file and sends it to PTCAD for step-by-step execution.

To see a sample of DDE script execution

- 1 Compile the source code of the DDESample project. The sample application is located at:
Source\PTCAD\api\DDE\Samples\DDESample
- 2 Run PTCAD.
- 3 Run the DDESample application with the sample script file that is included in the VC project.
PTCAD shows the results.

Using DCL with PTCAD

PTCAD completely supports the AutoCAD DCL (dialog control language). DCL is used by AutoLISP functions to define the look of dialog boxes. You can use all DCL files unmodified within PTCAD.

Using VBA

PTCAD can be customized using Visual Basic for Applications (VBA) through an integrated interface, available from the PTCAD Tools menu. PTCAD features a broad range of objects, giving you the power to write your own custom applications that can run within PTCAD.

To load a VBA project

Advanced experience level

- 1 Do one of the following:
 - On the ribbon, choose Tools > Load VBA Project (in Applications).
 - On the menu, choose Tools > Visual Basic > Load VBA Project.
 - Type *vbiload* and then press Enter.
- 2 Locate and select the Visual Basic project file (.vbi file) to load, then click Open.

To run a VBA macro

Advanced experience level

- 1 Do one of the following:
 - On the ribbon, choose Tools > Macros (in Applications).
 - On the menu, choose Tools > Visual Basic > Macros.
 - Type *vbarun* and then press Enter.
- 2 In the Run PTCAD VBA Macro dialog box, enter the name of an existing VBA macro, and then click Run.

You can also create, delete, and edit VBA macros from this dialog box.

To edit a VBA macro

Advanced experience level

- 1 Do one of the following:
 - On the ribbon, choose Tools > Visual Basic Editor (in Applications).
 - On the menu, choose Tools > Visual Basic > Visual Basic Editor.
 - Type *vba* and then press Enter.
- 2 Use the Visual Basic Editor to write or debug VBA macros.

For more information

- Read the online documentation for VBA, available from the editor Help menu and from the PTCAD Help menu.
- Many publications are available that explain how to program in VBA.

Using DIESEL with PTCAD

PTCAD supports AutoCAD DIESEL (Direct Interactively Evaluated String Expression Language). DIESEL is a separate interpretive programming language that allows you to customize the following:

- Status bar
- Menus
- LISP functions

For more information

- Read the online documentation for DIESEL, available from the PTCAD Help.
- Several publications and tutorials are available online that explain how to use DIESEL.





Loading custom applications automatically when PTCAD starts

If you have custom applications that you want loaded each time the program starts, you can add those applications to a startup suite in PTCAD or use an external script.

Using a startup suite to load applications automatically

A startup suite is a list of custom application files that load each time PTCAD starts. The applications are loaded in the order they appear in the list.

To load an application from a startup suite when PTCAD starts

- 1 Do one of the following to choose Load Application ():
 - On the ribbon, choose Tools > Load Application (in Applications).
 - On the menu, choose Tools > Load Application.
 - Type *apload* and then press Enter.
- 2 Click Startup Suite.
- 3 Click Add ().
- 4 Select the application file you want automatically loaded when PTCAD starts.
- 5 Click Open.
- 6 If necessary, click Move Up () or Move Down () to reorder the startup suite list.
- 7 Click OK.
- 8 Click OK again.

Using a script to load applications automatically

Custom applications can be referenced in an external script, and then loaded each time PTCAD starts. Each application type has its own script requirements.

To run a TX/IRX custom application from a script when PTCAD starts

- 1 Create an ASCII file named *icad.rx* that contains the path to the IRX files on each line of the file.
- 2 Place *icad.rx* in the same folder as *icad.exe*.
- 3 Run PTCAD.

To load a .NET Classic custom application from a script when PTCAD starts

- 1 Create an ASCII file named *icad.net* that contains the path to a .NET plug-in file on each line of the file.
- 2 Place *icad.net* in the same folder as *icad.exe*.
- 3 Run PTCAD.

To load a LISP routine from a script when PTCAD starts

- 1 Create an ASCII file that contains the path to a LISP routine on each line of the file. Example file content:

```
(load "C:\\Users\\Documents\\username\\lisp routines\\areap.lsp")
```

- 2 Save the file with one of the following names:
 - ICAD.LSP — Automatically loads the referenced LISP routines when PTCAD starts.
 - ICADDOC.LSP — Automatically loads the referenced LISP routines when creating or opening each drawing file.
- 3 Place the .lsp file in the same folder as icad.exe.
- 4 Run PTCAD.

Use a system variable.

To make all drawings share the same LISP environment (variables, functions, etc.), set the LISPINIT system variable to 2.

To load an SDS custom application from a script when PTCAD starts

- 1 Create an ASCII file named icad.sds that contains the path to the .dll files on each line of the file.
- 2 Place icad.sds in the same folder as icad.exe.
- 3 Run PTCAD.

Using specialized pointing devices

In addition to traditional mouse pointers, PTCAD can also be used with a specialized mouse for CAD drawing and a digitizer tablet.

Using a specialized mouse for CAD

PTCAD works with specialized input devices from 3Dconnexion to streamline the CAD workflow and improve control and efficiency while drawing and navigating in two or three dimensions. Simply connect your mouse from 3Dconnexion to your computer, and it will automatically set up for your version of PTCAD.

PTCAD supports the following devices from 3Dconnexion:

- **SpaceMouse** — For details, see 3Dconnexion's SpaceMouse 3DxWare documentation online at docs.PTCAD.org.
- **CadMouse** — For details, see 3Dconnexion's CadMouse 3DxWare documentation at docs.PTCAD.org.

Using a digitizer tablet

PTCAD supports tablets compatible with the TabletWorks driver and has its own tablet overlay. For instructions on installing the tablet driver and using the buttons on your pointing device, refer to your hardware documentation.

Three options are available with the Tablet command:

- **Configure** Maps the tablet overlay to the tablet itself. Configure the tablet if you plan to select PTCAD tools from the tablet overlay.
- **Calibrate** Maps points on the tablet, in absolute coordinates, to points on a drawing. Calibrate the tablet if you intend to digitize points. This process is convenient for tracing paper drawings.
- **Tablet ON/OFF** Turns tablet calibration (digitizer mode) on and off. Use this to switch between tool selection and digitizer input. To use the tablet for tool selection, tablet mode must be turned off.

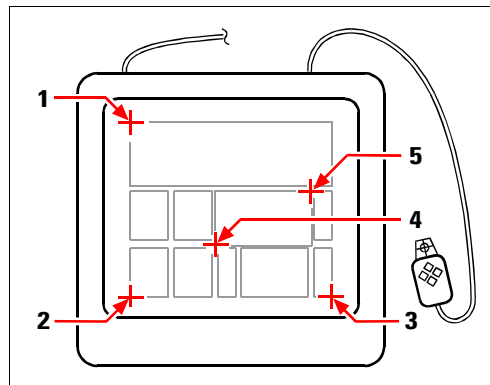
Configuring the tablet

Before you begin to configure the tablet, slip the tablet overlay underneath the tablet's plastic cover, and align it in accordance with the directions for your tablet. If your tablet does not have a plastic cover, align the overlay holes with the pegs on your tablet.

PTCAD provides a default configuration for tablets measuring 12" by 12". If you choose to accept the default, be sure to verify that the commands in the grid are activated correctly. If the default alignment does not work for your tablet, you'll need to begin the configuration process again and align the tablet yourself.

To configure the tablet for menu input

- 1 Do one of the following:
 - On the menu, choose Tools > Tablet > Configure.
 - Type *tablet*, choose Configure, and then press Enter.
- 2 In response to the prompt to align the tablet, do one of the following:
 - If your tablet is 12" by 12", choose No to accept the default alignment and conclude the configuration process.
 - To align the tablet yourself, choose Yes, and then continue with step 3.
- 3 Click the tablet pointer on the upper left alignment point of the overlay (⊕).
- 4 Click the tablet pointer on the lower left alignment point of the overlay (⊕).
- 5 Click the tablet pointer on the lower right alignment point of the overlay (⊕).
- 6 Click the tablet pointer on the lower left alignment point of the Workspace area of the overlay (⊕).
- 7 Click the tablet pointer on the upper right alignment of the Workspace area of the overlay (⊕).



Click points in the order shown to configure your tablet for menu input.

To turn tablet mode on or off

- 1 Do one of the following:
 - Choose Tools > Tablet > Tablet On (or Tablet Off).
 - Type *tablet*, choose On or Off, and then press Enter.
 - Press the F4 key to toggle tablet mode on or off.
 - On the status bar, double-click TABLET to turn tablet mode on or off.

Calibrating the tablet

You must specify at least two points for tablet calibration. However, the more points you specify, the more accurate the transformation between the tablet and the screen points. Specifying additional points is particularly useful if you plan to trace a paper drawing that is not orthogonal, such as an aerial photograph.

Understanding transformation types

Depending on the number of points specified, you have a choice of transformation types to use, along with the type recommended for PTCAD.

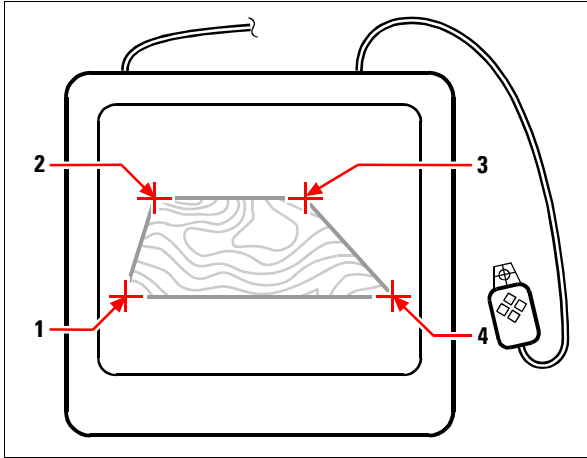
Transformation refers to the calculation of the points on the screen that correspond to points you digitize on the tablet.

- **Orthogonal** Preserves all angles between lines, preserves all relative distances, and, in general, preserves shapes. If you specify only two points, an orthogonal transformation is automatically generated. The orthogonal transformation type yields the most precise mapping between the digitized points on the tablet and the corresponding points on the screen.
- **Affine** Preserves parallel lines, but not necessarily the angles between intersecting lines. If you specified three points, PTCAD can no longer represent this mapping as an exact orthogonal transformation. Therefore, you have a choice of an exact affine transformation or a “best fit” orthogonal transformation.
- **Projective** Does not preserve parallel lines or angles. If you specify exactly four points, you have a choice of an exact projective transformation or a “best fit” orthogonal or affine transformation.

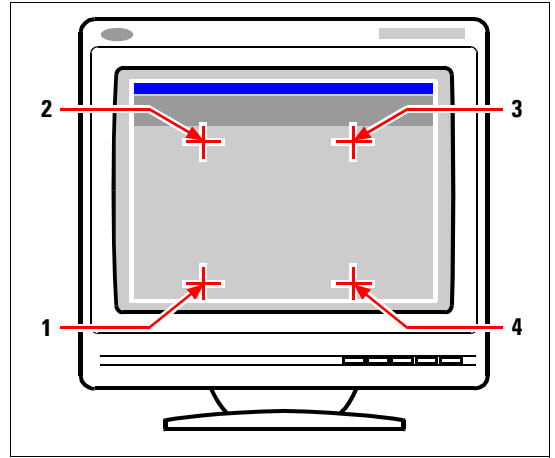
Recommended and “best fit” transformation types

Number of points specified	Transformation type recommended	“Best fit” (approximate)
2	Orthogonal	None
3	Affine	Orthogonal
4	Projective	Orthogonal, Affine
5-10	None	Orthogonal, Affine

Choose the recommended type unless you know it will not be appropriate for what you are digitizing. The most appropriate type is not always the one with the least error; for example, you might digitize three points and select the orthogonal transformation, even though the affine transformation would yield a closer representation of your calibration entries.

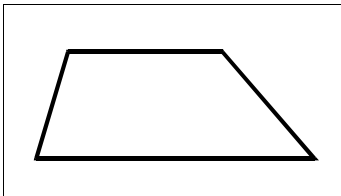


Digitizing by selecting points (1, 2, 3, and 4) on the tablet.

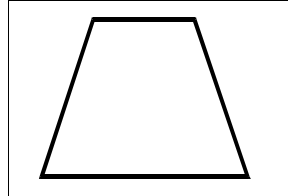


Corresponding screen coordinates:

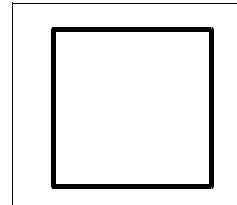
- 1 Coordinate specification X0, Y0, Z0
- 2 Coordinate specification X0, Y5, Z0
- 3 Coordinate specification X5, Y5, Z0
- 4 Coordinate specification X5, Y0, Z0



Screen result of orthogonal transformation after tracing polyline on tablet.



Screen result of affine transformation after tracing polyline on tablet.



Screen result of projective transformation after tracing polyline on tablet.

To calibrate your tablet for digitizing points

- 1 Do one of the following:
 - Choose Tools > Tablet > Calibrate.
 - Type *tablet*, choose Calibrate, and then press Enter.
- 2 Click a point on the tablet to define as the first calibration point.
- 3 Specify a point within the PTCAD drawing window to correspond to the point you digitized on the tablet, or enter coordinate values in the command bar.
- 4 Click a point on the tablet to define as the second calibration point.

Specify a point within the PTCAD drawing window to correspond to the point you digitized on the tablet, or enter coordinate values in the command bar. To specify more than the two points required, click a point on the tablet to define as the third calibration point. You can enter up to 10 points.
- 5 Choose the transformation type, and then press Enter.

Customizing the tablet interface

You can customize the digitizer tablet interface by using the LISP commands integrated with PTCAD, even if you are not familiar with LISP. For instructions, see “Customizing the Tablet Interface” in the PTCAD online Help.

Restoring PTCAD to its default settings

The Restore Default Settings tool backs up program settings and files and then restores them to their original state as they were when PTCAD was installed.

Restoring the default settings can be helpful if the program isn't working as expected. For example, if linetypes or print styles are no longer correct, restoring the default settings may help.

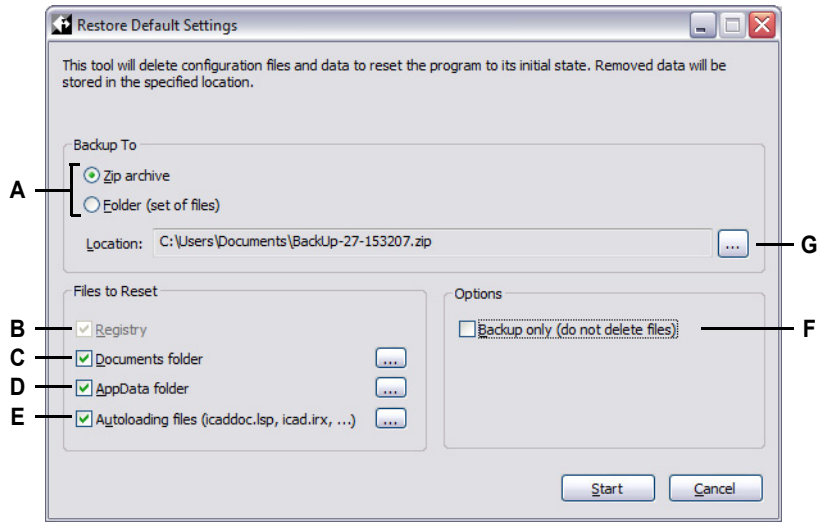
PTCAD can back up and restore PTCAD's registry settings, Documents folder, and AppData folder, and can back up (but not restore) custom files that automatically load when PTCAD starts.

Restoring the default program settings is different than resetting the user interface, which include menus, the ribbon (if your program supports it), toolbars, keyboard shortcuts, aliases, and double-click actions. For details about resetting the user interface, see "Loading customization files" on page 917 in this chapter.

To restore and back up PTCAD default settings

- 1 Close PTCAD, if it is open.
- 2 From the Windows Start menu, choose Restore Default Settings where PTCAD appears in the program list.
- 3 In Backup To, choose one of the following:
 - **Zip archive** Compresses and saves the files in a single .zip file.
 - **Folder (set of files)** Saves individual files in a folder.
- 4 Click [...] to specify the location of the backup.
- 5 Select any of the following:
 - **Registry** PTCAD's registry settings include the paths to folders where many settings are stored. You can unmark this option only if Backup Only is marked.
 - **Documents folder** PTCAD's Documents folder contains linetypes, print styles, printer configuration (.pc3) files, textures, and more.
 - **AppData folder** PTCAD's AppData folder contains color book files, backup drawing files, spelling libraries, and templates.
 - **Autoloading files** Custom applications that automatically load are located in the PTCAD installation folder, for example, icaddoc.lsp, icad.irx, icad.net, and icad.sds. These files can be backed up but cannot be restored.
- 6 To create a backup without restoring any settings, mark Backup Only.
- 7 Click Start.

The status displays in the Restore Status dialog box.
- 8 To view the backup files, click Open Folder.
- 9 When finished, click Exit.



- A** Select whether to backup all files to a zip file or a folder.
- B** Mark to include registry settings. This option is available only if Backup Only is marked.
- C** Mark to include PTCAD's Documents folder.
- D** Mark to include PTCAD's AppData folder.
- E** Mark to include custom application files that automatically load at program startup.
- F** Mark to create only a backup and not restore any default settings.
- G** Click to select the location where the backup will be saved.

Understanding AutoCAD compatibility

PTCAD was designed to interface with AutoCAD as seamlessly as possible. There are, however, some differences for which those accustomed to working in AutoCAD may need to adjust. This appendix provides information specifically for that audience.

Topics in this chapter

<i>Using enhanced AutoCAD commands in PTCAD</i>	<i>962</i>
<i>Using additional selection sets</i>	<i>963</i>
<i>Using additional PTCAD commands.....</i>	<i>964</i>
<i>Supporting AutoCAD customization</i>	<i>967</i>
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<i>Importing and exporting customization files</i>	<i>969</i>
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<i>PTCAD and AutoCAD list of terms</i>	<i>972</i>

Using enhanced AutoCAD commands in PTCAD

PTCAD enhances several AutoCAD commands by providing more options. For example, if you hold down the Shift key, PTCAD is placed temporarily in orthogonal mode, which you will find to be a useful feature for drawing at right angles. The following table lists examples of other commands with enhanced options.

Enhanced PTCAD commands

Command	Enhanced option name	Explanation
circle	Arc	Converts an arc to a circle.
donut	2point	Determines the outside diameter of a donut by two pick points.
donut	3point	Determines the outside diameter of a donut by three pick points.
donut	RadTanTan	Determines the outside diameter of a donut by tangent points.
line	Angle	Draws a line by angle, followed by a length.
line	Length	Draws a line by a length, followed by an angle.
msnapshot (mslide) and vsnapshot (vslide)	EMF	Saves and views screen images in enhanced metafile format.
msnapshot (mslide) and vsnapshot (vslide)	WMF	Saves and views screen images in Windows metafile format.
parallel (offset)	Both sides	Copies an entity parallel on both sides.
plane (solid)	Rectangle	Draws a rectangular solid at any angle.
plane (solid)	Square	Draws a square solid at any angle.
plane (solid)	Triangle	Draws an equilateral solid at any angle.
rectangle	Square	Draws a square rectangle.
rectangle	Rotated	Draws a rotated rectangle.

Using additional selection sets

PTCAD has additional selection-set options not found in AutoCAD. In particular, in the circle selection set, the Crossing Circle (CC), Outside Circle (OC), and Window Circle (WC) options select all entities relative to the same distance (radius) of a central pick point.

The following table lists and describes the additional selection sets in PTCAD.

Additional selection set options

Selection mode	Description
CC	Crossing Circle: Selects all entities within and crossing a circular area.
D	Selection method: Displays the Drawing Settings dialog box.
O	Outside Window: Selects all entities outside a rectangular area; this is the inverse of the Crossing Window option.
OC	Outside Circle: Selects all entities outside a circular area.
OP	Outside Polygon: Selects all entities outside a polygonal area; this is the inverse of the CP (Crossing Polygon) and WP (Window Polygon) options.
Filter	Filter: Selects according to properties, such as color, layer, linetype, lineweight, block, thickness, linetype scale, entity type, etc.
WC	Window Circle: Selects all entities within a circular area.

Using additional PTCAD commands

Although you can use the AutoCAD command structure with PTCAD, the program has its own set of commands. PTCAD has numerous command names not found in AutoCAD, although many of these commands have an equivalent function in AutoCAD. When you type the AutoCAD equivalent, the PTCAD alias system activates the correct command.

For example, the AutoCAD *offset* command is mapped to the PTCAD *parallel* command. You can type either *offset* or *parallel* in the command bar, or you can choose Modify > Parallel.

The following table lists optional PTCAD command names that you can use in place of AutoCAD command names to perform the equivalent AutoCAD function.

Command differences in PTCAD

PTCAD	AutoCAD	Action in PTCAD
audionote	No equivalent	Creates and plays audio notes attached to entities.
cmdbar	No equivalent	Positions the command window.
coordinate	Ctrl+D or F6	Changes the display of coordinates on the status line.
copyedata	No equivalent	Copies extended entity data from one entity to another.
customize	toolbar	Displays the Customize dialog box.
deledata	No equivalent	Deletes extended entity data from an entity.
delete	erase	Removes entities from the drawing.
dimension	dim	Switches to dimension mode.
editedata	No equivalent	Edits extended entity data found in an entity.
editlen	lengthen	Changes the length of open entities.
editpline	pedit	Edits polylines and polymeshes.
entprop	ddmodify and ddchprop	Displays the Entity Properties dialog box.
esnap	-osnap	Sets entity snaps from the command line.
expblocks	ddinsert	Displays Explorer - Blocks.
expdimstyles	ddim	Displays Explorer - Dimension Styles.
expfonts	style	Displays Explorer - Styles.

Command differences in PTCAD (*continued*)

PTCAD	AutoCAD	Action in PTCAD
exlayers	layer	Displays Explorer - Layers.
explorer	ddrename	Displays Explorer.
expltypes	linetype	Displays Explorer - Linetypes.
expucs	dducs	Displays Explorer - UCS.
expviews	ddview	Displays Explorer - Views.
face	3dface	Draws three-dimensional faces with three or four vertices.
flatten	No equivalent	Sets thickness to zero at user-specified elevation.
font	-style	Displays the Text Style dialog box.
freehand	sketch	Allows freehand sketching.
idpoint	id	Returns the x-, y-, and z-coordinates of a picked point.
inpline	xline	Draws a line of infinite length.
join	pedit join	Joins lines and arcs.
laymcur	ai_molc	Sets the current layer based on the selected entity's layer.
mesh	3dmesh	Draws a surface mesh.
moveedata	No equivalent	Moves extended entity data from one entity to another.
msnapshot	mslide	Makes an SLD, EMF, or WMF file of the current view.
orthogonal	ortho	Toggles orthogonal mode.
parallel	offset	Copies an entity by a parallel offset distance.
plane	solid	Draws a two-dimensional solid plane.
pmthist	F2	Switches between the Prompt History window and the graphics screen.
polyline	pline	Draws a polyline.
ppreview	preview	Previews the print.
print	plot	Prints the drawing.
printstyle	plotstyle	Assigns a print style.
psetup	No equivalent	Displays the Print Setup dialog box.
qprint	No equivalent	Quickly prints the current viewport (window) with no options.

Command differences in PTCAD (continued)

PTCAD	AutoCAD	Action in PTCAD
reassocapp	No equivalent	Reassociates extended entity data with an application.
recscript	No equivalent	Starts the Script Recorder.
rtrot	3dorbit	Rotates the view of entities.
rtrotx	No equivalent	Rotates the view of entities while maintaining the x-axis.
rtroty	No equivalent	Rotates the view of entities while maintaining the y-axis.
rtrotz	3dorbit	Rotates the view of entities while maintaining the z-axis.
saveall	No equivalent	Saves all currently open drawings.
setcolor	color	Displays the Color dialog box.
setdim	ddim	Displays the Dimension Styles Manager dialog box.
setesnap	osnap	Displays the Drawing Settings dialog box with the Coordinate Input tab displayed.
settings	No equivalent	Displays the Drawing Settings dialog box.
setucs	dducs	Displays the User Coordinate Systems dialog box.
setvpoint	No equivalent	Displays the Preset Viewpoints dialog box.
stopscript	No equivalent	Stops running the script.
tipofday	No equivalent	Displays the Tip of the Day.
undelete	oops	Restores the last deleted entity.
vba	vbaide	Opens the Visual Basic Applications editor.
viewctl	ddvpoint	Sets the three-dimensional viewing direction via an interactive dialog box.
viewpoint	vpoint	Sets the three-dimensional viewing direction via the command line.
vsnapshot	vslide	Displays an SLD, EMF, or WMF file in the current viewport.

Supporting AutoCAD customization

PTCAD supports AutoCAD .cui/.cuix files which are used to control the following aspects of the PTCAD user interface:

- Workspaces
- Menus
- Toolbars
- Ribbon, contextual ribbon states
- Quick Access toolbar
- Application button menu
- Keyboard shortcuts
- Aliases
- Double-click actions

Although PTCAD primarily utilizes .cui/.cuix files to control the user interface, PTCAD's Customize UI command also supports importing and exporting a variety of AutoCAD and PTCAD legacy interface file formats:

- MNU, MNS and ICM — Legacy file formats for storing menus.
- PGP and ICA — Legacy file formats for storing command aliases.
- ICK — Legacy file format for storing keyboard shortcuts

In addition to file formats that control the user interface, PTCAD supports a variety of additional external files supported by AutoCAD:

- LIN — Linetype file that contains linetype definitions, including complex linetypes with shapes and text.
- PAT — Pattern file that contains definitions for hatch patterns.
- MLN — Multiline pattern file.
- DWT — Drawing template file.
- BAK — Drawing backup file.
- FMP — Font mapping file.
- UNT — Unit translation file used by the LISP (cvunit) function.
- CFG — Graphics configuration file.
- ATC — Tool palette catalog.
- SHX, TFT — Font files.
- SHP — Shape file.
- ACB — Color book file.
- CHM — Help file.

- SLD — Slide file.
- STB — Standard named print style tables used for printing.
- CTB — Color defined print style tables used for printing.
- PC3 — Printer configuration file used for printing.
- DST — Drawing sheet set file used to describe the file set used by the Sheet Set Manager command.
- DSD — Drawing set description file that describes a set of drawings to be used by the Publish command.
- ICO, BMP, PNG — Image files supported for icons in the user interface.
- JPG, PGN — Image files supported for materials.
- LSP — PTCAD provides a LISP interpreter, however, not all methods are fully supported. Note that PTCAD cannot load and run FAS compiled LISP code.

Understanding menu compatibility

MNU files are menu files created by all AutoCAD releases, and MNS files are included in AutoCAD Releases 13, 14, and 2000. PTCAD reads both file formats, even when menu macros include AutoLISP code. This feature allows you to continue using your existing AutoCAD menus

PTCAD support of specific sections in AutoCAD MNU and MNS files

Menu section	Definition	PTCAD support
***POP0	Cursor menu	Supported
***POP n	Pull-down menus	Supported
***AUX n	Auxiliary menus	Not supported
***BUTTON n	Button menus	Not supported
***ICON	Icon menus	Not supported
***SCREEN	Screen menus	Not supported
***TABLET n	Tablet menus	Not supported

Importing and exporting customization files

You can continue using aliases and menu files from AutoCAD by importing the appropriate file. You can import AutoCAD customization files and export PTCAD formats using the Customize dialog box. All of the files listed in the following table are in ASCII format, which means you can view and edit them with a text editor, such as Notepad.

Customizing files

Customization	Import file formats	Export file formats
Aliases	PGP: AutoCAD aliases ICA: PTCAD aliases	PGP: AutoCAD aliases
Keyboard	ICK: PTCAD keyboard	ICK: PTCAD keyboard
Menus	MNU: All AutoCAD menus MNS: AutoCAD R13, R14 and 2000 menus ICM: PTCAD menu	ICM: PTCAD menu
Toolbars	MNU: All AutoCAD menus	None

TIP Use the *Customize UI* command to customize the user interface. For more information, see “Working with user interface customization files” on page 916.

Programming PTCAD

PTCAD supports several AutoCAD application programming interfaces (APIs), but not all of the AutoCAD APIs are available in PTCAD. The following table summarizes the AutoCAD APIs PTCAD supports.

PTCAD support of the AutoCAD programming interface

AutoCAD programming interface	PTCAD support
Scripts (SCR files)	Supported
AutoLISP (LSP files)	Partially supported
Dialog Control Language (DCL files)	Supported
AutoCAD Development System (ADS)	Supported; source code must be recompiled
Visual Basic Applications (VBA)	Supported, depending on your version of PTCAD
Direct Interactively Evaluated String Expression Language (Diesel)	Supported
AutoCAD SQL Interface (ASI)	Not supported
Autodesk® Threaded Language Application System Toolkit (Atlas)	Not needed
AutoCAD Runtime Extension (ARx)	IRX Runtime Extension, custom objects supported

Understanding AutoLISP compatibility

PTCAD adds LISP functionality that you will find useful. The following table lists functions that are unique to PTCAD LISP.

Additional PTCAD LISP functions

Unique LISP function	Definition
(log10)	Returns log base 10.
(lpad)	Pads a text string with spaces to the left.
(rpad)	Pads a text string with spaces to the right.
(tan)	Returns the tangent.
(trim)	Trims spaces from a string.

Not all PTCAD LISP functions are completely compatible with AutoLISP functions. The following table identifies PTCAD LISP functions that are partially compatible with AutoLISP functions.

Partially compatible LISP functions

LISP function	Description
(menucmd)	Supports <i>P0</i> (cursor menu), <i>P1</i> through <i>P16</i> (the pull-down menus), and <i>M</i> (diesel expressions), but does not support <i>A</i> (aux menus), <i>B</i> (button menus), <i>I</i> (icon menus), <i>S</i> (screen menu), or <i>T</i> (tablet menus).
(print1)	Does not support Unicode characters, such as \U+00B0 (the degree symbol) and M+Nxxxx (multi-byte Unicode sequences).
(ssget) and (ssadd)	Supports additional selection modes: CC = Crossing Circle O = Outside OC = Outside Circle OP = Outside Polygon PO = POint PRO = Properties

In addition, PTCAD LISP does not support all AutoLISP functions. The following areas are not supported by PTCAD LISP:

- (acdimenableupdate), (acet-attsync), (acet-layerpmode), (acet-layerp-mark), (acet-laytrans), (acet-ms-to-ps), (acet-ps-to-ms), (defun-q), (defun-q-list-ref), (defun-q-list-set), (entmakex), (initdia), (namedobjdict), (ssnamex), and (tablet)
- ARX-related functions that run ARX applications
- (dict)-related functions
- SQL-related functions that link AutoCAD entities with external database records. These functions start with ase_, for example, (ase_lsunite) and (ase_docmp)

PTCAD and AutoCAD list of terms

List of terms

PTCAD term	Meaning for AutoCAD users
command bar	command prompt area
edit length	lengthen
entity	object
entity snap	object snap
entity snap precision	aperture
fixed attribute	constant attribute
follow	continue
freehand	sketch
hidden attribute	invisible attribute
infinite line	XLine
insert	draw
orthogonal	ortho
parallel	offset
plane	solid (2D)
predefined attribute	preset attribute
print	plot
print style	plot style
prompt box	context menu
Prompt History window	text screen
reference grid	grid
point snap	node snap
snapshot	slide (SLD)
validate attribute	verify attribute



Glossary

2D Two-dimensional locations defined by x- and y-coordinates.

3D Three-dimensional locations defined by x-, y-, and z-coordinates.

3D solids Three-dimensional ACIS entity.

absolute coordinates Coordinates defined in relation to the origin point of the current user coordinate system. *See also* **coordinate system**, **coordinates**, **origin**, **relative coordinates**, **user coordinate system**, and **World Coordinate System**.

ActiveX A mechanism for exchanging information between different programs whereby a copy of a source document is embedded or a pointer to a source document is linked to a target document. *See also* **embed** and **link**.

acute angles Angles of fewer than 90 degrees.

alias An abbreviation or alternative word for an PTCAD command.

aligned dimension A dimension aligned parallel to an entity or measuring the distance between two points at any angle.

angle The difference in direction between two nonparallel linear entities, measured in degrees or radians.

angular dimension A dimension measuring the angle between two lines or subtended by an arc.

angular unit The unit of measurement for angles. Angular units can be measured in decimal degrees, degrees/minutes/seconds, grads, and radians.

annotation Any text, dimensions, tolerances, or notes added to a drawing.

ANSI Acronym for American National Standards Institute. In the context of text, a standard character set defined by ANSI used in computer-aided drafting.

arc A segment of a circle or ellipse.

area Measurement of a planar region or the calculated space within an entity.

array Multiple copies of selected entities in a circular or rectangular pattern.

ASCII Acronym for American Standard Code for Information Interchange, a commonly used system for assigning numbers to printable alphanumeric characters, punctuation, and symbols.

attribute A component of a block containing specific text or numeric information. You can copy the information contained in an attribute from the drawing to an external database.

attribute definition An entity composed of a name, prompt for information, display characteristics, and default text that, when incorporated into a block, creates an attribute when the block is inserted into a drawing.

attribute name Text that identifies an attribute within a block.

attribute text The text containing an attribute's information within a block.

B-spline curve *See spline.*

base point A point on an entity that serves as its reference or insertion point. A point of reference when specifying relative distances.

baseline The line on which text characters appear to sit. The descenders of individual characters drop below the baseline.

baseline dimension Multiple parallel dimensions measured from the same baseline origin.

bind To convert an externally referenced drawing into a standard block definition.

blips Temporary screen markers displayed in a drawing when you select a point. Also called *marker blips*.

block One or more entities grouped together to create a single entity. *See also nested block.*

block definition The name, base point, and entities grouped together when creating a block.

boundary A selected area bounded by a single closed entity or by multiple entities that intersect.

BYBLOCK A property whereby an entity inherits the color, linetype, lineweight, or print style of any block that contains it.

BYLAYER A property whereby an entity inherits the color, linetype, lineweight, or print style of its associated layer.

CAD Acronym for computer-aided design.

Cartesian coordinates Coordinates defined using three perpendicular axes (*x*, *y*, and *z*) to define locations in three-dimensional space. *See also cylindrical coordinates, polar coordinates, and spherical coordinates.*

center line A line used to indicate the center of a circle or an arc, usually consisting of a center mark and lines extending slightly beyond the diameter of the circle or the arc.

center mark A cross marking the center of a circle or an arc.

- chamfer** A beveled edge between two lines.
- chord** A line connecting two points on a circle or an arc.
- circumference** The measurement of the distance around a circle.
- closed** A condition whereby the start point and endpoint of an entity are the same.
- color-dependent print style table** A collection of print styles that determine how entities print according to their assigned color. *See* **print style table**.
- comma-delimited** Data that is separated by a comma to represent the end of a field.
- command bar** A dockable window in which you type PTCAD commands and view prompts and other program messages.
- cone** A three-dimensional entity where a vertex exists above or below the circular shape and where a surface has been applied between the vertex and the circular shape.
- contiguous** Connected, unbroken, or uninterrupted. Entities that share the same endpoint.
- continued dimension** A dimension measured from the previous extension line of an existing dimension, resulting in two or more dimensions positioned end to end.
- control vertex** A point used to define a spline.
- Coons patch** A surface interpolated among three or four boundary curves.
- coordinate filter** A function that extracts individual x-, y-, and z-coordinate values from different points to create a new composite point.
- coordinate system** A system of points that represents the drawing space in relation to an origin (0,0,0) and a set of axes that intersect at the origin. In two dimensions, the x- and y-axes represent horizontal and vertical directions, respectively. In three dimensions, the z-axis represents locations above and below the two-dimensional xy plane. Locations in the drawing can be represented using two-dimensional and three-dimensional rectangular (Cartesian) coordinates, two-dimensional polar coordinates, three-dimensional polar (cylindrical) coordinates, and three-dimensional spherical coordinates. *See also* **polar coordinates, relative coordinates, spherical coordinates, user coordinate system, and World Coordinate System**.
- coordinates** A set of values that determines a location in two-dimensional or three-dimensional space. *See also* **absolute coordinates, Cartesian coordinates, polar coordinates, relative coordinates, and spherical coordinates**.
- coplanar** Lying within the same plane.
- crosshairs** A cursor that consists of two or three lines that intersect at the cursor location.
- crosshatch** To fill an area with a pattern of evenly spaced perpendicular lines. *See also* **hatch**.

crossing circle An entity-selection method that selects entities contained within or crossing the boundary of a circular selection window.

crossing polygon An entity-selection method that selects entities contained within or crossing the boundary of a polygon selection window.

crossing window An entity-selection method that selects entities contained within or crossing the boundary of a rectangular selection window.

cube A boxed, three-dimensional, geometric shape where length, width, and height are equal.

cursor The insertion-point symbol on the screen. The appearance of the cursor changes based on the current task.

curve A smooth, continuous path made up of linear and arc segments. Curve types include arcs, splines, circles, and ellipses.

cylindrical coordinates Coordinates describing a point in three-dimensional space based on its distance from the origin, its angle in the xy plane, and its z-coordinate value. *See also* **polar coordinates** and **spherical coordinates**.

datum-line dimensioning *See* **ordinate dimension**.

default An initial or predefined setting.

detach To remove an external reference from a drawing. *See also* **external reference**.

diameter The distance across a circle or sphere.

digitizer tablet A hardware input device that incorporates an electronic pad and a hand-held pointer similar to a mouse. A digitizer tablet serves two purposes: (1) You can select tools from paper representations attached to the pad (called an overlay), providing access to all tools at once while freeing your screen space; and (2) you can input digital points into the computer that correspond to points on a paper drawing, photograph, or blueprint attached to the pad.

dimension A measurement, as in height or width. In the context of drafting, a set of lines, arrowheads, and text used to indicate a measurement.

dimension style A named group of dimension variable settings that determines the appearance of the dimension. You can save multiple dimension styles for reuse.

dimension text The measurement value. Dimension text can include prefixes, suffixes, tolerances, and other annotations.

dimension text rotation The angle in degrees between the x-axis and the dimension text baseline.

dimension tolerance A value specifying the allowed variation of a dimension (+ or - *n*).

dish The lower half of a sphere. *See also* **dome**.

displacement point The point to which a base, or reference, point will be relocated when moving or copying entities.

distance The measure of space between two points.

dock To position a toolbar or the command bar at the edge of the drawing window, where it locks into place. *See also float.*

dome The upper half of a sphere. *See also dish.*

donut A filled circle or flat ring created as a polyline.

drawing extents *See extents.*

drawing limits *See limits.*

drawing unit The linear measurement system used in a drawing. The user determines what a drawing unit represents, such as one inch, one centimeter, one foot, or one meter.

DWF Acronym for Autodesk Design Web Format™, a file format for viewing two-dimensional or three-dimensional drawings in a Web browser and distributing for review using free Autodesk® software and tools.

.dwg A standard file extension used by CAD programs to store drawing files of the DWG format.

.dwt A standard format used by CAD programs to store drawing templates, which are drawings that contain predefined settings that you can use when creating a new drawing. *See also template.*

DXF Acronym for Drawing Exchange Format, a standard ASCII or binary file format for importing and exporting files between most CAD programs.

elevation The z value measured from the xy plane. Positive values are above the xy plane; negative values are below the xy plane.

embed A technique for exchanging information between different programs whereby a copy of the source document is stored in the target document. *See also ActiveX and link.*

EMF Acronym for Enhanced Metafile, a file format with the type and extension of .emf. It is a native internal file format of Windows 98. EMF supports both raster and vector information and 24-bit RGB color. Most Windows-based programs support this format.

endpoint The point at which a line or curve ends.

entity Any single basic item in a drawing. Entities include arcs, attributes, blocks, circles, dimensions, ellipses, elliptical arcs, infinite lines, lines, polylines, rays, and text.

entity data Any of a variety of additional information, such as text, numbers, and distances, that can be attached to drawing entities.

entity snap A technique for accurately locating and specifying key geometric points on entities, such as endpoints and midpoints of lines, and center points and tangents of arcs and circles.

entity snap override To disable or change an entity snap mode for a single input. *See also* **entity snap** and **running entity snap**.

Esnap *See* **entity snap**.

explode The conversion of a complex entity into its basic entities.

extension lines Lines extending away from an entity being dimensioned so that you can place the dimension line away from the entity. Also called *projection lines*.

extents The smallest rectangle that can contain all the entities in a drawing. Infinite lines and rays do not affect a drawing's extents. *See also* **limits**.

external reference A drawing that is linked to another drawing.

extrude The process of converting a two-dimensional entity into a three-dimensional entity by stretching (extruding) the two-dimensional entity along a straight path. Changing the thickness of a two-dimensional entity effectively extrudes it along its z-axis. *See also* **tabulated surface**.

face A planar surface defined by three or four points.

fence An entity-selection method that selects entities crossing a multisegmented line.

fillet An arc that smoothly connects the end of one line to another.

fit point A point used to define a spline.

float To position a toolbar or the command bar away from the edges of the drawing window where it can be moved independently. *See also* **dock**.

freeze To suppress the display of, and ignore the entities on, a specified layer when regenerating a drawing, thus accelerating the display of the drawing. *See also* **thaw**.

grid An adjustable, regularly spaced pattern of dots on the screen, used as an aid in drawing and aligning entities. The grid is not printed.

grip A small square displayed at key positions on an entity when the entity is selected that can be used to modify the entity by clicking and dragging.

halfwidth The distance from the center of a wide polyline to its edge.

hatch To fill a selected area either with lines, crosshatching, or a hatch pattern. *See also* **crosshatch**.

hatch pattern A pattern, often representing a material such as steel, wood, or sand, for filling selected areas.

hidden-line removal A visualization technique in which all lines that are hidden behind other entities or surfaces from your viewpoint are clipped or removed, giving the image the appearance of a solid entity.

horizontal dimension A linear dimension measuring a horizontal distance.

infinite line A line extending infinitely in both directions.

insertion point The point where you place an entity into a particular space. A reference point by which an entity is inserted in a drawing.

intersection The point where two entities meet or cross.

isometric drawing A drawing aligned with an isometric plane.

isometric plane One of three planes representing the left, right, or top sides of an implied three-dimensional, orthogonal entity. Snap and grid points are aligned with the plane to constrain drawings.

layer The computer equivalent of overlays used in manual drafting. A tool by which drawing components can be organized into related sets, such as plumbing, framing, and electrical systems of a house, each being drawn on its own layer.

layout Similar to a sheet of paper, a representation of how a drawing will look when printed.

layout viewport A window in a Layout tab (paper space) that displays all or a portion of a drawing's model space entities. *See also* **paper space** and **model space**.

leader A line leading from a feature of a drawing to an annotation.

limits The user-defined boundary of a drawing, defined by its lower left and upper right corner coordinates. *See also* **extents**.

limits tolerance Dimension text in which the measured dimension is replaced by the largest and smallest dimensions allowed, with the upper limit displayed above the lower limit. *See also* **tolerance** and **variance tolerance**.

linetype The appearance of a line, defined as a solid (continuous) line or as a pattern of dashes, dots, and blank spaces.

lineweight The width of a line, defined in millimeters or inches.

link A technique for exchanging information between different programs whereby the target document maintains a pointer to the original source document. Any changes to the source document are reflected in all target documents containing links to the source. *See also* **ActiveX** and **embed**.

LISP Acronym for List Processing Language, a computer language invented in the late 1950s by John McCarthy for use in artificial intelligence. Because it is interpreted rather than compiled, and is relatively straightforward, it is a convenient language for users to write routines to extend the set of commands and functionality of PTCAD.

lock Prevents unauthorized access to drawing layers.

M direction In a polygon mesh, the direction from the first to the second row. *See also N direction.*

macro In menus and toolbars, several commands grouped together as one. Also Visual Basic for Applications code.

major axis The longest distance across an ellipse, from one end to the other. *See also minor axis.*

marker blips Temporary screen markers displayed in a drawing when you select a point. Also called *blips*.

MDI Acronym for multiple-document interface. *See multiple-document interface.*

mesh A set of connected polygon faces approximating a curved surface.

minor axis The shortest distance across an ellipse, from one side to the other. *See also major axis.*

mirror To create a reverse-image copy of selected entities by reflecting the entities symmetrically about a line or plane.

model space The primary drawing area in which you create entities. *See also paper space.*

multiple-document interface The ability to view and work with different drawings simultaneously.

N direction In a polygon mesh, the direction from the first to the second column. *See also M direction.*

named print style table A collection of print styles that determine how entities print according to the print styles you create and assign to entities and layers. *See print style table.*

named view A saved view that can be recalled at a later time by specifying its name.

nested block A block contained as part of the definition of another block. *See also block.*

nonassociative hatch A hatch that is not associated with or linked to an entity.

oblique Geometric lines or planes that are not parallel or perpendicular.

offset *See parallel.*

OLE Acronym for Object Linking and Embedding. *See ActiveX.*

ordinate dimension A measurement of the horizontal (x-ordinate) or vertical (y-ordinate) distance from an established reference base point or datum.

origin The intersection point of the coordinate system axes. In a Cartesian coordinate system, the origin is the point at which the x-, y-, and z-axes intersect (the 0,0,0 coordinate).

orthogonal Having perpendicular slopes or tangents at the point of intersection.

orthogonal mode (ortho) A drawing mode in which the entity creation is constrained to parallel the horizontal and vertical axes relative to the current snap angle.

orthographic projection A drafting technique by which a three-dimensional item is described in two dimensions by showing it from various directions, most commonly front, top, and side views.

outside circle An entity-selection method that selects entities falling completely outside a circular selection window.

outside polygon An entity-selection method that selects entities falling completely outside a polygon selection window.

outside window An entity-selection method that selects entities falling completely outside a rectangular selection window.

pan To shift the displayed view of a drawing without changing the magnification. *See also zoom.*

paper space A two-dimensional work area similar to a sheet of paper, in which you can arrange different views of your model as layout viewports. *See also model space.*

parallel Two or more coplanar lines that never intersect one another.

parallel dimension *See baseline dimension.*

PDF Acronym for Portable Document Format. PDF files can be viewed using Adobe® Acrobat Reader, which is free software that users can download; they can also be viewed, reviewed, and edited using Adobe® Acrobat.

perimeter The distance around the boundary of an entity.

perpendicular Entities at right angles to one another.

planar Entities whose extents are situated entirely within a plane.

plane Two-dimensional surface.

plan view A view of the drawing from above, looking down the z-axis perpendicular to the xy plane of the current UCS.

point A location in space specified by its x-, y-, and z-coordinates. A drawing entity consisting of a single x,y,z-coordinate location and represented by one of several symbols.

point filter *See coordinate filter.*

polar array Multiple copies of selected entities in a circular pattern.

polar coordinates Coordinates describing a two-dimensional point on a two-dimensional plane based on the point's distance from the origin and its angle in the plane. *See also* **Cartesian coordinates**, **coordinates**, **cylindrical coordinates**, **relative coordinates**, and **spherical coordinates**.

polygon A closed single entity with three or more sides.

polyline A drawing entity composed of one or more connected line or arc segments treated as a single entity.

print style A collection of settings, including color, pen width, linetype, and line-weight, that determine how a drawing looks when it is printed. *See* **print style table**.

print style table A collection of print styles that you can assign to the Model tab or to a Layout tab. Print style tables change how a drawing looks when you print it without modifying the actual entities. *See* **print style**.

profile File that contains your preferred drawing environment settings.

projection lines *See* **extension lines**.

prompt box A list of options displayed when a command or tool provides several choices.

Prompt History window A window containing a history of the most recent commands and prompts issued since you started the current session of PTCAD.

quadrant One-fourth of a circle, arc, or ellipse entity. In the context of entity snaps, the option that snaps to points on a circle, arc, or ellipse at each quadrant.

radial dimension A dimension that measures the radius of a circle or arc.

radian A unit of angular measurement; 360 degrees equals 6.283185 or 2π radians.

radius The distance from the center of a circle or sphere to its periphery.

ray A line that starts at a designated point and runs infinitely.

ray tracing A visualization technique in which rays from imaginary light sources are traced as they refract off the surfaces of a model, determining where shadows fall and how reflections on shiny materials such as metal and glass appear.

rectangle A four-sided, closed entity whereby opposite sides are equal in length.

rectangular array Multiple copies of selected entities in a rectangular pattern consisting of a specified number of columns and rows.

redo To reverse the effect of previous undo commands. *See also* **undo**.

redraw To quickly update or refresh the drawing screen display. *See also* **regenerate**.

regenerate To update or refresh the drawing screen display by recalculating the drawing from its database. *See also* **redraw**.

region A two-dimensional closed, surfaced, planar boundary.

relative coordinates Coordinates expressed in relation to a previous coordinate. *See also absolute coordinates.*

render A visualization technique in which all surfaces of a model are shaded as though they were illuminated from an imaginary light source located behind you as you face the screen. Rendered images are photo-realistic, having depth, shadow, reflection, and texture.

revolve Creating a three-dimensional surface entity by rotating a two-dimensional profile around an axis.

right-hand rule A visual aid for remembering the relative directions of the positive x-, y-, and z-axes of a Cartesian coordinate system and the positive rotation direction about an axis.

rotate To change the orientation of an entity, without modifying it, by repositioning it equidistant from, but at a new angle with respect to, a point or axis.

rotation angle The angle by which an entity is displaced from its original location when rotating it about a point or axis.

rubber-band line A ghosted image line that stretches dynamically on the screen with the movement of the cursor. The line extends between a fixed point and the cursor position to provide dynamic feedback.

ruled surface A three-dimensional polygon mesh that approximates a smooth surface between two entities.

running entity snap Setting an entity snap so that it continues for subsequent selections. *See also entity snap and entity snap override.*

scale To resize an entity. To draw according to the proportions of an entity.

script A set of commands stored in an ASCII script file and replayed in sequence by running the script.

SDS Acronym for Solutions Development System, a C programming interface for developing specialized programs to run inside PTCAD.

segment Any part of an entity bounded by two points.

selection set One or more drawing entities selected on which one can operate as a single unit.

shade To fill planar entities with solid colors for easier visualization.

snap angle The angle around which the snap grid is rotated.

snap grid An invisible grid that locks entity creation to a specified alignment and snap increment when Snap is enabled.

snap resolution The spacing between points on the snap grid.

snapshot A raster representation of the current view of one's drawing.

spherical coordinates Coordinates describing a point in three-dimensional space based on its distance from the origin, its angle in the xy plane, and its angle up from the xy plane. *See also* **Cartesian coordinates**, **coordinates**, **cylindrical coordinates**, and **polar coordinates**.

spline A curve generated along the path of three or more control points. The curve passes through the start point and endpoint, but does not necessarily pass through the intermediate control points.

status bar The bar at the bottom of the PTCAD window that displays information about the selected command or tool as well as the cursor coordinates, the name of the current layer, mode settings, and other information about drawing settings.

surface model A three-dimensional model consisting of both edges and the surfaces between those edges. *See also* **wire-frame model**.

surface of revolution A three-dimensional polygon mesh that approximates the surface generated by rotating a two-dimensional profile around an axis.

SVG An acronym for Scalable Vector Graphic, which is a file format for working with interactive graphics, including a Web development language.

system variable A setting or value that stores operating environment and command information (such as the drawing limits or global linetype scale factor).

tabulated surface A three-dimensional polygon mesh that approximates the surface generated by extruding a curve along a vector. *See also* **extrude**.

tangent A line that passes through a single point on a curve.

template A drawing with preset layers, linetypes, and other settings (and entities) that can be used as the basis for creating a new drawing. Templates are saved with a .dwt file extension.

text style A named, saved collection of format settings that determines the appearance of text.

thaw To redisplay a layer that was frozen. *See also* **freeze**.

thickness An entity's depth, as measured along its z-axis. The distance an entity is extruded above or below its elevation. *See also* **elevation** and **extrude**.

through point In creating a parallel entity, a point through which the new entity passes.

tolerance Dimension text indicating how much the actual dimension of a manufactured component can vary from the specified dimension. *See also* **limits tolerance** and **variance tolerance**.

tolerance command A command that creates a feature-control frame used in mechanical geometric dimensioning and tolerancing.

toolbar A collection of tools arranged on a palette that can be moved and resized anywhere on the screen.

torus A donut-shaped, three-dimensional entity.

transparent command A command started while another command is already active. You can use a command transparently by preceding it with an apostrophe.

true color Colors defined using 24-bit color. There are more than 16 million true colors from which you can choose.

UCS Acronym for user coordinate system. *See* **user coordinate system**.

UCS icon A user coordinate system icon that shows the orientation of the coordinate axes, the location of the coordinate system origin, and the viewing direction relative to the xy plane.

undo To reverse the effect of previous commands. *See also* **redo**.

unit *See* **drawing unit**.

unlock Free access to layers in a drawing that would be otherwise locked, thus prohibiting them from being viewed or edited by another user.

user coordinate system A Cartesian coordinate system with origins and orientation defined by the user. *See also* **World Coordinate System**.

variance tolerance Dimension text in which a plus/minus value is appended to the specified dimension to indicate how much the actual dimension of a manufactured component can vary from the specified dimension. *See also* **limits tolerance** and **tolerance**.

VBA Acronym for Visual Basic for Applications, a macro programming language embedded in programs that allows the user to customize the program.

vector A means of describing a displacement using magnitude and orientation. For example, you can create a line entity, or move an entity, by specifying an initial point, a direction, and a distance.

vertex The point of intersection of the sides of an angle. The start points or endpoints of a line or arc segment in a polyline.

vertical dimension A linear dimension measuring a vertical distance.

view A representation of a drawing or portion of a drawing from a specific viewpoint in three-dimensional space.

viewpoint A location in three-dimensional space for viewing one's drafting.

viewport A window that displays all or a portion of a drawing's model space entities while in model space on the Model tab. *See also* **model space** and **paper space**.

viewport configuration A named arrangement of windows that can be saved and restored.

WCS Acronym for World Coordinate System. *See* **World Coordinate System**.

wedge A three-dimensional entity that resembles a box divided along one side from one corner to the opposite corner; for example, a doorstep or a ramp.

window circle An entity-selection method that selects entities contained entirely within a circular selection window.

window inside An entity-selection method that selects entities contained entirely within a rectangular selection window.

window polygon An entity-selection method that selects entities contained entirely within a polygon selection window.

wipeout An entity that displays with the current background color, so the details behind it do not display or print.

wire-frame model A three-dimensional model consisting of lines and curves that define the edges of a three-dimensional entity. *See also* **surface model**.

WMF Acronym for Windows metafile, a format containing vector and color information to render entities.

World Coordinate System The fixed Cartesian coordinate system used as the basis for defining other coordinate systems. *See also* **user coordinate system**.

xref *See* **external reference**.

zoom To increase or decrease the magnification of the display of a drawing. *See also* **pan**.

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