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## Getting Started

Welcome to TotalCAD for Windows 1.0, the world's best value in desktop CAD software. CAD, or Computer Aided Design, is a powerful tool for creating geometrically precise drawings. TotalCAD 1.0 is designed for value, speed, and ease of use. It is the perfect tool to help you learn and use CAD.

The following topics deal with how to get the information you need to learn and use TotalCAD:

[Using the Help System](#)

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## Using the Help System

This online help system provides a complete reference to TotalCAD's features. To learn more about the features of the Windows help system, choose from the list of topics:

[Context-Sensitive Help](#)

[Using Windows Help](#)



## **Context-Sensitive Help**

If you need particular information about a tool, command, or dialog, you can go directly to the relevant help topic from the TotalCAD interface. Or, you can click on the Help tool, then click on the tool, menuitem, or interface element about which you want information.

You can also place the mouse cursor over a tool, or highlight a menu item, then press the <F1> key.

## **Tooltips**

Tooltips are a convenient way to identify a tool button. Just position the mouse cursor over a tool for a second, and small yellow box containing the name of the tool will appear.

## Running the Tutorial

TotalCAD's electronic tutorial provides step-by-step, hands-on instructions that teach you how to create drawings in TotalCAD. The tutorial stays on top of your TotalCAD window so that you can draw in TotalCAD while following the instructions. To start the Tutorial, choose Tutorial from the Help menu. The Tutorial Topics page will appear.

You can go to any tutorial topic by clicking on the topic.

Once you are in a topic, click on the buttons at the top of the tutorial screen to navigate. At the bottom of many of the tutorial topics, you will find a "Show Me" button. Click on this button to see what your drawing should look like at any time during the tutorial.





## **Tips of the Day**



TotalCAD's Tips of the Day are a collection of advice, techniques, and tricks that could make your work with TotalCAD easier, point out features that are not so obvious, or show you how to do things that you perhaps didn't know you could do. Some of the tips are directed toward novice users, and others are directed to more experienced users.

The first time you start TotalCAD, you will see a Tip of the Day. You can stop the tips from appearing at startup by clearing the Show Tips at Startup check box. If you subsequently want to turn the tips back on, choose Tip of the Day from the Help menu and click the Show Tips at Startup check box.

## Contacting Technical Support

If you can't find the answer to a question in the manual or in this Help system, the IMSI Technical Support staff is there to help you. You must be a registered TotalCAD user to get technical support. Technical support by e-mail and the World Wide Web is provided free of charge.

Before you call, have your computer turned on. If you have a question about how something in TotalCAD works, then you only need to have the first three of the following items available. If you are having a system-related problem, then please try to have all the following information available:

- Your name, address, and phone number.
- Product name and version number (choose Help|About).
- Computer make and model.
- A precise description of how to reproduce the problem, including any error messages.
- The version of Windows that you are running.
- Amount of RAM installed and type of memory manager.
- Type and capacity of your hard disk.
- Graphics adapter and monitor type.
-  Printer/plotter make and model.
-  List of any special peripherals installed.

To contact technical support by phone, dial (415) 257-3000, extension 3. You can also send your request by e-mail to [support@imsisoft.com](mailto:support@imsisoft.com). You may also be able to find the answer you need by checking out the TotalCAD support pages on the World Wide Web, at <http://www.imsisoft.com/techsupp/>.

## Quick Tour

The following will take you on a quick tour through TotalCAD's basic features. It will cover the following topics:

[Choosing a Template](#)

[TotalCAD Desktop](#)

[Setting up your Desktop](#)

[Getting around the Interface](#)

## Choosing a Template

When you start a new drawing, the Create from Template dialog appears. This is where you choose a standard template to use as the basis of your drawing. TotalCAD comes with a number of templates (stored in the Templates folder), with standard settings for different types of drawing disciplines and preferences.

Once you choose a template a [Wizard](#) may run (depending on which template you choose), which will help you complete the setup of your drawing. One very useful template is WzNormal.tct. This template begins with a [Setup Wizard](#), which will prompt you to set up your drawing. It will ensure that hatches and double lines are correctly proportioned.

You can also create your own templates by simply saving a drawing as a template using the Save As dialog. A template contains all of the information stored in a drawing, including units and scale, grid setup and symbol libraries.



You can keep this dialog from appearing by removing the check mark from the Prompt for Template Name option in the [General Property Sheet of the Program Setup dialog](#).

[Create from Template Dialog](#)




## Wizards

Some of TotalCAD's templates and scripts begin with wizards. Wizards are like dialogs that contain several pages of options that you complete sequentially. Choose settings on each page, then click the Next button. When you are finished with the last page, click the Finish button. For more information, see [Setup Wizard](#).

## Setup Wizard

The Setup [Wizard](#) appears when you start TotalCAD using the WzNormal template.

The Setup Wizard contains three pages which will help you:

-  Choose your [drawing units and number format](#)
-  [Set up your page and enter user and drawing information](#) that will be saved with your drawing file.
-  Choose your [drawing Scale](#).



## Drawing Units and Number Format

The first page of the WzNormal template's [Setup Wizard](#) contains the following options for formatting your drawing units and number format. As you choose different options, an example of the unit and number format settings will appear in the Samples box.

*English and Metric.* Choose one of these settings, depending on whether you want to work in the English (inches and feet), or Metric measuring system.

*Format Category.* Click in this pulldown list to choose a general category for the number format of your rulers and dimensions. You can choose Architectural, Decimal, Engineering, Fractional, or Scientific number format. The following table shows examples of the available number formats:

<b>Format</b>	<b>Example</b>
Decimal	3.25
Scientific	1E+01 mm
Fractional	3 3/16
Architectural (Feet in Fractional)	1'-5 11/16"
Engineering (Feet in Decimal)	1"= 200'

*World Units.* Click in this pulldown list to choose a basic unit of measurement. The choices in this box will vary, depending on other options you have chosen.

*Decimal Precision.* Use this control to set the precision of decimal numbers (i.e. how many digits appear to the right of the decimal point).

Note that you can reset these options for your drawing using the [Units and Scale](#) property sheet.

## Paper Setup and Drawing Info

The second page of the WzNormal template's [Setup Wizard](#) contains the following options for setting up your page and entering drawing information.

*Paper Size Options.* Use this control to either let TotalCAD choose a default paper size, or choose a custom paper size from the pulldown list.

*Paper Orientation.* Use this control to choose either a portrait (tall) or landscape (wide) paper orientation.

*User/Drawing Information.* Use these control to enter specific information about your drawing and the author of the drawing.

Note that you can reset the options contained in this page of the wizard using the [Page Setup](#) and [Summary Info](#) dialogs.

## Drawing Scale

The third page of the WzNormal [Setup Wizard](#) contains the following options for setting your drawing scale, that is, the ratio of the size of your drawing to the size of the real-world objects that your object is modelling. Check the Samples box to see how your choices will affect scale display.

*Scale Type.* The scale type will determine how your scale is displayed. If you are using metric units, only the Absolute scale type will be available. If you are using English units, you can choose an Absolute, Architectural, or Engineering Scale type. Check the Samples box to see examples of each type as you change settings.

*Scale.* Use this box to choose an actual scale (the ratio shown in the type and format you have chosen).

*Let TotalCAD choose a scale.* Choose this option if you want TotalCAD to choose a scale based on the dimensions of the object you want to draw. Type the height and width of the object you are drawing into the Height and Width boxes.

Note that you can reset the options contained in this page of the wizard using the [Units and Scale](#) property sheet.

## TotalCAD Desktop

Once you have started TotalCAD and chosen a template, you will see the TotalCAD desktop with a new, blank drawing. The following topics provide brief descriptions of the TotalCAD interface components on your screen.

[Menu Bar](#)

[Toolbars](#)

[Flyout Toolbar](#)

[Rulers](#)

[World/Paper Button](#)

[Scroll Bars](#)

[Drawing Area](#)

[Paper](#)

[Edit Bar](#)

[Local Menu](#)

[Coordinate Fields](#)

[Status Bar](#)

## **Menu Bar**

The menu bar is where you access TotalCAD's menu system. To choose a command from the menu, click on one of the menu items on the menu bar, then click again to choose a command from the menu that appears below. If you choose a submenu from the menu, a secondary menu appears, from which you can choose a command.

## Toolbars

Toolbars provide a convenient alternative to the menu system. The TotalCAD default screen displays four toolbars: the Standard and Insert toolbars at the top of the screen, and the Snaps and View toolbars on the left and right sides of your screen, respectively. Clicking on any of the buttons on the toolbars will give a TotalCAD command. You can access many more toolbars by choosing Toolbars from the View menu, displaying the Desktop Property Sheet.

The position of toolbars is configurable: just drag the toolbar by its edge the top of the screen to position it horizontally, to the side of the screen to position it vertically, or position it as a floating palette over the drawing area.



**See Also**

Desktop Property Sheet

## **Flyout Toolbars**

Flyout toolbars gives you convenient and intuitive access to all the drawing tools in a way that takes up minimal screen space. Tools that have flyouts have a small yellow triangle in their lower-right corner. To access a flyout tool from the Insert toolbar:

1. Make sure the Insert toolbar is visible on your screen. If it isn't, use the View|Toolbars dialog box to display it.
2. Choose the type of tool you need. To do this, place the mouse cursor over a tool button and hold the left mouse button down. A *flyout* toolbar will appear under the mouse cursor.
3. Click to choose the specific tool that you want from the flyout.



## Rulers

The Rulers gives you feedback about the size and position of objects in your drawing. The ruler can display either World Units or Paper Units. For example, you may be drawing a picture of a table that is five feet long, while the drawing that represents the table is five inches long. Feet, in other words, are the drawing's *World Units*; inches are the drawings *Paper Units*.



[See Also](#)

Units and Scale Property Sheet

## **World/Paper Button**

This button, in the upper-left corner of the ruler, lets you change the ruler between [Paper Units](#) and [World Units](#). Click on the button, then observe how the ruler is changed.

## **Scroll bars**

By clicking in the scroll bars, you can "scroll" to a different view of your drawing. Click on the arrow to scroll a small amount. Hold the mouse cursor over the "thumb" of the scroll bar (the rectangle in the middle of the bar) and drag it to move through your drawing. Click in the area between the thumb and the arrow to scroll a larger amount.

## **Drawing Area**

The drawing area is the area in the middle of your screen in which your drawing objects appear. You can increase the size of your drawing area by turning off interface elements that you don't use. You do this through the [Desktop Property Sheet](#).

## **Paper**

The Paper, the large white rectangle in the middle of the screen, shows you how your drawing will be layed out on the page when printed. You don't have to draw on the Paper; you can draw anywhere in the drawing space. You can change the layout of the paper using the [Page Setup dialog](#). You can display and hide the paper using the [Desktop Property Sheet](#).

## Edit Bar

The Edit Bar allows you to draw by specifying exact quantities, such as lengths and angles. You might, for example, use the Edit Bar to draw a line that is exactly two inches long, at a 45 degree angle; or a circle with a radius of exactly 1.75 feet.

The position of the Edit Bar is configurable. You can place it horizontally along the top of the screen, or position it over the drawing area as a floating palette. You can't position it vertically. To reposition the Edit Bar, place the mouse cursor at any point just inside the edge of the Edit Bar and drag the it to a new location.

The Edit Bar contains one or more fields (depending on the active tool). To enter a value in a field, simply click in the field to activate it, then type the value into the field. You can also move between fields of the Edit Bar by pressing <Tab> or <Shift>+<Tab>. To accept the values in the Edit Bar, press <Enter>.

Many Edit Bar fields have lockdown boxes to their immediate left. Checking a lockdown box makes the value in the field insensitive to mouse movements in the drawing area. This means that you can fix settings in one or more fields, then activate the drawing area and move the mouse without changing the values in the locked fields.



**See Also**

[Finishing and Canceling](#), [Defining Points and Specifying Measurements](#), [Drawing with the Edit Bar](#)



## Local Menu

To see the local menu, right-click in the drawing area. The local menu, like the Edit Bar, is context-sensitive. The menu items available in the menu change, depending on the active drawing or editing tool, and sometimes even change depending on the position of the mouse cursor relative to an entity. You can use the local menu, among other things, to cancel the current operation, finish the current operation, set temporary snap modes, and undo the previous operation.



**Tip**



**See Also**

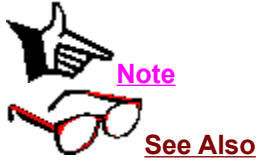
Because the Local Menu is context-sensitive, it's not always obvious what functions it may contain in a given situation. While learning TotalCAD it's a good idea to check the contents of the Local Menu when you are using different tools and editing techniques. You may be surprised by what you find!

[Editing with the Mouse and Local Menu](#), [Finishing and Canceling](#), [Local Snap](#), [Snap Modes](#)

## Coordinate Fields

Unlike the Edit Bar and toolbars, the Coordinate Fields are fixed on the Status Bar and located on the lower-right corner of the screen. The Coordinate Fields provide a way of defining points by specifying their exact coordinates.

To define a point using the Coordinate Fields, type the coordinates into the fields and press <Enter>.



The types of coordinates that you enter will depend on the currently active coordinate system (absolute, relative, or polar).

Coordinate Systems, Defining Points with the Mouse

## Status Bar

The Status Bar is displayed across the bottom of your screen. On its left it displays a brief message that lets you know what action TotalCAD expects next. On the right side of the Edit Bar are the [Coordinate Fields](#), as well as an indicator that tells you the zoom percentage, which is how far in or out of the drawing your view is zoomed. 100% zoom shows your drawing at its printed size.

You can display or hide the Status Bar using the [View|Statusbar](#) command.

## Setting up your Desktop

The following topics discuss how to hide and display TotalCAD's screen components, how to position toolbars and the Edit Bar, and how to save your desktop settings for the next time you run TotalCAD.

[Choosing What You See On The Screen](#)

[Positioning Toolbars](#)

[Saving your Setup](#)



## Choosing What You See On The Screen

You can customize your screen in TotalCAD using the commands of the [View menu](#), as well as the [Desktop property sheet](#). You can also move the toolbars and the Edit Bar to positions that you prefer, and save your screen setup for the next time you start TotalCAD.

## **Positioning Toolbars**

You can move a toolbar or the Edit Bar by positioning the mouse cursor over the edge of the control and dragging it to a new position. Most toolbars can be positioned horizontally at the top of the screen, vertically along the side of the screen, or positioned over the drawing as floating palettes. The Edit Bar and toolbars that contain text fields cannot be placed vertically.

## **Saving your Setup**

If you want to return to a customized setup the next time you run TotalCAD, you need to make sure that TotalCAD saves the Desktop settings when it exits. To do this:

1. Select Tools|Program Setup. The Program Setup dialog appears.
2. Check the Save Desktop on Exit box.

Now when you exit TotalCAD, the desktop configuration of toolbars and other controls will be saved. If you don't want to save your desktop settings, make sure that the Save Desktop on Exit box is not checked before you exit.

## Getting around the Interface

When you are drawing and editing, you will need to move back and forth between TotalCAD's various controls, sometimes in the middle of an operation. You may start an entity by defining a point with the mouse, then continue by defining a point in the Coordinate Fields, then set values in the Edit Bar, then return to the drawing area to complete the operation. You may even want to set up a construction line, zoom into an area of the screen, change the coordinate system, then return to the drawing area to continue an operation already in progress. In TotalCAD it's possible to do all of this and more.

[Changing Focus with the Mouse](#)


[Changing Focus with the Keyboard](#)


## **Changing Focus with the Mouse**


Keep in mind that only one control on the screen can have the focus at a given time. When you move from control to control, you change the focus from one control to another. Generally speaking, users change the focus from one control to another by clicking the control they want to change to. If you are in the drawing area, you can move to the Edit Bar by simply clicking in one of its fields, or move to the Coordinate Fields by clicking in one of its fields. Then you return to the drawing area by clicking there.

## Changing Focus with the Keyboard

You can also move the focus between controls using the keyboard. Remember these keystrokes:

 If you are in the drawing area or the coordinate fields, you can move the focus to the Edit Bar by pressing <Ctrl>+<E>.

 If you are in the Edit Bar or the coordinate fields, you can move to the drawing area by pressing <Ctrl>+<D>.

 If you are in the Edit Bar or the drawing area, you can return the focus to the coordinate fields by pressing <Ctrl>+<R>.

There are also a couple of alternative ways to get out of the drawing area. You can also just start typing values that you want to go into the Edit Bar: This will change the focus to the Edit Bar, and values you type will be entered in the Edit Bar's first field. This last approach can be very convenient, but it won't work if you are using the Text tool.

 **Note**

 **See Also**

If you type values that are not appropriate for a particular Edit Bar field, the field may not accept the values, in which case it will warn you by beeping.

Defining Points and Specifying Measurements, Finishing and Canceling



## **Microsoft Office Compatibility**

TotalCAD is a Microsoft Office Compatible product, which means that its toolbars, menus, and accelerator keys are similar to those used by Microsoft Office. If you are already using Microsoft Office, which includes Microsoft Excel, Word, Microsoft Access and PowerPoint, then many of the tasks you have learned to complete in Office can be completed in a similar manner in TotalCAD. IMSI and Microsoft hope these similarities will make it easier for you to use our products together and with other Microsoft Office Compatible products.

Look for the Microsoft Office Compatible logo when purchasing software. For more information about the Microsoft Office Compatible program, and for a complete listing of Microsoft Office Compatible products, in the United States call Microsoft Customer Service at 1-800-426-9400. Customers outside the United States should contact their local Microsoft office.

[Microsoft Office Features](#)

[Using TotalCAD with Microsoft Office](#)

## **Microsoft Office Features**

TotalCAD's toolbars, menu system, and accelerator keys have been revised for consistency with Microsoft Office. TotalCAD's template Wizards and Tips of the Day will also be familiar to Microsoft Office Users.

## **Using TotalCAD with Microsoft Office**

TotalCAD fully supports OLE 2.0, so you can include Microsoft Word, Excel, and Powerpoint objects in TotalCAD via the Insert Object and Paste Special commands. Simply double-click on OLE objects to edit them in their source application, or use the Edit|Object command. Links can be updated and modified using the Edit|Links command.

TotalCAD objects can similarly be included in other OLE 2 compliant applications.

For more information on integrating Microsoft Office products, see your Microsoft Office documentation.

## **File Management and Printing**

TotalCAD 1.0 provides all standard Windows features for creating, opening, and saving files. Its open and save features support all standard CAD file types.

TotalCAD's printing features make it easy to print your drawings on single pages or to "tile" them across multiple pages that can be joined into a single large printout.

[Supported File Types](#)

[Printing](#)

## Supported File Types

TotalCAD supports a variety of file types, including the standard.DXF format. This means that you can open files created in other CAD programs using a standard file format, such as .DXF, then edit and save the files in TotalCAD format; or you can save TotalCAD files as .DXF, then edit and save them in another application.

The following table describes the file types supported by TotalCAD 1.0:

<u>File Type</u>	<u>Extension</u>	<u>Comment</u>
TotalCAD 1.0	.TCW	Default file type for TotalCAD 1.0.
TotalCAD Template	.TCT	Use predefined or custom templates as basis for new drawings.
AutoCAD DXF	.DXF	Common CAD file exchange type.



**Warning**

 **See Also**

If you open a file created in another version of CAD, then save the file in TotalCAD 1.0, the file type will be updated to TotalCAD 1.0. You can also save a TotalCAD 1.0 in the DXF format so you can open and edit the file in another CAD program.

Open Dialog, Save As Dialog

## Printing

TotalCAD 1.0 has a wide range of printing features, which make it easy to scale and center your drawing on the page, or to tile your drawing across multiple pages. TotalCAD also provides a full range of advanced printing options for experienced users.

[Printer Paper and the Drawing Sheet](#)

[Simple Printing](#)

[Tiled Printing](#)



## **Printer Paper and the Drawing Sheet**

Before dealing with TotalCAD's printing capabilities, it will help if you understand a potentially confusing concept: printer paper versus the drawing sheet. Printer paper means the physical sheets of paper that go through the printer. The drawing sheet is the total surface area on which your drawing will be printed. The drawing sheet can correspond to a single sheet of printer paper, or it can span multiple sheets of printer paper.

Both the printer paper and the drawing sheet have settable sizes, as well as landscape and portrait orientations. This provides a great deal of flexibility when you want to print a large drawing on multiple sheets. You could, for example, print a poster in landscape orientation three feet wide and two feet tall. You could print this virtual sheet out on 32 sheets of 8.5" x 11" paper oriented in portrait, or on 30 sheets of the same size paper oriented in landscape.

## Simple Printing

TotalCAD's many printing options may seem confusing at first glance, but in most cases you can print your drawing in just a few simple steps. Let's assume that you have created a drawing in World scale, without paying any attention to how your drawing will be set up on the page. Assume as well that you want to print on a single sheet of paper. Follow these steps:

1. Choose File|Print. The Print dialog will appear.
2. In the Print Range control group, click the All option to print your entire drawing.
3. Check the Fit on 1 Page option.
4. Click the Print Setup button to bring up the Print Setup dialog. Choose your printer, paper size, and paper orientation, then click OK to return to the Print dialog.
5. Click the OK button to print your drawing.

Your drawing will be centered and printed on a single sheet of paper.

## Tiled Printing

TotalCAD gives you two different approaches to printing a drawing on multiple tiled sheets of paper, using the Paper and Layout property pages of the Page Setup dialog. The two following topics describe these two options.

[Tiled Printing by Setting Sheet Size](#)

[Tiled Printing by Setting Rows and Columns](#)



When printing on multiple tiled sheets, it is usually best have the Print Crop Marks option on, unless you set your margins to zero. Crop marks will enable you to easily trim the edges of the paper so that the sheets can be pasted onto a backing for display.

Page Setup Dialog

## Tiled Printing by Setting Sheet Size

If you want to create a printout of a specific size tiled on multiple sheets, you can specify the size of the printout in the Paper property sheet. TotalCAD will determine the number and arrangement of the sheets of printer paper.

1. Choose File|Page Setup. The Paper property sheet will be visible in front.
2. In the Printer Paper control group, specify the size of the sheets of paper that you want to print on, and whether you want the sheets of printer paper to be oriented in landscape or portrait.
3. In the Sheet Size control group, specify the size of the area on which you want to print the drawing, and its orientation. You can do this in any units listed in the Tools|Drawing Setup|Units and Size property sheet. For example, if you want a printout 2 feet tall and 3 feet wide, you would type 2 ft x 3 ft in the list box, and choose the Landscape option.
4. Click the Fit button to place your drawing on the tiled sheets.
5. Click the Print button to print your drawing immediately, or click OK and then choose File|Print if you want to go through the Print dialog.



You can also set the sheet size in either World or Page units by specifying values in the Height and Width fields on the Layout property sheet.







## **Tiled Printing by Setting Rows and Columns**

In the Layout property sheet of the Page Setup dialog, you can specify the number of rows (the number of sheets of printer paper from top to bottom) and columns (the number of sheets of printer paper from left to right). You can then return to the Paper property sheet to fit your drawing onto the tiled printer paper. TotalCAD will automatically adjust the dimensions of the drawing sheet to accommodate changes in the number of rows and columns.





## Editing Objects

The Edit menu contains commands and tools for selecting, editing, and copying objects. These features enable you to

-  Undo a previous operation, or redo what was previously undone.
-  Perform standard Windows clipboard operations: Cut, Copy, and Paste.
-  Clear (delete) objects using several options.
-  Select objects with the Select tool, or categorically by type, layer, or several other criteria.
-  Create multiple copies of entities along lines, around arcs, and in two-dimensional arrays.
-  Edit OLE objects using the source application and make changes to how OLE objects are linked to your TotalCAD drawing.

In addition to the features available from the Edit menu and toolbars, TotalCAD has two separate modes for [mouse-based editing](#):

-  [Select Edit](#), in which you scale, rotate, or move objects.
-  [Node Edit](#), in which you reshape entities by editing special points called *nodes*.

## Editing with the Mouse and Local Menu

TotalCAD offers powerful capabilities for editing directly with the mouse and Local menu (which you access by right-clicking in the drawing area). There are two distinct editing modes that you can use for this purpose: [Select Edit](#), and [Node Edit](#).

## Select Edit

Select Edit is the editing mode an object is in when you first select it. In this mode, the selected objects are bounded by a *selection rectangle* that has handles on each corner and at the midpoint of each side. At the center of the selection rectangle is the *reference point*, which is connected to the *rotation handle*, located just outside of the selection rectangle.

[Moving the Selection's Reference Point](#)

[Moving a Selection](#)

[Scaling a Selection](#)

[Copying While Moving or Scaling](#)

[Rubber Stamp](#)

[Rotating a Selection](#)

## Moving the Selection's Reference Point

You can move the reference point of a selection to any position in the drawing area. To do this, hold down the <Ctrl> key while clicking on the reference point. The cursor will change to a hand grasping the point. You can now move the reference point to any position by defining a point. You can do this with the mouse using any snap mode, or you can type the coordinates of the point into the Coordinate Fields and press <Enter>.



When you rotate an object, you do so in relation to its reference point. This means that you can rotate the object not only around its own center, but around any arbitrary point or object in the drawing space. For example, you can use Snap to Arc Center mode to move the reference point of the selection to the geometric center of a circle or arc, then “orbit” the selection around that point.

To restore the reference point to its original location at the center of the selection, you can deselect and then reselect the objects.

## **Moving a Selection**

There are three ways to move a selection in Select Edit mode: "picking it up" by its reference point, dragging it by its reference point (OLE drag and drop), and "simple dragging" (that is, dragging by any point within the selection rectangle other than the reference point).

[Picking the Reference Point](#)

[OLE Drag and Drop](#)

[Simple Dragging](#)

## Picking the Reference Point

In most cases the most convenient and accurate way to move an object is to click on its reference point, then define a new location for the reference point. The object moves with the reference point:

1. Select the object or objects that you want to move.
2. If desired, move the selection's reference point relative to the selection. For example, if in moving a rectangle you wish to precisely place its lower-left corner, rather than its center, first move the reference point to the lower-left corner of the rectangle. You can use snap modes for precise placement of the reference point.
3. Click on the selection's reference point. The cursor changes to four-directional pointer (crossed arrows).
4. Define the point where you want to place the selection. You can do this by clicking the mouse, using any snap mode. Alternatively you can press <Tab> to leave the drawing area, then type the coordinates of the desired location in the Coordinate Fields and press <Enter>.

 **Tip**

 **See Also**

When you move a selection by picking its reference point, you can use Ortho mode when you place the selection. You can turn on Ortho mode in the menu, toolbar, or local menu before or during the placement of the selection; you can also toggle Ortho mode during placement by holding down the <Shift> key.

To copy while moving, hold down the <Ctrl> key immediately after you have clicked on the reference point.



## Moving the Selection's Reference Point

## OLE Drag and Drop

You are using OLE drag and drop whenever you drag an object by its reference point

1. If you're going to use a snap mode, set it before starting the move. You may also want to move the reference point relative to the selection before beginning the move.
2. Place the mouse cursor directly over the reference point. You will know it is there when the mouse cursor turns into a four-directional pointer.
3. Hold the mouse button down while "dragging" the object to the new location, then release the mouse.

The main advantage of OLE drag and drop is that it lets you *copy* the selection from one drawing into another, or even from a TotalCAD drawing into another application.



**Warning**



**See Also**

The behavior of snap modes is slightly unusual when you are moving an object using OLE drag and drop. If TotalCAD finds a snap point based on the location of the mouse at the point of release, then the reference point will snap to the snap point. If TotalCAD cannot find a snap point, then the object will simply be dropped at the point of release.

## Snap Modes

## **Simple Dragging**

If you place the mouse pointer anywhere inside a selection other than on the reference point, you can simply drag the selection to another location within the drawing. Simple dragging does not allow you to use snap modes or the Coordinate Fields for accurate placement, but it is convenient if you don't need accurate placement.

## Scaling a Selection

You can scale a selection either by dragging its handles, or by entering values for X and Y scale in the Edit Bar.

Drag the handles on the corners to freely resize the selection both horizontally and vertically. To scale the selection only horizontally or only vertically, drag the handles on the sides, top, or bottom of the selection rectangle. To scale the selection while keeping the same proportions of height and width, hold down the <Shift> key while dragging the handles.

To scale a selection using the Edit Bar, type values into the XScale and YScale Edit Bar fields, then press <Enter>. For example, if you want to double the width of the selection while increasing its height by 50 percent, type 2 into the XScale field and type 1.5 into the YScale field, then press

## **Copying While Moving or Scaling**

The Make a Copy option in the Local menu lets you make a copy of the selection at the same time that you move or resize the object. To use this option:

1. Select the object or objects that you want to copy.
2. Right-click in the drawing area to open the Local menu, then choose Make a Copy, if it isn't already checked. (This is a toggle, so if it already has a check mark next to it, choosing it will turn it off.)
3. Move or scale the selection by either picking the reference point or dragging by the reference point.

When you move or scale the selection, the original will remain at its original location and size, while a copy is created at the new location and size.

## Rubber Stamp

Turning on the Rubber Stamp option allows you to place multiple copies of selected objects in your drawing. To use this option:

1. Select the object or objects that you want to copy.
2. Right-click in the drawing area to open the Local menu, then choose Rubber Stamp, if it isn't already checked. The cursor will change into a rubber stamp symbol.
3. Click anywhere in the drawing to place a copy of the selection.

You can continue clicking to place as many copies of the selection as you like. To turn Rubber Stamp off, right-click in the drawing area to open the Local menu and choose Cancel, or press <Esc>.





Rubber Stamp placement does not respond to snap modes.

## Rotating a Selection

To rotate a selection either move its rotation handle, or type a value into the Edit Bar. To move the rotation handle of the selection, click on the handle, then define a point by clicking the mouse, using any snap mode.

To rotate the selection using the Edit Bar, type a value for the degrees of rotation into the Edit Bar's Rotation field, then press <Enter>. The value you enter in the Edit Bar is the number of degrees that the selection will be rotated in the positive direction of rotation. To rotate the object in the opposite direction, enter a negative value in the Rotation field.

 **Note**

 **Tip**

 **See Also**












The positive direction of rotation is counterclockwise by default. You can reverse the direction in the Angle property sheet of the Drawing Setup dialog.

When you rotate a selection you can use Ortho mode to constrain the angle of rotation. You can turn on Ortho mode in the menu, toolbar, or local menu before or during the placement of the selection; you can also toggle Ortho mode during placement by holding down the <Shift> key.

Ortho

## Node Edit

In Node Edit mode you can reshape entities by directly manipulating particular points called *nodes*.

-  Add, delete, and move nodes on entities composed of line segments (including lines, double lines, polygons, and sketches).
-  Divide line segments into any number of subsegments of equal length.
-  Break polygons, making them into multilines.
-  Close multilines, making them into polygons.
-  Close the endpoints of double lines.
-  Reshape Bezier and spline curves by adding, deleting, and moving nodes on the line segments that form the curves' frame.
-  Change the radius of circles and circular arcs.
-  Change the major and minor axes of ellipses and elliptical arcs.
-  Change the start and end angles of circles, ellipses, and arcs.
-  Close arcs, making them into circles.
-  Divide circles, arcs, and ellipses into arc segments of equal angular length.

To activate Node Edit mode, first select the entity that you wish to edit. Then click the right mouse button in the drawing area to display the Local menu. From the local menu, choose Node Edit. The appearance of the selection will change. Instead of a selection box with handles, you will see highlighted objects with nodes displayed as blue rectangles. The number and placement of the nodes depends on the type of object selected.

To deselect an entity in Node Edit mode, choose Cancel or Select Edit from the Local menu to return to Select Edit mode, then click anywhere in the drawing area outside the selection rectangle to deselect.

[Node Edit with Lines and Polygons](#)

[Node Edit with Curves and Sketched Lines](#)

[Node Edit with Circles, Ellipses, and Arcs](#)

[Node Edit of Dimensions](#)

[Manipulating Text](#)

 **Note**

You can only edit one entity at a time in Node edit. You cannot use this mode to work with multiple selected entities or with groups. To edit the nodes of an entity that is in a group, first select the group, then break it into its component entities using the Format|Explode command. Because groups can be nested, you may need to explode the selection more than once in order to isolate a single entity.

## **Node Edit with Lines and Polygons**

There are several editing actions you can perform on entities composed of line segments. These actions include adding a node, deleting a node, moving a node, and dividing a line or double-line segment.

[Adding a Node to a Line Segment](#)

[Deleting a Node from a Line Segment](#)

[Moving a Node](#)

[Dividing a Line or Double Line Segment](#)

[Breaking Polygons](#)

[Closing Multilines](#)

[Closing Endpoints of Double Lines](#)



## Adding a Node to a Line Segment

To add a node to a single line segment or to an entity composed of line segments, such as a multiline or polygon:

1. Select the entity that you want to edit. Click the right mouse button in the drawing area to open the Local menu, then choose Node Edit.
2. Hold down the <Ctrl> key while placing the mouse cursor over a part of a line or double-line segment where there is no node. When you are in the correct position, the cursor will show a node being placed at the center of a cross.
3. Click to place the node on the line segment.

A node will appear on the line or double-line segment at the location of the mouse cursor. You can now drag the node to reshape the line segment into a multiline.



As an alternative to holding down the <Ctrl> key while clicking on the line segment, you can place the mouse cursor at the position where you want to add the node, then right-click and choose Add Node from the Local menu.

## Deleting a Node from a Line Segment

To delete a node from a line segment:

1. Select the entity that you want to edit. Click the right mouse button in drawing area to open the Local menu, then choose Node Edit.
2. Hold down the <Ctrl> key while placing the mouse cursor over the node that you want to delete. When you are in the correct position, the cursor will change to an icon showing a node being thrown into a trash can.
3. Click to delete the node.

If you remove a node from a polygon, TotalCAD will keep the polygon closed by connecting the opposite endpoints of the line segments that were connected at the deleted node.

 **Note**

 **See Also**

You cannot delete a node that is necessary to define an entity as a line segment or polygon. This means that you cannot delete the endpoint of a single line segment, and you cannot delete any node of a triangle. If you delete the endpoint of a multiline, you will delete the entire line segment that terminates at that point.

Closing Multilines, Breaking Polygons

## Moving a Node

You can reshape a single line segment or any entity composed of line segments, such as a multiline or polygon, by defining a point for the location of any node. To do this:

1. Select the entity that you want to edit.
2. Click the right mouse button to open the Local menu and choose Node Edit.
3. Click on the node that you want to move.
4. Define the point to which you want to move the node. You can do this either using the mouse and any snap mode, or by typing the coordinates of the point in the Coordinate Fields and pressing <Enter>.



You can also simply drag the node to any position you want; dragging a node also gives you access to all snap modes.

## **Dividing a Line or Double Line Segment**

The Segment Divide command from the Local menu lets you add nodes to a line segment, breaking the line segment into a multiline composed of segments of equal length. To do this:

1. Select the entity that you want to edit.
2. Click the right mouse button to open the Local menu and choose Node Edit.
3. Place the mouse cursor over the line segment that you want to divide.
4. Click the right mouse button to open the Local menu and choose Segment Divide. TotalCAD will display the Divide Segment By dialog.
5. Enter the number of parts into which you want to divide the line into the dialog, then press <Enter>.



## Breaking Polygons

Node Edit allows you to break a polygon at any node, turning it into a multiline. To break a polygon:

1. Select the polygon that you want to edit.
2. Click the right mouse button to open the Local menu and choose Node Edit.
3. Position the cursor over the node where you want to break the polygon.
4. Click the right mouse button to open the Local menu and choose Break.

The polygon is now an open multiline, and there are two nodes located at the point where you broke the polygon. You can open the polygon visually by moving these nodes apart.



You can also use this technique to break a closed Bezier or Spline curve.

## Closing Multilines

Node Edit allows you to close a multiline, making it into a polygon. To close a multiline:

1. Select the multiline entity that you want to close.
2. Click the right mouse button to open the Local menu and choose Node Edit.
3. Position the cursor over either endpoint of the multiline.
4. Click the right mouse button to open the Local menu and choose Close.

TotalCAD will draw a line between the two endpoints of the multiline (assuming that these two nodes don't share the same location), forming a geometrically closed polygon. If the two endpoints of the multiline share the same location, TotalCAD will simply close the polygon geometrically, since it is unnecessary to draw a line between the two points.

 **Note**

 **Warning**

You can also close a Bezier or Spline curve using this technique.

If you simply snap the endpoints of a multiline together you have something that looks exactly like a polygon; however, TotalCAD still considers it a multiline. This means that you won't be able to use a fill by setting the entity's brush color and pattern in its Properties sheet. In order to turn a multiline into a true polygon, you must close the multiline.

## **Closing Endpoints of Double Lines**

Node Edit allows you to close the endpoints of double lines. To do this:

1. Select the double-line entity.
2. Click the right mouse button to open the Local menu and choose Node Edit.
3. Position the cursor over either endpoint of the double line.
4. Click the right mouse button to open the Local menu and choose Close Double End.

TotalCAD will close the double line with a flat end.

## **Editing Double Line Cutlines and Gaps**

Cutlines and gaps occur at the intersections of double lines. For example, if you form a clean T intersection of two double lines at an angle other than 90 degrees, the end of one of the double lines will be "cut" at an angle; the other double line will have a "gap" to accommodate the first double line. Node Edit mode allows you to drop the cutlines in the first line, "evening" out the end of the line; it also allows you to fill in the gap in the second line.

## Node Edit with Curves and Sketched Lines

Sketches (entities created with the Sketch tool) are actually multilines composed of miniscule line segments, so they can be edited in Node Edit mode as multilines that have a very large number of nodes.

Bezier and spline curves are both drawn in reference to a sequence of points (spline curves actually pass through the points, while Bezier curves gravitate toward the points). In Node Edit, each of the defining points of the curve is represented by a node.

In Node Edit mode, you can move or delete any of the nodes that define a curve, and you can add new nodes that you can move to reshape the curve. You can move and delete the nodes on a curve the same way that you move and delete nodes on a line segment, multiline, or polygon.

You can also close a Bezier or Spline curve, or break a closed Bezier or Spline curve at any node, using the same technique that you would use to close a multiline or break a polygon.

[Adding a Node to a Curve](#)

[Dividing a Line Segment in the Frame of a Curve](#)

 **See Also**



[Adding a Node to a Line Segment](#), [Breaking Polygons](#), [Closing Multilines](#), [Deleting a Node from a Line Segment](#), [Moving a Node](#)

## Adding a Node to a Curve

To add a node to a Bezier or spline curve:

1. Select the curve that you want to edit. Click the right mouse button in drawing area to open the Local menu, and choose Node Edit.
2. Hold down the <Ctrl> key while placing the mouse cursor over a segment of the curve where there is no node. When you are in the correct position, the cursor will change to an icon showing a node being placed at the center of a cross.
3. Click to place the node on the line segment.

A node will appear on the line segment at the location of the mouse cursor. You can now drag the node to reshape the curve.



As an alternative to holding down the <Ctrl> key while clicking on the line segment, you can place the mouse cursor at the position where you want to add the node, then right-click and choose Add Node from the Local menu.

## **Dividing a Line Segment in the Frame of a Curve**

The Divide Line command from the Local menu lets you add nodes to a line segment on the frame of a curve, breaking the line segment into any number of subsegments of equal length. To do this:

1. Select the curve that you want to edit. Click the right mouse button to open the Local menu, then choose Node Edit.
2. If the frame of the curve is not visible, choose Format|Properties from the menu (or right-click and choose Properties from the Local menu) to display the Properties dialog. Bring the Curve property sheet to the front by clicking on its tab, then click the Show Frame check box. Press <Enter> to close the dialog. You will now see the line segments that form the frame of the curve.
3. Place the mouse cursor over the line segment that you want to divide on the frame of the curve that you want to divide.
4. Click the right mouse button to open the Local menu and choose Divide. TotalCAD will display the Divide Segment By dialog.
5. Enter the number of parts into which you want to divide the line segment into the dialog, then press <Enter>.

You can move any of the nodes on the curve's frame to reshape the curve.

## Node Edit with Circles, Ellipses, and Arcs

In Node Edit mode you can change the radius of a circle or circular arc, the major and minor axes of an ellipse or elliptical arc, and the starting or ending angles of any arc, circle, or ellipse. You can also divide any circular or elliptical entity into any number of equal arcs.

To edit a circle, ellipse, or arc in Node Edit mode, first select the entity that you want to edit, then click the right mouse button in the drawing area to open the Local menu and choose Node Edit. The entity that you selected will be highlighted with its nodes displayed. In the case of a circle or circular arc, you will see three nodes: two for manipulating its starting and ending angles, and one for manipulating its radius. In the case of an ellipse, you will see four nodes: two for manipulating its starting and ending angles, and two for manipulating its major and minor axes.

[Changing the Radius of a Circle or Circular Arc](#)

[Editing the Major and Minor Axes of Ellipses and Elliptical Arcs](#)

[Changing the Starting and Ending Angle of an Arc](#)

[Dividing an Arc](#)



**Tip**

If you are confused by the notion of editing the starting and ending angles of a circle or ellipse, try thinking of circles and ellipses as arcs that have their starting and ending angles at zero and 360 degrees.

## Changing the Radius of a Circle or Circular Arc

To alter the radius of a circle or circular arc:

1. Select the circular entity that you want to edit. Click the right mouse button to open the Local menu, then choose Node Edit. You will see three nodes, one of which, a blue square, is the *radius node*: You can manipulate this node to change the circle's radius.
2. You can now either define a point for the radius node, or directly enter the radius of the circle in the Edit Bar. To define a point, click on the radius node, then define a point by clicking the mouse, using any snap mode; or type values for the point's coordinates in the Coordinate Fields, then press <Enter>.


To specify the radius in the Edit Bar, press <Tab> to leave the drawing area, type a value for the radius into the Radius field of the Edit Bar, then press <Enter>.


You can also simply drag the radius point to another position, but in doing this you lose the precision of using snap modes, the Coordinate Fields, or the Edit Bar.

## Editing the Major and Minor Axes of Ellipses and Elliptical Arcs

To scale an ellipse or elliptical arc:

1. Select the elliptical entity that you want to edit. Click the right mouse button to open the Local menu and choose Node Edit. You will see four nodes, two of which are blue rectangles on the perimeter of the ellipse: You can manipulate these nodes to scale the ellipse.
2. You can now either define points for the the two nodes, or directly enter val-ues for the lengths of the major and minor axes of the ellipse in the Edit Bar:

 To define points for the nodes, first click on one of the nodes, then define a point either by clicking the mouse using any snap mode, or by typing values for the point's coordinates in the Coordinate Fields and pressing <Enter>. Repeat this procedure for the other node.

 To specify the lengths of the axes in the Edit Bar, press <Tab> to leave the drawing area, type a value for the axes in the Major and Minor fields of the Edit Bar, then press <Enter>.

You can also simply drag the nodes to another position, using any snap mode.

 **Tip**



While changing the length of an ellipse's axis, you can hold down the <Shift> key, constraining the proportions of the ellipse and changing the length of both axes simultaneously.

## Changing the Starting and Ending Angle of an Arc

To alter the starting and ending angles of a circle, circular arc, ellipse, or elliptical arc:

1. Select the entity that you want to edit. Click the right mouse button to open the Local menu and choose Node Edit. You will see three nodes for circular entities and four nodes for elliptical entities. Two of the nodes are represent-ed as a yellow and a blue circle at the ends of dotted lines. You can manipulate these nodes to change the starting and ending angles of the entity.
2. You can now either move the nodes to specify starting and ending angles, or enter values for the starting and ending angles in the Edit Bar:  
To move the nodes, first click on one of the nodes, then define a point by clicking the mouse, using any snap mode. Repeat this procedure for the other node.  
To specify the starting and ending angles in the Edit Bar, press <Tab> to leave the drawing area, type a value for the axes in the Start Ang and End Ang fields of the Edit Bar, then press <Enter>.

## **Dividing an Arc**

The Divide command from the Local menu lets you divide an arc or circle into subarcs of equal angular length. To do this:

1. Select the entity that you want to edit. Click the right mouse button to open the Local menu and choose Node Edit from the Local menu.
2. Click the right mouse button to open the Local menu, and choose Arc Divide. TotalCAD will display the Divide Arc By dialog.
3. Enter the number of parts into which you want to divide the arc into the dialog, then press <Enter>.

## **Node Edit of Dimensions**

Dimensions have nodes that let you manipulate the position of the dimension line, extension lines, and dimension text. You can also move the associative nodes of the dimension, including the vertex of an angular dimension. This has the effect of changing the dimension's automatic text, and breaking the association between the dimension and the object with which the associative nodes were linked.

## Manipulating Text

TotalCAD has a rich set of features for manipulating text, based on the principle that you can edit the text either as a TrueType font, or as a polygon.

When you first create text, it is a TrueType font. You can select the text, resize it, rotate it, and stretch it. One advantage of this is that the text will always look "correct" onscreen and in printed copy, no matter how it is scaled or rotated. When text is in this state you can edit it by changing the Attribute field of the General property sheet in the Properties dialog. Double-click on the text with the Select tool to bring up this dialog. You can reformat the text (changing its font, size, attributes, and justification) using the Format Text toolbar or the Text property sheet of the Properties dialog.

If you want more freedom to manipulate the text graphically, you need to turn the text into an irregular polygon. To do this, select the text object using the Select tool, then choose Format|Explode. The first time you do this, you will turn the text object into a group of polygons. You can now resize, stretch, rotate, or flip the graphic text using handle-based editing, producing a variety of effects.

If you explode the text a second time, it will ungroup so that each individual character can be manipulated as a polygon. You can resize, stretch, move, flip, or rotate each individual character in Select Edit mode. You can change the pen, brush pattern, or other properties of the multiline by double-clicking on the selection and changing the settings in the Properties dialog.

If you select an individual polygon (character), and then right-click and choose Node Edit from the local menu, you can edit individual nodes. You can move, delete, and add nodes; and you can also break the polygon into a multiline.

If you explode an individual polygon character, it will be broken into individual line segments.

 **See Also**

[Node Edit with Lines and Polygons](#), [Select Edit](#), [Text Property Sheet](#)

## Drawing Environment

The View menu provides features that let you control two aspects of the drawing environment. These features enable you to:



Change your [view of the drawing area](#) by zooming and panning.



[Set up your desktop](#) by displaying and hiding toolbars and other interface controls.

## Views of the Drawing Space

The TotalCAD drawing space is a plane on which you can place drawing entities and other objects. A *view* is the area of the drawing space that appears on your screen. As your drawing become larger and more complex, it's important to be able to find views that let you work at the right location in your drawing, at an appropriate level of detail. You may need to concentrate your work within a tiny portion of the drawing, you may need to pull back and see all the objects on the drawing, or you may need to see the drawing at the size it will be when printed. TotalCAD offers a variety of options for [panning](#) and [zooming](#) that get you quickly to the exact view that you need.





Another powerful way to control your view of the drawing is by selectively viewing layers. You can control the visibility of layers through the Tools|Layers property sheet.

Layers Property Sheet

## Zooming

*Zooming* means to move in or out of the drawing space, viewing the drawing at a greater or lesser level of detail. The [View menu](#) provides a number of zooming commands.

One quick and easy way to zoom is to use the plus and minus keys of the numeric keypad: plus zooms in one step; minus zooms out one step. The amount that you zoom in each step depends on the *zoom factor*, which by default is set to 2. You can set the zoom factor in the [Tools|Program Setup](#) dialog.

 **Notes**

 **Tip**

 **See Also**

When you zoom in and out using the plus and minus keys, the view centers on the mouse cursor. If you zoom in using the menu command, you will zoom in on the center of the current view, regardless of the position of the mouse cursor.

As you zoom out, TotalCAD simplifies the representation of very small objects on the screen to increase redraw speed.

To zoom in on a particular point in the drawing and place it at the center of the screen, place the mouse cursor over the point and press the <+> key on the numeric keypad.

[Program Setup Dialog, Zoom Commands](#)

## Panning

*Panning* means to move the view to another location across the plane of the drawing. TotalCAD's [View menu](#) provides several different options for panning.

Two simple ways of panning are to use the scroll bars or arrow keys. The scroll bars work the same way that they do in other Windows programs. You can click the scrollbar arrow buttons to scroll one step, click in the body of the scrollbar to scroll about one third of a screen, or drag the “thumb” of the scrollbar to move quickly across the drawing. The arrow keys are also simple and intuitive: press the up, down, left, or right arrow key to pan a single step.



To quickly pan to a specific point, place the mouse cursor at the point, then type <Ctrl>+<End>. You will instantly move to a view with the point you chose at the center of the screen.



## Setting up the Desktop

In TotalCAD 1.0 you can set up the user interface to suit your own preferences, or to fit the requirements of a particular drawing task. You do this by selectively [displaying and hiding](#) toolbars and other interface controls, and by positioning or ["docking" toolbars and other movable controls, such as palettes, on the TotalCAD desktop.](#)

 **Tip**

 **See Also**

You can control whether TotalCAD saves your desktop arrangement when you exit by setting the Save Desktop on Exit option in the Program Setup dialog.

## Program Setup Dialog

## Displaying and Hiding Controls

The View menu provides a number of commands for displaying and hiding interface controls such as the Edit Bar, Coordinate Fields, and rulers. Each of these commands is a *toggle*, which means that it acts as an on/off switch. If the menu command is displayed with a check mark, the control is currently displayed. If the menu command is displayed without a check mark, the control is hidden.

You can display and hide the TotalCAD toolbars through the Tools|Desktop property sheet, which is also accessible through the View|Toolbars command.

 **Tip**

 **See Also**

It's generally a good idea to set your preferred settings in the Tools|Desktop property sheet, and then use the View menu to show and hide individual controls as you are working.



Desktop Property Sheet, TotalCAD Desktop

## **Docking Toolbars**



You can move any toolbar, or the Edit Bar, by positioning the mouse over its edge and dragging it to another position. There are three places on the TotalCAD screen where you can “dock” toolbars: the upper toolbar area, just below the menu bar; the left or right edge of the screen, which expands to form a vertical docking area; and the drawing area. When you dock a toolbar on the upper part of the screen it will be oriented horizontally. If you place it along the left or right edge of the screen it will be positioned vertically. Toolbars placed over the drawing area will act as a floating palette. A floating palette is always visible in front of your drawing.

## Creating and Inserting Objects

The Insert menu gives you access to all of the TotalCAD drawing tools, as well as to dialogs that let you insert various types of objects into your drawing. You can use the drawing tools to create:

-  Drawing entities (such as lines, circles, and polygons).
-  Construction entities (temporary lines that you use as reference points or to mark off areas in your drawing).

You can also insert the following types of objects into your drawing:

-  Pictures (Windows metafiles or bitmaps).
-  OLE objects, which you can edit using other Windows applications from within TotalCAD.

[Creating Entities](#)

 **See Also**



[View Symbol Library](#)

## Creating Entities

To use any drawing tool you need to understand a set of common techniques. These techniques include :



[Formatting objects](#) by setting their properties.



[Defining points](#) by using the mouse in combination with snap modes, or by using the Coordinate Fields, and [specifying measurements](#), such as length and angle, using the Edit Bar.



[Finishing and canceling](#) operations.



[Using the Local Menu](#).

For a more detailed discussion of how to apply the techniques in the process of creating an actual drawing, run the TotalCAD online tutorial by choosing Tutorial from the Help menu.

## Formatting Entities

When you draw an entity in TotalCAD, the entity is formatted using the current default properties of the tool with which you draw it. These properties include, among many other things, the width, style, and color of the pen, as well as the pattern and color of the brush. In addition to these common properties, some types of entities, such as text, double lines, curves, and dimensions, have properties that belong only to the specific entity type. To set the properties for a drawing tool, you can activate the tool and choose Format|Properties, or you can simply right-click on the drawing tool.

Once an entity is drawn, you can format it by changing its properties. To do this, first select the entity, then choose Format|Properties; or simply activate the Select tool, then double-click on the entity that you want to format.

 **Note**




 **See Also**

You cannot access or change the properties of construction lines.

[Setting Properties](#), [Property Toolbar](#) , [Properties](#)




## Defining Points and Specifying Measurements

To create any entity you must use some combination of the following three methods:

-  Define points by clicking the [mouse](#) in the drawing area, using snap modes.
-  Define points by entering their coordinates in the [Coordinate Fields](#).
-  Specify measurements (lengths, angles, and sometimes other information) in the [Edit Bar](#).

Almost always, the first step in drawing any entity is to define its starting point. After the first point is defined, you create the entity by defining a series of points and/or specifying measurements.

For example, to draw a circle using the Center and Point tool, you first need to define a point for the center of the circle. After this point is defined, you have three ways of completing the circle:

-  Define any point that will be on the perimeter of the circle using the Coordinate Fields or mouse.
-  Specify the radius of the circle in the Edit Bar.
-  Specify the circumference of the circle in the Edit Bar.

The method you choose depends on the specific characteristics of the tool you are using, the drawing task, and your own preferences.



**Warning**

Trying to draw an entity in the Edit Bar without defining the entity's starting point can sometimes lead to confusing results. It's almost always simpler to start drawing an entity by defining its starting point.

## ***Defining Points with the Mouse***

In a sense, you define points with the mouse by simply pointing and clicking. However, what exactly happens when you define a point this way depends on *snap modes* and on TotalCAD's *Ortho* setting. The following topics discuss these features briefly.

[Using Snap Modes](#)

[Constraining Angles with Ortho](#)

[Setting Snap Modes and Ortho](#)



## Using Snap Modes

The result you get when you click the mouse in the drawing area depends, in part, on the current snap mode. If your snap mode is currently set to No Snap, then the point will simply be defined at the position where you clicked the mouse. Otherwise, the location of the point will be calculated relative to a drawing object or to the grid. If there is no point to snap to based on the mouse position, TotalCAD will beep, and it won't define a point at all. To find out how to change the snap modes, see [Setting Snap Modes and Ortho](#).

 **See Also**

## Snap Modes

## Constraining Angles with Ortho

To constrain angles when you are drawing with the mouse, use Ortho mode. You can activate Ortho through the Modes menu, the Snap toolbar, or the Local Menu. The Local Menu turns Ortho on or off only until the next point is defined; TotalCAD then reverts to the Ortho setting in the Modes menu.

If you are using TotalCAD's factory settings, Ortho restricts lines to orthogonal (vertical and horizontal) angles. You can configure ortho mode to less restrictive settings in the Angle property sheet (choose Tools|Angle). Some CAD users, for example, prefer to restrict angles to increments of 15 degrees.

 [See Also](#)

Angle Property Sheet, Ortho

## Setting Snap Modes and Ortho

You can change Snap modes and Ortho mode any time, even while you are in the middle of drawing an entity. For example, to create a spoke on a wheel you could snap the first point of a line to the center of a circle, then snap the second point of the line to a point on the perimeter of the circle. To set a default snap mode, use the Snaps toolbar or the main menu; to set a "one shot" snap mode that stays in effect until you define the next point, use the Local Menu. To use the Local Menu, click the right mouse button in the drawing area to open the menu, choose the desired Snap mode, then click the mouse to define the point.

 **Note**

 **Tip**

The Local Menu gives you “one shot” mode changes. If you change modes using the Local Menu, the mode will change back to the setting in the Modes menu after you define the next point.

Another convenient way to turn Ortho on or off temporarily is to hold down the <Shift> key while drawing.

## Defining Points Using the Coordinate Fields

In addition to using the mouse with Snap Modes and Ortho, you can also define points by entering them directly into the Coordinate Fields and then pressing <Enter>. Points defined in the Coordinate Fields are always defined at the exact coordinate positions specified, regardless of the snap mode. The type of coordinates you type in the Coordinate Fields depends on the coordinate system you are using. In the absolute and relative coordinate systems you use X,Y coordinates. In the polar coordinate system you enter an angle and distance.

You can lock down the coordinates in the Coordinate Fields, so that the values in the boxes don't change when you move the mouse cursor over the drawing area. To do this, check their lockdown boxes (these are immediately to the left of the fields).

 **See Also**

 **Tip**



Coordinate Fields, Coordinate Systems

The Coordinate Field lockdown boxes are particularly useful when you lock down just one dimension. If you enter a value in just one of the fields and lock it down, you can then set the second coordinate using the mouse. This makes it easy to define a series of points along a specific horizontal or vertical line.

## Drawing with the Edit Bar

The Edit Bar lets you draw entities *parametrically* by specifying dimensions and angles. To do this, you must define a starting point for the entity that you are drawing. For example, to draw a regular polygon using the Edit Bar, define the centerpoint of the polygon using the mouse or Coordinate Fields, and define the polygon's number of sides, radius, and start angle in the Edit Bar.

[Using the Edit Bar Lockdown Boxes](#)



Creating or editing an entity with the Edit Bar ignores Ortho and snap mode settings. The entity will be defined using the exact values that you enter in the Edit Bar.

## Using the Edit Bar Lockdown Boxes

Many fields on the Edit Bar have lockdown check boxes, similar to those on the Coordinate Fields. When these boxes are checked, the values in the fields don't change when you move the mouse over the drawing area. This makes it possible to lock down one of the values and manipulate the second value with the mouse.


For example, you might want to specify the exact radius of an arc in the Edit Bar, then use the mouse to determine the locations of the arc's starting and ending angles. After defining the starting point for the arc, you could type its radius into the Edit Bar, then lock down the Radius field. This prevents the radius from changing when you re-enter the drawing area. You could then use the mouse and snap modes to determine the arc's starting and ending angles.


## Finishing and Canceling


Some entities are defined by a specific number of points. For example, you can draw a triple-point circle by defining three points. When you define the circle's third point, the circle is finished automatically.

Other entities, such as multilines, are composed of an indeterminate number of parts. You can add as many segments to a multiline as you like.

Users of earlier versions of TotalCAD may be confused by the way you cancel and finish such entities in TotalCAD 1.0. However, with a little practice you will find that TotalCAD 1.0 is consistent and intuitive in the way it handles these operations. It will help the transition if you remember these rules:

 *Cancel* always means to stop the current drawing operation and start over. It removes the entire entity you are currently drawing. For example, if you press <Esc> or choose Cancel from the Local Menu while you are drawing a multiline, the entire multiline will vanish, not just the last segment.

 *Finish* means to complete the entire entity. For example, if you are creating a multiline you can finish the entity by (1) double-clicking, (2) pressing the right mouse button and choosing Finish from the Local Menu, or (3) pressing <Ctrl>+<F> while working in the Edit Bar.

 Pressing <Enter> in the Edit Bar means to accept the current setting in the Edit Bar fields. If you press <Enter> while drawing a multiline, the settings in the Edit Bar will be used to draw the next segment of the multiline. If you are creating an entity with a fixed number of steps, then pressing <Enter> in the Edit Bar will finish the entity, provided you have already defined a starting point.

 **Note**

 **Tip**

When you are working in the Coordinate Fields, pressing <Enter> simply defines a point as specified in the Coordinate Fields.

Entities that can be closed geometrically, specifically the Multiline, Double Multiline, Bezier Curve, and Spline Curve tools, provide another option for finishing. If you choose Close from the Local Menu, TotalCAD will complete the entity by drawing a line or curve from the entity's first point to its last point.









## **Using the Local Menu**

The Local Menu is context-sensitive: The specific commands and options available in the Local Menu change, depending on the currently active drawing or editing tool. Many of these commands and options are quite powerful, so it really pays to right-click in the drawing area and see what options are there when you are learning to use a drawing tool.

For example, when you are using the Text tool, the Local Menu lets you size and rotate the text "on the fly," and specify with great precision exactly where the text will be placed relative to the insertion point.

## Formatting Objects

TotalCAD's [Format menu](#) contains commands and tools for setting the appearance and format of drawing entities and other objects. These features enable you to:

-  Format objects by setting their properties.
-  Combine objects into groups.
-  Explode groups and objects into their constituent parts.
-  Apply hatch patterns to objects.
-  Send objects to the back and bring them to the front.
-  Align multiple selected objects.

[Setting Properties](#)


[Property Toolbar](#)


[Formatting Text](#)

[Groups](#)

## Setting Properties

Every object in TotalCAD has a set of *properties*. The exact set of properties depends on the type of object. All drawing entities share the following common sets of properties, organized by property sheet in the Properties dialog:

 *General*. The layer on which the object resides, and a text attribute (generally its name) associated with it.

 *Pen*. The color, width, and style of lines.

 *Brush*. Determine the characteristics of fills and hatches.

There are also properties associated with specific classes of entities. Dimensions, for example, have a number of properties that apply specifically to them, located on the Format property sheet. Double lines, text, and curves also have properties and corresponding property sheets specific to them.

When you set properties, you can either set the default properties for a drawing tool, or set the individual properties of selected objects. These options are covered in the following topics.

[Setting the Default Properties of a Tool](#)

[Setting the Properties of Selected Objects](#)

 **Note**

 **See Also**

The Properties dialog is context-sensitive, meaning that it will display only the property sheets appropriate to the currently selected tool (or the currently selected objects). For example, if you select a line segment and a double-line segment, then bring up the Properties dialog, you will see the General, Pen, Brush, and Double-Line property sheets. You won't see the Curve property sheet, for example, because the objects you have selected don't include any curves.

## Properties

## **Setting the Default Properties of a Tool**

When you set the default properties of a tool, all subsequent entities drawn with that tool will have the settings you specified, until you explicitly change them. To access the default properties, either right-click on the tool icon, or activate the tool and then choose Format|Properties.

## Setting the Properties of Selected Objects

The Properties dialog is context-sensitive, meaning that when you open it it will display the set of property sheets that are appropriate to the objects selected. If you select a double line, or a set of entities that include a double-line entity, then bring up the Properties dialog, the properties dialog will contain the double-line property sheet.

You can use this dialog to set the default properties for a drawing tool; or to set the individual properties of a particular object or set of objects.

If you have property settings that you want to reuse, you can save them as a style, then reapply the style later to other entities.

 **See Also**

[Layers Property Sheet](#), [Property Toolbar](#) , [Properties](#)



## Property Toolbar

The Property toolbar gives you a quick and convenient way of setting the properties of any selected object or set of objects, or for setting the default properties of a drawing tool. If the Property toolbar is not visible on the screen, you can display it using the View Toolbars dialog.

If no objects are currently selected, the settings on the Property toolbar apply to the active drawing tool. For example, if you activate any single line tool and then change the settings on the Property toolbar, the settings will apply to all objects drawn with any single line tool.

If there are objects selected, settings in the Property toolbar apply to the selected objects.

The Property toolbar consists of five pulldown list boxes. From left to right they are:



*Style.* Use this list box to apply predefined property settings to a drawing tool. You can save styles in the Properties dialog. This control can be used only to set the properties of a drawing tool; it is unavailable if you are setting the properties of selected objects.



*Layer.* Use this list box to set the layer on which objects are drawn.



*Color.* Use this list box to set the pen color.



*Line Style.* Use this list box to set the style of the line. You can use a continuous line or choose from several types of dotted or dashed lines.



*Brush Style.* Use this list box to choose a fill pattern for closed entities.



**Note**



**See Also**

In TotalCAD 1.0 you cannot apply a style to a selection. The Style pulldown box is accessible only when you are setting the default properties of a drawing tool.

Layers Property Sheet, Properties, Formatting Entities

## Formatting Text

You can format text in TotalCAD using either the Text Format toolbar or the Text property sheet. The Text property sheet offers more options than the Text Format.

You can display and hide the Text Format toolbar via the Desktop property sheet, which you can access by choosing either View|Toolbars or Tools|Desktop. You can use Text Format toolbar either to set default properties for the Text tool (which will be saved with your drawing), or you can use it to format selected text. To change default settings, activate the Text tool on the Insert toolbar, then choose settings on the Text Property toolbar. These setting will stay in effect until you explicitly reset them. To format selected text, first select the text using the Select tool, then choose settings in the Text Format toolbar.

The Text Format toolbar has the following controls:



*Font*. Pull down this list box to set the font. You can use any TrueType font that is installed on your system.



*Size*. Pull down this list box to set the text size. (The size can be in either World or Paper units, depending on the current mode. Check the icon in the upper-left corner of the ruler.)



*Attributes*. Click the Bold, Italic, and Underline buttons to turn these text attributes on and off.



*Justification*. Click these buttons to choose whether the text is left, center, or right-justified in its frame. Left-justified text will be typed in to the right of the point where you placed the text; right-justified text extends to the left of this point; center-justified text will be centered on the point.



**See Also**

[Manipulating Text](#), [Text Tool](#), [Text Property Sheet](#)

## Groups

TotalCAD drawing objects can be combined into groups. The drawing objects can be entities, such as lines circles and arcs, or they can be other groups. TotalCAD treats a group as a single object for purposes of selecting and editing.

To break up a group into its constituent parts, select it, then use the Explode command.

The names of groups are held in the Attribute field of the properties dialog. To view or edit this field, activate the Select tool, then double-click on the group.

Because groups can themselves contain groups , you can think of them as complex *hierarchical* structures.

 **Tip**

 **See Also**

You can view the structure of a group insert by selecting it and exploring the Selection Info palette (View| Selection Info).


Create Group, Explode




## Snaps and Coordinates

Modes enable you to control how TotalCAD responds to your actions when you are drawing or editing. Some modes are mutually exclusive; others can be used in combination. This provides you with an enormous number of possible techniques, but the fundamental principles underlying all the modes are consistent and relatively simple. Once you understand these principles, you will be able to exploit them to find efficient and precise methods of drawing and editing.

There are two major categories of modes:

 [Snap Modes](#), which determine where exactly TotalCAD defines a point when you click the mouse in the drawing area.

 [Coordinate System Modes](#), which determine how you can use mathematical coordinates to specify the location of any point on the drawing.

## Snap Modes

Snap modes let you define points relative to objects in the drawing area when you are drawing or editing with the mouse. You can define points relative to drawing entities, construction lines, and the grid. All snap mode functions are similar: You first set the snap mode, then click on or near the object that you want to snap to. Rather than defining the point exactly where you clicked the mouse, TotalCAD will define the point precisely on the object. The point that TotalCAD chooses depends on the snap mode. You can, for example, snap to the end of a line, the midpoint of a line, the center of a circle, or a point on the grid.

[Snap Aperture](#)

[Local Snap](#)

## Snap Aperture

The critical distance between the point of the mouse click and an object or grid point is the *snap aperture*. You can think of the snap aperture as a circular area centered on the point of the mouse click. If an object containing a qualifying snap point is located within the snap aperture at the time of the mouse click, your point will be defined at the nearest qualifying snap point. If more than one object is located within the snap aperture, your point will still be defined at the closest qualifying snap point.

Note that in most cases the snap point itself does not need to be located within the snap aperture. (Snap to Grid and Snap to Near Point are obvious exceptions to this.)

 **Tips**

 **Notes**

 **See Also**

To see the snap aperture while you are working, choose View|Aperture.

To get additional visual feedback when using Snap modes, turn on the Magnetic Point tool. When Magnetic Point is activated, the rubber band line that provides you feedback about the entity you are creating is attracted to the snap point. This lets you see exactly where the point will be defined after you click the mouse.

You can set the radius of the snap aperture in the General property sheet of the Tools|Program Setup dialog.

Defining points with the status bar overrides the current snap mode and defines the point at precise coordinates that you specify.

## Program Setup Dialog

## Local Snap

You can change snap modes in three different ways. You can use the menu, the Snap toolbar, or the local menu. When you choose a mode from the menu or the Snap toolbar, the effect is the same: The snap mode changes until you reset it. The local menu is different. Use it for “one shot” mode changes, or *local snap*. This means that when you change snap modes in the local menu, the mode remains in effect until you define the next point. TotalCAD then switches back to the last mode you set in the main menu or toolbar, or defaults to No Snap mode.

This is a useful feature, because it lets you set the snap mode you use most frequently in the main menu or toolbar, then switch modes "on the fly" in the local menu when you are working in your drawing. Many users, for example, like to work with Snap to Grid as their default snap mode, so they set the mode to Snap to Grid in the main menu. While working in a drawing they may need to snap, for example, to the corner of a polygon. To do this, they change the snap mode temporarily to Vertex in the local menu and define a point on the polygon corner. TotalCAD then automatically switches back to Snap to Grid, the mode they most commonly use.

## **Coordinate Systems**

Coordinate systems provide different methods of specifying the location of any point in the drawing using coordinates. TotalCAD has three coordinate system modes: absolute, relative, and polar. Both the absolute and the relative coordinate modes are in the Cartesian coordinate system.

[Cartesian Coordinates](#)

[Polar Coordinates](#)

[Drawing with Coordinate Systems](#)



## Cartesian Coordinates

A *Cartesian coordinate* specifies the distance of a point from the X or Y axis along a line that is parallel to the other axis. (When the drawing area is labeled with Cartesian coordinates it is a *Cartesian plane*.) You can specify the location of any point on the drawing using two coordinates, the first being the distance from the Y axis on a line parallel to the X axis, the second being the distance from X axis on a line parallel to the Y axis. The intersection of the X and Y axes is the *origin* of the coordinate system; its coordinates are (0,0).

For example, to specify the location of a point that is 2 units measured horizontally from the Y axis and 3 units measured vertically from the X axis, you would use the coordinates (2,3). For locations to the left of the Y axis or below the X axis you need to use negative values. To specify a location 2 units to the left of the Y axis and 2 units below the X axis, you would use the coordinates (-2,-2).

[Absolute Coordinates](#)

[Relative Coordinates](#)

## Absolute Coordinates

When using absolute coordinates in TotalCAD, you specify the location of points relative to a location called the Absolute Origin, point (0,0) in the drawing plane. The Absolute Origin never changes location. By default, this point is in the lower-left corner of the page (although you can change the position of the page using in the Page Setup dialog).

 **See Also**

Page Setup Dialog

## Relative Coordinates

When you are working in relative coordinates in TotalCAD, you specify points on the drawing relative to a point that we'll call the Relative Origin. This point moves in the drawing space as you work. Suppose, for example, that you are drawing a multiline. As you define each point of the multiline, the point you just defined becomes (0,0). This enables you to draw each segment of the multiline relative to the last segment.

 **See Also**

Program Setup Dialog, Relocate Origin

## Polar Coordinates

In the polar coordinate system, you define the location of a point by its absolute distance and angle from the Relative Origin. As in the relative coordinate system, the Relative Origin follows you as you work.

 [See Also](#)

Relative Coordinates, Relocate Origin

## Drawing with Coordinate Systems

TotalCAD lets you change coordinate systems at any time while you are working. If you are drawing the outer wall of a house, for example, you may want to start the first wall at an absolute location in the drawing space, so you would use absolute coordinates for the first point. Each successive wall, however, would be defined by its length and angle relative to the first wall, so you might use polar coordinates for the remaining points.





Advanced users can "force" TotalCAD to use a coordinate system other than the current system while working in the Coordinate Fields. If you precede a coordinate with a \$ sign, it will be interpreted as an absolute coordinate; if you precede it with an @ sign it will be interpreted as a relative coordinate; if you precede it with a > sign it will be interpreted as a polar coordinate.

## Setting Options

The [Tools menu](#) gives you access to dialog boxes that let you control TotalCAD program and drawing settings. These features let you:



Set program options that are saved with the application.



Set drawing options that are saved with the drawing or template.

[Program and Drawing Setup](#)

[Setting Up Scale and Units](#)

[Setting Up the Grid](#)

[Setting Up Layers](#)

## Program and Drawing Setup

The Tools menu contains two large dialogs, Program Setup and Drawing Setup. Changes that you make in the Program Setup are saved whenever you exit TotalCAD: The next time you start TotalCAD, the settings will be exactly the ones you selected. Changes that you make in the Drawing Setup dialog are saved with your drawing, and they will come back exactly as you left them the next time you open the drawing in which they were saved.

If you have a standard way of setting up a drawing and you want to use the settings in other drawings as well, you can save the Drawing Setup options in a *template* using the Edit|Save As dialog. You can make the template that you use most frequently the *default template* (the template that TotalCAD chooses if you don't specify a template) in the Create from Template dialog. You can also create multiple templates for different types of CAD tasks.



To save the setup of your Desktop controls (that is, the display and position of toolbars and other interface controls), you must check the Save Desktop on Exit box in the General property sheet of the Program Setup dialog.

Create from Template Dialog, Drawing Setup Dialog, Program Setup Dialog, Save As Dialog

## Setting Up Scale and Units

TotalCAD has two distinct "spaces": world space and paper space. The units in world space apply to the real-world object that your drawing models. The units in paper space apply to the paper on which your drawing will be printed. *Scale* is the ratio of Paper units to World units. A scale of  $3/16" = 1'$ , for example, means that an object  $3/16"$  long in your drawing is modelling a real-world object that is one foot long.

Normally the coordinate system, the ruler, and the grid all show units in world space. You can switch to paper space by clicking the icon in the upper-left corner of the Ruler.

The Units and Scale property sheet gives you control over:



The type of units used in both paper space and world space.



The scale of your drawing.



The format and precision of the numbers in your drawing.

You can set these options either by choosing from one of TotalCAD's standard settings, which correspond to settings common to specific CAD disciplines, or you can specify custom settings.



**Note**



**See Also**

Dimensions always show World units, unless you manually override the dimension text.

Units and Scale Property Sheet



## Setting Up the Grid

The grid is a matrix, displayed as a pattern of dots, crosses, or lines, that divides your drawing geometrically into Cartesian coordinate locations. You can make the grid visible or invisible using the View|Grid command. You make the grid active or inactive by switching the Snap to Grid option on and off (using the Modes|Snaps|Snap to Grid menu option, the Snap toolbar, or the local menu). Note that you can snap to an invisible grid, and you can view an inactive grid: Whether the grid is visible has nothing to do with whether it is active.

TotalCAD has a *major grid* and a *minor grid*. The major grid shows larger divisions of your drawing space; the minor grid shows finer divisions of the drawing space between the major divisions. Using the Grid property sheet of the Drawing Setup dialog, you can specify how far apart the grid points are spaced, how many minor grid divisions there are between major grid points; and you can specify how frequently the grid points are visible. This lets you make only larger grid divisions are visible, but still enables you to snap to the invisible finer divisions.

 **See Also**

[Grid Property Sheet](#), [Snap to Grid](#), [View Grid](#)

## Setting Up Layers

One of the most powerful ways you can organize a complex drawing is to create layers on which to place different categories of object. Layers are like acetate sheets used in traditional drafting.

Layers in TotalCAD can be visible or invisible, editable or not editable. These properties allow you to view only the drawing objects that are relevant to the current task, and to protect objects on the drawing from inadvertent change.

You can also distinguish layers of the drawing by giving them distinct colors and line styles. You can assign colors and styles to layers in the Layers property sheet. Objects on the layer will then have the color and line style assigned to the layer, provided the relevant properties of the object are set to By Layer. For example, if you want an object's pen color to have the default color of the object's layer:

1. Double-click on the the object with the Select tool to go to the object's property sheets.
2. Bring the Pen property sheet to the front.
3. Set the Color property to By Layer.
4. Click <OK> to close the dialog.

 **See Also**

Layers Property Sheet

## Working with Multiple Documents

TotalCAD uses a *multiple document interface*. This means that the TotalCAD application window can contain multiple document windows, each of which contains a TotalCAD drawing. Whenever you open a TotalCAD drawing or start a new TotalCAD drawing, the drawing will appear in its own window, within the TotalCAD application window.

If you are familiar with other Microsoft Windows programs, you already know how to work with windows and minimized window icons. For instruction on basic techniques, such as moving, sizing, restoring, maximizing, and minimizing document windows under Windows 3.1 and Windows 95, see your Microsoft Windows documentation.

One big advantage of the multiple document interface is that it allows you to intuitively move and copy objects from one drawing to another.

[Moving and Copying Objects between Drawings](#)

## Moving and Copying Objects between Drawings

There are two basic methods for copying and moving objects between drawings: copy and paste (or cut and paste), and OLE drag and drop.

[Copying/Cutting and Pasting Objects between Drawings](#)

[Drag and Drop](#)

 **Note**

 **Tip**

In describing how to use these techniques, we'll refer to the drawing from which you'll be copying or moving objects as the *source document*; we'll refer to the drawing into which you'll be moving or copying the objects as the *target document*.

Before dragging and dropping objects, it's a good idea to choose the Tile command from the Window menu. This will arrange the windows on the screen so that they are both completely visible, without overlap.



## **Copying/Cutting and Pasting Objects between Drawings**

You can use the Windows clipboard to transfer objects between drawings, regardless of whether both of the drawings are open at the same time. Cut and paste allows you to remove an object from the source document at the same time that you paste the object into the target document.

1. Select the object or objects in the source document that you want to copy. You can use any of the select tools (available from the Select toolbar), or any of the Select commands from the Edit menu.
2. Choose Edit|Copy (<Ctrl>+<C>), or Edit|Cut (<Ctrl>+<X>) from the menu. This places a copy of the objects on the Windows clipboard. (Edit|Cut will remove the selected objects from the source document.)
3. Activate the target document by clicking in its window, or by choosing it from the Window menu.
4. Choose Edit|Paste from the menu, or press <Ctrl>+<V>. The object will be pasted from the Windows clipboard into the target document.

The objects that you pasted from the clipboard will appear at the center of the target document window.

## **Drag and Drop**

OLE drag and drop provides a simple and intuitive way to copy objects between drawings. You can drag objects from one drawing window into another, and you can drag objects stored in libraries and symbols from their library palette into another drawing.

[Between Drawings](#)

[Dragging and Dropping Library Objects between Drawings](#)

## Between Drawings

To drag a drawing object from one drawing into another:

1. Select the object or objects that you want to drag into another drawing.
2. Place the cursor over the reference point of the selection. This is a movable handle located by default at the center of the selection. You will know that the cursor is in the right place when it turns into a four-directional pointer.
3. Press and hold down the mouse button while dragging the selected objects into the window of the target document. An outline of the selection will follow the mouse cursor as you drag.
4. Release the mouse button to drop the selected objects. A copy of the selected objects will appear at the point where you released the mouse button.



**Warning**

If you drag an object by any point other than the reference point, you will only be able to drag the object within the window of the source document. Make sure that the mouse cursor is placed over the reference point of the selection before attempting to drag the selection into another window.

## Dragging and Dropping Library Objects between Drawings

You can also drag and drop objects from a drawing's libraries into another document. Moving a symbol to another drawing accomplishes two things: It inserts the symbol into the target drawing, and it places the symbol itself into the corresponding library of the target document.

1. Open the symbol palette using View|Symbol.
2. Place the mouse cursor over the symbol in the palette that you want to copy, then hold down the mouse button while you drag the symbol into the target document window.
3. Release the mouse button. The symbol will appear in the target document at the place where you released the mouse button. The symbol will also appear in the target document's symbol palette.



After you have moved the object, the target document becomes the active window. Notice that the symbol palette will change, displaying the contents of the target document's symbol library.

Paper units are the unit of measure for the paper, and for the drawing objects on the paper at their printed size.

World Units are the units of measure for the "real world" objects that your drawing objects represent.





## Command Reference



[File Menu and Dialogs](#)



[Edit Menu and Tools](#)



[View Menu and Tools](#)



[Insert Menu and Tools](#)



[Format Menu and Tools](#)



[Modify Menu and Tools](#)



[Modes Menu and Tools](#)



[Tools Menu and Dialogs](#)



[Windows Menu](#)

## File Menu and Dialogs

### [New](#)

Start a new TotalCAD drawing.

### [Open Dialog](#)

Open a drawing in one of several file formats.

### [Close](#)

Close the current drawing.

### [Save](#)

Save the current drawing to disk.

### [Save As Dialog](#)

Save the current drawing under a new name.

### [Save All](#)

Save all open drawings.

### [Libraries](#)

Bring up the Symbol Libraries property sheet of the Drawing Setup dialog.

### [Summary Info Dialog](#)

View or edit information about the current drawing.

### [Page Setup Dialog](#)

Choose options for printing your drawing.

### [Print Dialog](#)

Print the current drawing.

### [Send](#)

Send current drawing over network using Microsoft Mail.

### [Recent File List](#)

Re-open one of the four most recently used files.

### [Exit](#)

Close all open drawings and exit TotalCAD.



## **New**

**Menu:** File|New      **Hotkey:** <Ctrl>+<N>      **Toolbar:** Standard

Use the New command to start a new TotalCAD drawing.

[Create from Template Dialog](#)

## Create from Template Dialog

Templates are a special type of drawing file that provides a basis for a new drawing. Templates preserve all of the settings that are saved with the drawing, such as unit and scale options, angle conventions, grid settings, and colors, as well as symbol libraries. If the Prompt for Template Name option in the General property sheet of the Program Setup dialog is checked when you choose the New command, then the Create from Template dialog will be displayed.

When the dialog opens, it will display the templates that are stored in the current TotalCAD Template folder. Click on a template then click OK to use the template as a basis for your drawing. TotalCAD will then run a wizard that will assist you in making additional choices based on the template you have chosen.

The Open a Template dialog contains these additional controls:



*Don't Ask Again.* Check this option if you want to bypass when you start a new drawing.



*Set Default.* Click this button if you want the template you are now opening to become the default template (that is, the template that will be used if no other template is chosen).



*Show Preview.* Check this box if you want TotalCAD to display a preview of the template in the Preview box.



*Description.* This area displays information about the currently selected template.



**Tip**



**Note**



**See Also**

You can change the default template directory using the File Locations property sheet of the Program Setup dialog.

If you create a single template that you want to use most of the time, check both the Don't Ask Again and Use as Default options, then open the template. The new template will become the default, and the dialog will no longer be displayed when you start a new drawing.

If you choose Don't Ask Again in the dialog box, you can reverse this setting in the General property sheet of the Program Setup dialog. Choose Tools|Program Setup and check the Prompt for Template Name check box.

Program Setup Dialog, Save As Dialog





## Open Dialog


**Menu:** File|Open      **Hotkey:** <Ctrl>+<O>      **Toolbar:** Standard


The Open command displays the Open dialog, which allows you to open an existing TotalCAD drawing, or a CAD drawing in any of several different standard formats. The appearance and features of the Open dialog differ significantly in Windows 3.1 and Windows 95.


This is a variation of the Windows Open dialog, in which you use standard Windows navigation to choose a template. To open a drawing:

1. Use standard Windows controls to navigate to the directory where the file is located.
2. Use the Files of Type control to choose the file types to display in the File Name list box. If you do nothing with this control, then only standard TotalCAD drawing files (.TCW) will be displayed.
3. Select the file that you want to open, then click Open (or simply double-click on the file you want to open). In Windows 3.1, click OK.

The Open dialog also provides the following controls:

 *Read Only.* Check this option if you wish to view a file while making sure that you don't inadvertently change it.

 *Description.* This area displays text saved in the Subject field of the Summary Info dialog. You can place information in this field to help you identify and categorize your drawings.

 *Show Preview.* Check this option to display a thumbnail of the drawing you have selected in the preview area of the dialog.

 **See Also**

[Save As Dialog, Supported File Types](#)

## Close

**Menu:** File|Close

The Close command closes the current file. If the file has been changed since it was last saved, TotalCAD will ask you whether you want to save the file before closing.

 **See Also**

Close All



## Save

**Menu:** File|Save      **Hotkey:** <Ctrl>+<S>      **Toolbar:** Standard

The Save command saves the current drawing to disk, using the directory and file name in which the file was previously saved. If this is the first time you have saved the file, this command will display the Save As dialog, prompting you for a name and location for the file.

[Tip](#)

[See Also](#)

To create a new copy of the file, use Save As.

Save As Dialog

## Save As Dialog

**Menu:** File|Save As

The Save As command displays the Save As dialog, which lets you save the current drawing using a file name and directory that you specify. The appearance and function of the Save As dialog differ significantly in Windows 3.1 and Windows 95.

This is a standard Windows Save dialog. To save a drawing in a specific directory and under a specific file name:

1. Use the Drive and Directory controls to navigate to the directory where you want to save the file.
2. Type a name for the file in the File Name text box.
3. Use the List Files of Type control to choose the type of file you want to save. If you do nothing with this control, then the file will be saved as a normal TotalCAD (.TCW) file.
4. Select the file that you want to open, then press <Enter> to open the file.





Other graphics programs are most likely to be able to read your TotalCAD drawing if it is saved as a .DXF file. This has become the standard file exchange type for CAD drawings.

[Save](#), [Save All](#), [Supported File Types](#)

## Save All

**Menu:** File|Save All

The Save All command saves all open drawing files. If any of the files have not been previously saved, TotalCAD will prompt you for a directory and file name for those files.

 **See Also**

Save, Save As Dialog



## Libraries

**Menu:** File|Libraries    **Toolbar:** Standard

The Libraries command brings up the Symbol Libraries property sheet of the Drawing Setup dialog.

 **See Also**

Symbol Libraries Property Sheet

## Summary Info Dialog

**Menu:** File|Summary Info

The Summary Info command opens the Summary Info dialog, in which you can record information that will be saved with your drawing file.

Use the text fields of the dialog box to record information about your drawing. The contents of these fields could be useful in helping you to identify drawings or versions of drawings. It can also potentially be used by Windows file search utilities. The Save Preview Graphics with Drawing options let you choose the type of graphic image that will be stored with the drawing, so that it can be previewed in TotalCAD and in other Windows programs:

*None.* Choose this option if you do not want preview graphics to be saved with your drawing.

*Bitmap.* Choose this option to store the preview graphic as a bitmap. This option is better suited to large files because it displays less detail.

*Metafile.* Choose this option to store the preview graphic as a Windows metafile. This option is better suited to small files because it displays more detail.

## Page Setup Dialog

**Menu:** File|Page Setup

The Page Setup dialog provides a rich set of features that let you determine exactly how drawings will be printed. It consists of two property sheets: [Paper](#) and [Layout](#).



## Paper Property Sheet

The Paper property sheet lets you determine the size and orientation of the printer paper and the drawing sheet. The printer paper is the physical paper that goes through the printer; the drawing sheet is the area on which your drawing will be printed, which can span multiple sheets of printer paper. This property sheet also contains a variety of option controls and a function for viewing or setting the scale of the drawing.

[Printer Paper](#)

[Drawing Sheet Size](#)

[Options \(Printing Only\)](#)

[Page Layout Scale](#)

[Keep Centered](#)

 **See Also**

Page Setup Dialog, Printer Paper and the Drawing Sheet, Layout Property Sheet

## **Printer Paper**

The Printer Paper control group lets you set the size and orientation of the printer paper. These are the same settings that you can control through the Print Setup dialog, so changes that you make in this dialog will be reflected in Print Setup, and vice versa. To set the paper size and orientation:

1. Click in the Size list box and choose a standard paper size.
2. Choose either portrait or landscape orientation by clicking one of the option buttons. The icon to the left of the option buttons will change to reflect your choice.

Clicking the Setup button in this control group brings up the Print Setup dialog, in which you can choose another printer or access functions that directly control your printer.

## Drawing Sheet Size

The Drawing Sheet Size control group lets you control the size and orientation of the area on which your drawing will be printed. This area can be imposed on a single sheet of printer paper, or it can span multiple sheets of printer paper. Its orientation can also be set separately from the orientation of the printer paper. For a description of how printer paper and drawing sheet size are related, see Printer Paper and the Drawing Sheet. To set the size and orientation of the drawing sheet:

1. Click in the Size list box and choose a standard paper size, or type in a custom size using standard measurements (for example **2 ft x 2 ft**).
2. Choose either portrait or landscape orientation by clicking one of the option buttons. The icon to the left of the option buttons will change to reflect your choice.

The page display on the right will change to reflect the relationship between printer paper and drawing sheet size and orientation.

### ***Options (Printing Only)***

The buttons in the Options (Printing Only) control group let you determine what elements are printed. These options affect only the printing of the drawing and are not reflected in the appearance of the drawing on screen.

*Print margins.* Check this button to print a border on the margin of the printer sheets. This is useful for framing the drawing if it is printed on a single sheet.

*Print crop marks.* Check this option button to print crop marks on the printer sheets. This is useful if you want to print across multiple printer sheets and crop the sheets so that they can be pasted against a backing.

*Print grid.* Check this option if you want to print grid marks. The grid will be printed if it is currently displayed.

*Print construction.* Check this option if you want to print constructions lines.

## ***Page Layout Scale***

The controls in the Scale control group let you scale the drawing so that it fits within the drawing sheet or choose a custom scale.

To scale the drawing so that it fits into the drawing sheet, click the Fit button. If the Keep Centered option is turned on, the drawing will be centered as it is fit into the drawing sheet.

To choose a custom scale, click in the list box and choose a pre-set scale, or type a scale directly into the list box in the form **1 in = 1 ft**. When typing custom scales, you can use any measurement available in the Units and Scale property sheet of the Drawing Setup dialog (available from the Tools menu). You can click the Reset button at any time to return the scale that was set when you entered the Page Layout dialog.

### ***Keep Centered***

Check the Keep Centered option to keep the drawing centered over the drawing sheet.

## ***Layout Property Sheet***

The Layout property sheet gives you direct control of the number of rows (sheets of printer paper arranged vertically) and columns (sheets of printer paper arranged horizontally). It also has functions for setting the printer paper margins, setting the height and width of the drawing sheet in either World or Paper coordinates, and relocating the paper in the drawing area.

[Margins](#)

[Rows and Columns](#)

[Origin](#)



## Margins

The margins control group contains four text boxes in which you can specify the top, bottom, left, and right margins of the printer paper in Paper units. By default, the margins are displayed in units set in the Units and Scale property sheet, but you can enter values in other units as well.



You cannot set the margins of the drawing sheet independently of the margins of the printer paper.

## **Rows and Columns**

The Rows and Columns controls let you set the number of rows and columns of printer sheet paper, for tiled printing. As you increase the number of rows and columns, the size of the drawing sheet—the area on which your drawing will be printed—increases correspondingly. The Paper Height and Paper Width controls let you directly set the height and width of the drawing sheet. As you increase these values, TotalCAD will automatically increase the number of rows and columns to accommodate the change in the drawing sheet size.

## Origin

The Origin controls let you re-set the position of the lower-left corner of the page in the drawing area. To use the control, enter X (horizontal) and Y (vertical) absolute coordinates in the text boxes, using the current World units.

 **Tip**

 **See Also**

This feature is useful if you want to print a particular area of a large drawing by relocating the paper to the area that you want to print.

Coordinate Systems, Units and Scale Property Sheet



## Print Dialog

**Menu:** File|Print      **Hotkey:** <Ctrl>+<P>      **Toolbar:** Standard

The Print dialog provides a set of functions which include printing a single view of a drawing, or printing specific sheets in a drawing that spans multiple sheets of printer paper.

[Print Source](#)

[Print Range](#)

[Print Quality](#)

[Copies](#)

[Print Setup Dialog](#)

**See Also**

Page Setup Dialog, Views of the Drawing Space



## Print Source

The Print Source control group lets you choose whether to print all of the drawing objects, or only those that are currently selected.



You can control the printing of other elements, such as crop marks, margins, and constructions, in the Paper property sheet of the Page Layout dialog.

## Print Range

The Print Range control group provides options for setting a specific view of the drawing to print, and for selecting a range of pages when printing a drawing that spans multiple printed sheets.

*All.* Select this option if you want to print all of the current drawing. If you have chosen a Selection in the Print Source control group, only objects that are selected will be printed. (If your drawing spans multiple printer sheets, the total number of sheets will be shown in this option label.)

*View.* Select this option to print a single view of the drawing. You can print the current view by default, or click in the list box and choose any named view.

*Range.* Select this option to print a range of pages if the Page Layout dialog is set up to print multiple pages. Specify the first page to be printed in the left text box, and the last page to be printed in the right text box. TotalCAD counts the printer sheets starting in the lower-left corner from left to right, and from bottom to top.

*Fit on 1 page.* Check this box if you want to print everything on a single page. This option is compatible either with printing a view, or with printing the entire drawing. You can't have this option checked and the Range option turned on at the same time.

## **Print Quality**

The contents of this list box will vary, depending on settings available in your current printer. In some cases the list box may contain specific settings, such as 300 or 600 DPI (Dots Per Inch). In other cases, the list box may contain High, Medium, Low, and Draft settings. Higher quality print settings produce finer resolution; lower quality print settings print out more quickly.

## **Copies**

Type the number of copies of the drawing that you want to print in the copies text box.

## **Print Setup Dialog**

The Setup button brings up the Print Setup dialog. This is a Microsoft Windows dialog, and its appearance will vary depending on the version of Windows that you are running.

The Print Setup dialog enables you to choose a printer, if there are multiple printers available, as well as to set the paper size, paper source, and paper orientation. The contents of the dialog will vary, depending on the current printer.

Under Windows 3.1 the Options button will bring up a dialog through which you can control your printer. If you are running Windows 95, the Properties button brings up a set of property sheets through which you can control your printer.

For more information, see your Windows documentation and the documentation for your printer.

## Send

**Menu:** File|Send

The Send command is a standard Microsoft Office function. If Microsoft Mail is installed on your system, or if you are running Windows 95 on a network, this command will open a new message containing the current document as an attachment. You can then send the document over your network via Microsoft Mail or Microsoft Exchange.

## Recent File List

Below the Send command on the File menu, TotalCAD displays the four most recently opened drawings. These can be drawings that are currently open but not active, or they can be drawings that were recently saved, but which are not currently open.

Choosing a drawing from this list makes it the currently active drawing. If the drawing is already opened, TotalCAD will make it active; if the drawing is not opened, TotalCAD will open the drawing.

 **Warning**



If you rename a drawing or move it to another directory using File Manager or another utility, the file name will remain in the recent file list, but TotalCAD will not be able to locate the file.

## Exit

**Menu:** File|Exit **Hotkey:** <Ctrl>+<X>

Use the Exit command to close all open drawings and exit TotalCAD. TotalCAD will warn you about any files that have been changed since they were last saved, giving you an opportunity to save them.

## **Edit Menu and Tools**

The following table summarizes the commands accessible from the Edit menu:

### Undo

Reverse previous action.

### Redo

Reverses the effect of the previous Undo command.

### Cut

Remove selected objects and place them on Windows clipboard.

### Copy

Copy selected objects to clipboard.

### Paste

Paste objects into drawing from clipboard.

### Paste Special

Paste objects from clipboard as data or as OLE objects.

### Clear Commands

Remove selection, all objects, a single construction line, or all construction lines from drawing.

### Select Commands

Activate the Select tool.

### Copy Entities Tools

Create multiple copies of objects along lines, around arcs, in two-dimensional arrays, and more.

### Edit Links

Make changes to linked OLE objects.

### Edit Object

Edit OLE object using its source application.



## Undo

**Menu:** Edit|Undo      **Hotkey:** <Ctrl>+<Z>      **Toolbar:** Standard

The Undo command reverses the previous action, restoring your drawing to its state prior to your last editing or drawing command. The name of this command will change, depending on the specific action that it undoes.

 **See Also**

Redo



## Redo

**Menu:** Edit|Redo      **Hotkey:** <Ctrl>+<A>      **Toolbar:** Standard

The Redo command reverses the effects of an Undo command. The name of this command changes, depending on the specific action that it redoes.

 **See Also**

Undo



## Cut

**Menu:** Edit|Cut      **Hotkey:** <Ctrl>+<X>      **Toolbar:** Standard

The Cut commands removes any selected object or objects from your drawing and places them in the clipboard. From the clipboard you can paste them into TotalCAD drawings or into other Windows applications using the Paste or Paste Special commands.

**See Also**



Copy, Paste, Paste Special, Copy Entities Tools



## Copy

**Menu:** Edit|Copy      **Hotkey:** <Ctrl>+<C>      **Toolbar:** Standard

The Copy commands copies any selected object or objects from your drawing and places them in the clipboard. From the clipboard you can paste them into TotalCAD drawings or into other Windows applications using the Paste or Paste Special commands.

**See Also**

Cut, Paste, Paste Special, Copy Entities Tools



## Paste

**Menu:** Edit|Paste      **Hotkey:** <Ctrl>+<V>      **Toolbar:** Standard

The Paste command pastes into your drawing any data that has been copied or cut into the clipboard. You can paste TotalCAD drawing objects, as well as text, bitmap, or Windows metafile data from that has been placed in the clipboard in other programs.




## Paste Special

**Menu:** Edit|Paste Special

The Paste Special command displays the Paste Special dialog.

The Paste Special dialog lets you choose how to paste into your drawing an object that was previously cut or copied to the clipboard. The object on the clipboard can be from TotalCAD, or it can come from another application. The options available in the Paste Special dialog depend on the type of object in the clipboard, and on the program in which the object was created (the *source* application).

When you open the Paste Special dialog, you will see a number of choices in the As list box. For example, if there is a document on the clipboard created in Microsoft Word, the As text box will contain the following options:

-  Word Document
-  Picture (Metafile)
-  Text

If you choose Word Document, the data in the clipboard will be inserted into your drawing as an OLE object, which you can edit within TotalCAD using Microsoft Word features. If you choose Picture, the data will be inserted as an image. If you choose Text, the data will be inserted as text, creating a TotalCAD text object, which you can treat exactly like text created with the TotalCAD Text tool.

If you choose to insert a clipboard object into your drawing as an OLE object, you have a choice of either *linking* or *embedding* the object. If you click the Paste option button, you will embed the OLE object in your drawing. This means that the drawing will have its own independent copy of the OLE object: Any changes you make to this object will *not* affect the file on which it was based. If you activate the Paste Link option button, the OLE object will be *linked* to the file on which it was based. This means that any changes made to the file will be reflected in the OLE object, and any changes made to the OLE object will also affect the file.

If you check the Display as Icon check box, the OLE object will appear in your drawing as an icon. (When this box is checked, you can click on the Change Icon button to select another icon.)

 **Note**

 **Tip**

 **See Also**

There are only two types of data that can be inserted into TotalCAD as an editable TotalCAD drawing object. These are text data, which is inserted as a TotalCAD text object, and actual TotalCAD drawing entities that have been copied onto the clipboard from TotalCAD drawings. All other clipboard objects will be pasted either as OLE objects, or as pictures (either bitmaps or metafiles).

If you plan to insert multiple copies of a single object into your drawing, you can keep your drawing file from growing too large by linking the objects rather than embedding them. This is possible because each linked object is simply a reference to the file on which it is based. If you link the object multiple times, each copy of the object shares the same source of data.

[Edit Links](#), [Edit Object](#), [Insert Object](#), [Insert Picture](#), [Paste](#)



## Clear Commands

**Menu:** Edit|Clear

The Clear commands let you remove graphic objects and other objects from a drawing. The Edit|Clear menu item opens a submenu containing four options:

[Clear Selection](#)

Clear all objects that are currently selected.

[Clear All](#)

Clear all objects in the drawing.

[Clear Construction](#)

Activate the clear construction tool, with which you can delete individual construction lines.

[Clear All Constructions](#)

Clear all construction lines in the drawing.

You can reverse any clear with the Undo command.

## Clear Selection

**Menu:** Edit|Clear|Selection      **Hotkey:** <Del>

Use the Clear Selection command to delete any graphics or other objects that have been selected. If no objects have been selected, then this command is unavailable.

 **See Also**

Select Commands, Cut

## **Clear All**

**Menu:** Edit|Clear|All

Use the Clear All command to delete everything on the drawing space and begin a new drawing.

## **Clear Construction**

**Menu:** Edit|Clear|Construction

The Clear Construction command activates the Clear Construction tool, with which you can delete individual construction lines. To use it, first activate the tool, then click on any construction that you wish to delete. Choose Cancel from the local menu or switch to another tool when you are finished.

## **Clear All Constructions**

**Menu:** Edit|Clear|All Constructions

Use the Clear All Constructions command to delete all construction lines from your drawing.

## Select Commands

**Menu:** Edit|Select

If you want to perform an editing action on an object or a set of objects in your drawing, you generally need to select the objects first. When you select objects, they change to a pale magenta color and appear inside a frame with handles.

### Select

Activate the selection tool.

### Select All

Select all objects in the drawing.

### Select By Commands

Access a submenu that lets you select objects that by their type, layer, attribute, or with a freeform "fence."



## Select

**Menu:** Edit|Select      **Toolbar:** Insert Entity

The Select tool lets you select and deselect objects in the drawing area using the mouse. You can select a graphic entity (as well as an OLE object or bitmap) by clicking on it, or you can select objects using a selection rectangle. Once objects are selected, you can deselect everything by clicking the mouse anywhere in the drawing area outside of the selection rectangle. You can also deselect everything by pressing <Esc> or choosing Cancel from the Local menu.

[Selection by Clicking](#)

[Selection within Rectangle](#)

 **Warning**



If you try to deselect everything by clicking inside the selection rectangle, nothing will happen: You need to click outside the selection rectangle. Normally this is not a problem, but it might be confusing if you are zoomed in so that your view is *inside* the selection rectangle. If you find yourself in this situation just press <Esc> or choose Cancel from the Local menu to deselect everything.

## **Selection by Clicking**

To select any object, simply activate the Select tool, then click on the object.

To select a series of objects, adding them one at a time to the selection, hold down the <Shift> key while clicking on each object in succession. To deselected multiple objects one at a time, hold down the <Shift> key and click on each selected object in succession.

## **Selection within Rectangle**

The selection rectangle gives you different results, depending on whether Open Window mode is on or off. To change this option, click the right mouse button in the drawing area to open the Local menu, then choose Open Window mode. If the menu item has a check next to it, the option is on; otherwise the option is off.

To use the selection rectangle with the Open Window option on, hold the left mouse button down and move the cursor, creating a rectangle over an area that contains *any part* of the objects that you want to select, then release the mouse button. If any portion of an object is inside the rectangle, the entire object will be selected.

To use the selection rectangle with the Open Window option off, hold the left mouse button down and move the cursor, creating a rectangle over an area that contains the objects that you want to select *in their entirety*, then release the mouse button. If any portion of an object is outside the rectangle, it will not be selected.

## **Select All**

**Menu:** Edit|Select All

Use the Select All command to select all objects that are currently on the drawing.

## Select By Commands

**Menu:** Edit|Select By

Select By opens a submenu of special selection commands:

[Select by Entity Type Dialog](#)

Brings up a dialog that lets you select objects by their type.

Brings up a dialog that lets you select all the objects on a layer or subset of layers.

[Select by Fence](#)

Activates a tool that lets you select a set of objects by drawing a polygonal "fence" around them.

## Select by Entity Type Dialog

**Menu:** Edit|Select By|Entity Type

The Select By Entity Type command opens a dialog that lets you select all the objects in the drawing that belong to specific types.

You can use this dialog to select objects that belong to any type or combination of types, using common Windows list box selection techniques. For example, if you wish to select all linear and angular dimensions in your drawing, you would hold down the <Ctrl> key while clicking on Linear Dimension and Angular Dimension, highlighting both items. If you wish to add objects of these types to a current selection, place a check in the Add to the Current Selection check box. If this box is not checked, then any currently selected items will be deselected as the new objects are selected.

## Select by Fence

**Menu:** Edit|Select By|Fence

The Select by Fence tool lets you select a set of objects by drawing a polygonal "fence" around them.

1. Activate the Select by Fence tool.
2. Define a series of points by clicking and moving the mouse, using any snap mode. Each point you define will become a new vertex (corner) of an irregular polygon fence. Make sure that any objects you wish to select are contained within the fence.
3. Right-click in the drawing area and choose Finish to complete the fence and select the objects.

 **Note**

 **See Also**

Like the Select tool, the Select by Fence tool responds to the state of the Open Window Mode option. You can change in the Local menu after right-clicking in the drawing area. If Open Window Mode is on, then any objects that are partially contained within the fence will be selected. If Open Window Mode is off, then only objects that are entirely contained within the fence will be selected.



Selection within Rectangle

## Copy Entities Tools



**Menu:** Edit|Copy Entities

**Toolbar:** Copy

TotalCAD's tools for copying entities enable you to make single or multiple copies of entities, and at the same time precisely control the placement of the copies.

### [Linear Copy](#)

Make copies along a line, specifying the interval between each copy.

### [Radial Copy](#)

Arrange copies in an arc, specifying the angular distance between each copy and the rotation of each copy relative to the previous copy.

### [Array Copy](#)

Copy entities in a two-dimensional array, specifying the distance between each entity.

### [Fit Linear Copy](#)

Make copies along a line, specifying the total distance into which all the copies fit.

### [Fit Radial Copy](#)

Arrange copies in an arc, specifying the total angular distance in which all the copies must fit, and the rotation of each copy relative to the previous copy.

### [Fit Array Copy](#)

Copy entities in a two-dimensional array, specifying the dimension of the rectangle into which they fit.

### [Mirror Copy](#)

Create a mirror image of an object on the opposite side of a line that you specify.

### [Vector Copy](#)

Make a single copy of an object, specifying the angle and distance of the copy from the original.

 **Tip**

 **Note**

When you are using the Copy Entities tools, you may often find it necessary to switch back and forth between the Edit Bar and the drawing area. One reason for this is to use the Edit Bar to specify the number of copies that you want to make. Remember that you can move to the Edit Bar without moving the mouse by pressing <Tab>. You can return to the drawing space from the Edit Bar, with the cursor in exactly the position you left it, by either clicking once in the drawing area, or by pressing <Ctrl>+<D>.

If you do not change the rotation or scale of the copies that you create, TotalCAD will include the original copy in the number of sets. If you do rotate or scale the copies, the original will *not* be included in the number of sets. For example, if you specify a new scale or rotation and specify 20 sets, TotalCAD will make 20 copies of the original. If you do not specify a new scale or rotation, TotalCAD will make 19 copies, using the original as the 20th copy.



## Linear Copy

**Menu:** Edit|Copy Entities|Linear

**Toolbar:** Copy

Use Linear Copy to create one or more copies of an object along a straight line, specifying the distance between each copy, the angle of the line along which the copies will be made, and the number of copies.

1. Select the object or objects that you want to copy.
2. Choose the Linear Copy tool.
3. Press <Tab> or click in the Edit Bar to activate it
4. Type these values into the Edit Bar fields:



Optionally type values into XScale, YScale, and Rotation if you want to scale and rotate at the same time as you copy.



Type values into XStep and YStep to specify the horizontal and vertical distance of each copy from the previous copy.



Type the number of copies you want to make into Sets.

5. Press <Enter> to create the copies.



**Note**





You can also specify the XStep and YStep values by defining a point relative to the selection in the drawing area using the mouse or Coordinate Fields. However, the number of sets, as well as values for rotating and resizing the copies, must be specified in the Edit Bar first, unless you want to accept the current values.



## Radial Copy

**Menu:** Edit|Copy Entities|Radial **Toolbar:** Copy

Use Radial Copy to arrange one or more copies of an object in an arc, specifying the angular distance from one copy to the next.

1. Select the object or objects that you want to copy.
2. Choose the Radial Copy tool.
3. Define a point for the centerpoint of the arc around which you want to arrange the copies. The radius of the arc is the distance from this point to the reference point of the selection. This arc is visible as a rubber-band circle in the drawing area.
4. Press <Tab> or click in Edit Bar to activate it, then enter the following values into the Edit Bar fields:
  -  Optionally type values into XScale, YScale, and Rotation if you want to scale and rotate at the same time as you copy.
  -  Type the number of copies you want to make into Sets. If you have scaled or rotated the copies, the original will not be included in the number of sets. TotalCAD will choose appropriate default values for Angle and +Rot based on the number of sets you enter.
  -  Optionally type a value into the Angle text box to specify the angular distance between the copies.
  -  Optionally type a value into +Rot to specify the rotation of each copy relative to the previous copy.
5. Press <Enter> to create the copies.

### Notes

You can also specify Angle by defining a point within the rubber band circle in the drawing area using the mouse. However, the number of sets, the +Rot rotation value, as well as values for rotating and resizing the copies, must be specified in the Edit Bar first, unless you want to accept the current values.

The Sets, Angle, and +Rot values are interdependent. Changing the number of sets will change the values in Angle and +Rot; changing Angle will affect +Rot. These default ratios are designed to give you an elegant, symmetrical radial copy. Once you have entered values in these fields, however, the automatic interdependency is switched off, and you can enter any values you like.








## Array Copy

**Menu:** Edit|Copy Entities|Array **Toolbar:** Copy

Use Array Copy to arrange copies of selected objects into a two-dimensional array (rows and columns). You specify the horizontal and vertical distance between the copies.

1. Select the object or objects that you want to copy.
2. Choose the Array Copy tool.
3. Press <Tab> or click in the Edit Bar to activate it, then enter the following values in the Edit Bar fields:
  -  Optionally type values into XScale, YScale, and Rotation if you want to scale and rotate at the same time as you copy.
  -  Type values in the XStep and YStep fields to specify the horizontal and vertical distance between the centerpoints of the copies.
  -  Type values into the Rows and Columns fields to specify the number of rows and columns in the array.
4. Press <Enter> to create the copies.

 **Note**




You can also specify the XStep and YStep values by defining a point relative to the selection in the drawing area using the mouse or Coordinate Fields. However, the number of rows and columns, as well as values for rotating and resizing the copies, must be specified in the Edit Bar first, unless you want to accept the current values.



## Fit Linear Copy

**Menu:** Edit|Copy Entities|Fit Linear      **Toolbar:** Copy

Use Linear Fit Copy to create one or more copies of an object along a straight line, specifying a length into which all the copies will fit.

1. Select the object or objects that you want to copy.
2. Choose the Linear Copy tool.
3. Press <Tab> or click in the Edit Bar to activate it, then enter these values into the Edit Bar fields:
  -  Optionally type values into XScale, YScale, and Rotation if you want to scale and rotate at the same time as you copy.
  -  Type values into XSize and YSize to specify the length of the line onto which all the copies will fit. The copies will be placed on this line an equal distance apart.
  -  Type the number of copies you want to make into the Sets field. If you have scaled or rotated the copies, the original will be included in the number of sets.
4. Press <Enter> or create the copies.





 **Note**

You can also specify the XSize and YSize values by defining a point relative to the selection in the drawing area using the mouse or Coordinate Fields. However, the number of sets, as well as values for rotating and resizing the copies, must be specified in the Edit Bar first, unless you want to accept the current values.



## Fit Radial Copy

**Menu:** Edit|Copy Entities|Fit Radial      **Toolbar:** Copy

1. Select the object or objects that you want to copy.
2. Choose the Radial Fit Copy tool.
3. Define a point for the centerpoint of the arc around which you want to arrange the copies. The radius of the arc is the distance from this point to the reference point of the selection. This arc is visible as a rubber-band circle in the drawing area.
4. Press <Tab> or click in Edit Bar to activate it, then type the following values in the Edit Bar fields:
  -  Optionally type values into XScale, YScale, and Rotation if you want to scale and rotate at the same time as you copy.
  -  Type the number of copies you want to make into the Sets field.
  -  Optionally type a value into the Angle text box to specify the angular distance into which all the copies will fit.
  -  Optionally type a value into the +Rot field to specify the rotation of each copy relative to the previous copy.
5. Press <Enter> or click the mouse in the drawing area to create the copies.

TotalCAD will make radial copies in a counterclockwise direction, starting with the original object and ending at the angle set in step 4. The copy direction will be counterclockwise regardless of the direction setting in the Angle property sheet of the Drawing Setup dialog.

### Notes

You can also specify Angle by defining a point within the rubber band circle in the drawing area using the mouse. However, the number of sets, the +Rot rotation value, as well as values for rotating and resizing the copies, must be specified in the Edit Bar first, unless you want to accept the current values.

The Sets, Angle, and +Rot values are interdependent. Changing the number of sets will change the values in Angle and +Rot; and changing Angle will affect +Rot. These default ratios are designed to give you an elegant, symmetrical radial copy. Once you have entered values in these fields, however, the automatic interdependency is switched off, and you can enter values into fields without affecting values in the related fields.



## Fit Array Copy

**Menu:** Edit|Copy Entities|Fit Array

**Toolbar:** Copy

Use Fit Array Copy to arrange copies of selected objects into a two-dimensional array (rows and columns). The reference points of the outermost objects in the array will align to the edges of a rectangle that you define..

1. Select the object or objects that you want to copy.
2. Choose the Fit Array Copy tool.
3. Press <Tab> or click in the Edit Bar to activate it.



Optionally type values into XScale, YScale, and Rotation if you want to scale and rotate at the same time as you copy.



Type values in the XSize and YSize fields to specify the horizontal and vertical extents of the rectangle into which all the copies will fit.



Type values into the Rows and Columns fields to specify the number of rows and columns in the array.

4. Press <Enter> to create the copies.



**Note**

You can also specify the XSize and YSize values by defining a point relative to the selection in the drawing area using the mouse or Coordinate Fields. However, the number of rows and columns, as well as values for rotating and resizing the copies, must be specified in the Edit Bar first, unless you want to accept the current values.





## Mirror Copy

**Menu:** Edit|Copy Entities|Mirror **Toolbar:** Copy

Use Mirror Copy to create a mirror image of an object on the opposite side of a line that you specify.

1. Select the object or objects that you want to copy. If the Edit Bar is not visible, display it using View| Toolbars.
2. Choose the Mirror Copy tool.
3. Click in the drawing area or use the Coordinate Fields to define a point in the drawing area through which a mirror construction line will pass.
4. Enter the angle of the mirror construction line in the Angle field, and press <Enter>. Alternatively, you can use the Coordinate Fields or click in the drawing area to define a second point through which the mirror construction line will pass.

TotalCAD will draw a mirror image of the selected objects at the same distance from the line on the opposite side.



## Vector Copy

**Menu:** Edit|Copy Entities|Vector

**Toolbar:** Copy

Use Vector Copy to place a copy of an object at a specified angle and distance from the reference point of the original.

1. Select the object or objects that you want to copy.
2. Choose the Vector Copy tool.
3. Use the Coordinate Fields or click in the drawing area to define two points of a virtual line to specify the angle and distance of the copy from the original. This line appears in the drawing area as an arrow.

Alternatively, you can define enter the angle and distance of the vector directly in the Edit Bar and press <Enter>.


# Edit Links


**Menu:** Edit|Links


Edit Links displays a dialog in which you can make changes to any linked OLE objects in the current drawing. OLE objects are created in other Windows applications and inserted into your TotalCAD drawing with the Insert Object command. You can edit an OLE object within TotalCAD using the Windows application in which the object was created (the *source* application). For example, you can insert a Windows Paintbrush object into your drawing and edit it within TotalCAD using the Windows Paintbrush tools.


A *linked* object is one that retains its link to the file on which it is based. Changes to the OLE object affect the file, and changes to the file affect the object. These changes take place when the OLE object is *updated*. You can set each linked object so that the update happens automatically, or so that the update must be done manually.


The Links list box contains all the links in the active drawing. The changes that you make in the Edit Links dialog affect any links that are highlighted in the list. You can highlight multiple links by holding down the <Shift> or <Ctrl> key and clicking in the list. The Links column in the list gives you the name and path of the file to which the OLE object is linked. The Type column tells you the name of the source application. The Update column tells you whether the link is set to be updated automatically or manually.

 **Update Option.** These option buttons enable you set whether the selected links are updated automatically or manually.

 **Update Now.** This button updates the current link immediately. This is how you update a link if it is set to be manually updated.

 **Open Source.** This button opens the source file in the application that created it. This enables you to compare the linked object to the source file, and it allows you to edit the file in the source application.

 **Change Source.** This button opens a dialog that lets you choose a new source for the linked OLE object. Once the OLE object is linked to the new file, its contents will change to match the new file.

 **Break Link.** This button terminates the link between the OLE object and its source file. Once the link is broken, the OLE object is treated as an *embedded* object. It is still an OLE object, but changes to it no longer affect the source file, and changes to the source file no longer can affect the object. Breaking the link also affects the size (in bytes) of your drawing, since the embedded object contains its own copy of the data in the object, while the linked object contains only a reference to the object.

 **See Also**

Edit Object, Embedded and Linked OLE Objects, Insert Object

## Edit Object

**Menu:** Edit|Objects

Edit Object enables you to edit a selected OLE object in the object's source application.

1. Select an OLE object in your drawing. This object may appear as a document or a bitmap, or it may appear as an icon, depending on the options you chose when you inserted the OLE object.
2. Choose Edit|Objects from the menu, or simply double-click on the object.

If open editing is available, you will be given two choices in a submenu under Edit|Objects. Choose Edit|Objects|Edit for in-place editing. Choose Edit|Objects|Open for open editing in the source application.

 **Note**

 **See Also**

Other names can appear in place of Object on the Edit menu, depending on the OLE source application. If you select a Microsoft Word object, for example, the name on the menu will change to Document Object.

Edit Object, Embedded and Linked OLE Objects, Insert Object

## **View Menu and Tools**

### [Redraw](#)

Manually refresh a drawing that has changed.

### [Zoom Commands](#)

Access tools and commands for moving your view closer to or further away from an area of the drawing.

### [Pan Commands](#)

Pick particular points in the drawing and move them to specific locations on the screen.

### [Previous View](#)

Return to the previous view of the drawing.

### [View Grid](#)

Display or hide the grid.

### [View Aperture](#)

Change cursor to a circle that represents the snap aperture.

### [View Crosshairs](#)

Display crosshairs in place of mouse cursor.

### [View Rulers](#)

Display or hide the rulers.

### [View Status Bar](#)

Display or hide the Status Bar.

### [View Edit Bar](#)

Display or hide the Edit Bar.

### [View Symbol Library](#)

Display or hide the Symbol Library palette.

### [View Toolbars](#)

Display the Desktop property sheet in which you can display or hide toolbars.

### [View Selection Info](#)

Display or hide the information about the currently selected object.





## Redraw

**Menu:** View|Redraw    **Hotkey:** <F5>    **Toolbar:** View

In most cases TotalCAD will automatically redraw your screen whenever the drawing changes. Occasionally, however, you may find that the screen is not completely “cleaned up” after a drawing or editing operation. If this happens, you can manually redraw the screen using the Redraw command.

## Zoom Commands



**Menu:** View|Zoom      **Toolbar:** View

The Zoom commands and tools let you “zoom” into your drawing to view detail areas close up, or pull back to view larger areas of your drawing. The following table summarizes the Zoom commands and tools:

### [Zoom In](#)

Move one step into the drawing.

### [Zoom Out](#)

Move one step further from the drawing.

### [Zoom Window](#)

Activate the Zoom Window tool, with which you can select an area to view.

### [View Extents](#)

Zoom to a view that contains all of the objects in the drawing.

### [Full View](#)

Return to “starting” view of the drawing.

### [View Printed Size](#)

View the drawing at the size it will be when printed.



## Zoom In

**Menu:** View|Zoom|Zoom In      **Hotkey:** <Gray +>      **Toolbar:** View

The Zoom In command magnifies the current view of the screen. The scale of magnification is determined by the *zoom factor*, which by default is equal to two. You can reset the zoom factor in the [General property sheet](#) of the Tools|Program Setup dialog.

**Note**

**See Also**

If you zoom in using the menu command, you will zoom in on an area centered on the current view. If you zoom in using the <Gray +> key, you will zoom in to an area defined so that the mouse cursor is kept in the same relative position within the view rectangle.

## Program Setup Dialog



## Zoom Out

**Menu:** View|Zoom|Zoom Out    **Hotkey:** <Gray ->    **Toolbar:** View

The Zoom Out command backs away from the area where the mouse cursor is currently located. The amount that the view pulls back in each step is determined by the *zoom factor*, which by default is equal to two. You can reset the zoom factor in the [General property sheet](#) of the Tools|Program Options dialog.

**Note**

**See Also**

If you zoom out using the menu command, you will zoom out to an area centered on the current view. If you zoom out using the <Gray +> key, you will zoom out to an area defined so that the mouse cursor is kept in the same relative position within the view rectangle.

## Program Setup Dialog





## Zoom Window

**Menu:** View|Zoom|Zoom Window

**Hotkey:** <Ctrl>+<Home>

**Toolbar:** View

The Zoom Window tool enables you select a rectangle area of the screen and zoom in to a view of the that area. To use this tool:

1. Activate the Zoom Window tool.
2. Click to specify one corner of the zoom window.
3. Move the mouse cursor to the point where you want to place the diagonally opposite corner of the zoom window, so that the zoom window encompass the area that you want to view.
4. Click to specify the opposite corner of the window. TotalCAD will immediately zoom in on the area that the zoom window encloses.

You can also drag the mouse to create the zoom window, in which case your view will zoom in immediately after you release the mouse button.

**Note**

Because the dimensions of the zoom rectangle will probably not be exactly proportional to the screen, TotalCAD chooses the closest view based on the zoom rectangle you define.



## **View Extents**

**Menu:** View|Zoom|Extents      **Hotkey:** <Ctrl>+<Backspace>      **Toolbar:** View

The View Extents command zooms to a view of the smallest area of your drawing that completely contains all of the objects in your drawing. This is the same as the area enclosed by the selection rectangle if you select all the objects in the drawing.



## Full View

**Menu:** View|Zoom|Full View    **Hotkey:** <Shift>+<Backspace>    **Toolbar:** View

The Full View command returns you to the default view at which you began your drawing.



## View Printed Size

**Menu:** View|Zoom|Printed Size **Toolbar:** View

The View Printed Size command takes you to a view of the drawing at approximately the size it will be when printed. In this view one inch of screen space is approximately equal to one inch on paper.

 **Note**

The exact correspondence of screen space to paper space will vary, depending on your graphics display and screen resolution.

## **Pan Commands**

The Pan commands and tools move your view across the drawing area, in any direction.

### [Pan to Point](#)

Choose a point and scroll the drawing so that the point is at the center of the screen.

### [Vector Pan](#)

Choose a point and a destination and scroll the drawing so that the point moves to the destination.

## **Pan to Point**

**Menu:** View|Pan to Point

**Hotkey:** <Ctrl>+<End>

Use the Pan to Point feature to choose a point on the screen and scroll the drawing so that the point you chose moves to the center of the screen. This feature is available from the menu as a tool and from the keyboard as a command. To use the menu version of the feature:

1. Choose View|Pan to Point to activate the Pan to Point tool. The cursor will change to an arrow pointing to the center of a small crosshair icon.
2. Click on any point in the drawing. This point will move to the center of the screen.

To use the keyboard Pan to Point command, place the cursor at a point on the screen and press <Ctrl>+<End>. The point at the location of the cursor will scroll to the center of the screen.



## Vector Pan

**Menu:** View|Vector Pan

Use the Vector Pan tool to choose a point and a destination, and scroll the drawing so that the point moves to the destination. To use Vector Pan:

1. Activate the Vector Pan tool.
2. Click at any location in the drawing area. This specifies the location of the point that you would like to move (the *base point*).
3. Move the cursor to the destination point and click. The drawing will scroll so that the base point moves to this destination.



## Previous View

**Menu:** View|Previous View      **Toolbar:** View

The Previous View command returns you to the previous view of the drawing.



**Tip**

This command is particularly useful if you want to switch back and forth between two views.

## View Grid

**Menu:** View|Grid

The View Grid command is a toggle that displays and hides the grid. The grid is a pattern of dots, crosses, or lines that divides your drawing space geometrically into Cartesian or polar coordinate locations. You can define points at positions on the grid using the Snap to Grid snap mode. You can set the spacing, type, and display parameters of the grid in the Tools|Grid property sheet.

 **Note**

 **See Also**

Making the grid visible or invisible does not affect whether the grid is active. The grid can be visible and inactive, and it can be invisible and active. To control whether the grid is active, use the Snap to Grid command from the Modes|Snaps menu, the Snaps toolbar, or the Local menu.

[Coordinate Systems](#), [Grid Property Sheet](#), [Snap to Grid](#)

## View Aperture

**Menu:** View|Aperture

The View Aperture command is a toggle that displays the snap aperture. The snap aperture is the circular area in which an entity needs to be located in order to snap to a point located on the entity, or in which a grid point or graphic point must be located in order to snap to that point.

You can have View Aperture and View Crosshairs turned on simultaneously.

 **Tip**

 **See Also**

You can set the radius of the snap aperture in the Tools|Program Setup dialog.



[Snap Aperture](#), [View Crosshairs](#)

## View Crosshairs

**Menu:** View|Crosshairs

The View Crosshairs command is a toggle that changes the cursor to crosshairs and back to a normal cursor. The crosshair extension lines let you see more clearly how the current location of the cursor corresponds to positions on the ruler and on graphic objects.

 **See Also**

Program Setup Dialog, View Aperture

## View Rulers

**Menu:** View|Rulers

The View Rulers command is a toggle that displays and hides the Rulers. Rulers are displayed along the top and left edges of the screen and show the measurements of your drawing in Page or World units. You can change between Page and World units by clicking the Units button located near the upper-left corner of the drawing area at the intersection of the horizontal and vertical rulers.

You can change the units and scale of the drawing in the [Units and Scale property sheet](#) of the Tools|Drawing Setup dialog. The ruler will change to reflect the current scale and display.

 **See Also**

Drawing Setup Dialog

## View Status Bar

**Menu:** View|Status Bar

The View Status Bar command is a toggle that displays and hides the Status Bar. The Status Bar is displayed along the bottom of your screen. It contains a brief status message, the Coordinate Fields, and a zoom percentage box on its right side, which displays the [zoom percentage](#) of your current view.

## View Edit Bar

**Menu:** View|Edit Bar

The View Edit Bar command is a toggle that displays and hides the Edit Bar. The Edit Bar is a control that enables you to create and edit entities by entering distances, angles, and other data directly into the Edit Bar fields. You can drag the Edit Bar and place it anywhere in the tool area under the menubar. You can also place the Edit Bar over the drawing space, where it will float over the drawing as a palette.

 **Tip**

 **Note**

 **See Also**

Although you can draw in TotalCAD using only the mouse, many drawing and editing tools are more accurate if you work in the Edit Bar, and some tools have limited use without the Edit Bar. You will probably want to keep the Edit Bar on screen at all times.



The fields in the Edit Bar change dynamically, depending on the currently active drawing or editing tool. The number of fields can vary quite a lot, so it's a good idea to place the Edit Bar toward the left edge of the screen so that all the fields will always be visible.

[Drawing with the Edit Bar](#)

## View Symbol Library

**Menu:** View|Symbol Library

The View Symbol Library command is a toggle that displays and hides the Symbol Library palette. The Symbol Library palette shows the symbols in the current symbol library.

Symbols are actually fully independent TotalCAD drawings. A symbol library is simply a Windows folder (directory) that contains any number of TotalCAD files. The Symbol palette displays the contents of a symbol library, as large icons, small icons, or file names. If you click on any of the symbols displayed in the palette, a description of the symbol will appear in the description area at the bottom of the palette, as shown in the preceding illustration.

[Placing a Symbol in a Drawing](#)

[Turning a Selection into a Symbol](#)

[Changing to Another Open Symbol Library](#)

[Opening a New Symbol Library](#)

[Changing Your View of a Symbol Library](#)

 **Tip**

 **See Also**

To access much of the power of the Symbol palette, you need to go through its local menu, which you display by right-clicking in the Symbol palette title bar.

Symbol Libraries Property Sheet

## Placing a Symbol in a Drawing

To place a symbol in your drawing, simply drag it out of the Symbol palette and drop it in your drawing. The symbol will be placed in the drawing at the point where you release it. The symbol will remain selected after you place it, so that it can be repositioned and scaled.

The symbols that IMSI provides with TotalCAD are groups. This means that they have names, which makes them easier to keep track of, and it means that they can be selected and manipulated as a single unit.

 **Warning**

Don't expect all symbols to be this well-behaved. A symbol is really just a TotalCAD drawing, and it won't be a group unless the person who created the symbol saved it as a group.

## Turning a Selection into a Symbol

Any TotalCAD drawing can be a symbol. To create your own symbol library, all you really need to do is save all the drawings that comprise the library into a single Windows folder. However, another equally good way to save symbols is to drag selections from drawings into the Symbol palette. This gives you the additional option of saving only part of an existing drawing as a symbol.

1. Select the parts of your drawing that you want to turn into a symbol. (If you want to drag your entire drawing into the Symbol palette, choose Edit|Select All.)
2. Before dragging it into the Symbol palette, it's a good idea to group the selection by choosing Format|Create Group. (This isn't strictly necessary, but it will make the symbol easier to work with later.)
3. Hold the mouse over the selection's reference point (the dot in the middle of the selection). You will know the mouse is in the right position when the cursor turns into a four-directional pointer.
4. Hold the mouse button down and drag the selection into the Symbol palette. Release the mouse button to drop the selection. The Save Symbol As dialog appears.
5. Save the symbol as a file, just as you would in the regular Save As dialog. You have the option of navigating to another folder, which will have the effect of placing the symbol into a different symbol library. If you put it in a folder that isn't already a symbol library, TotalCAD will open a new symbol library in your drawing.

 **See Also**



Save As Dialog

## **Changing to Another Open Symbol Library**

You can have multiple symbol libraries open in your drawing. (You open and name symbol libraries in the Tools|Symbol Library property sheet.) To view the open libraries from the Symbol palette, right-click in the palette's title bar to display its local menu. At the bottom of the local menu all of the open symbol libraries are displayed. To switch to another library, click on its name in the local menu. The contents of that library will then be displayed in the Symbol palette.

## Opening a New Symbol Library

To open a new symbol library, right-click in the Symbol palette title bar and choose Open. This displays the Symbol Libraries property sheet, in which you can name and open new libraries.

 **See Also**

Symbol Libraries Property Sheet

## **Changing Your View of a Symbol Library**

The Symbol palette can display its symbols as icons, small icons, or as names. To switch between these three options, first right-click in the Symbol palette title bar to display its local menu, then choose Names, Small Icons, or Large Icons. A check mark will appear next to the currently selected option.

## View Toolbars

**Menu:** View|Toolbars

The View Toolbars command displays the Desktop property sheet, which enables you to hide and view TotalCAD toolbars, as well as other interface controls.

 **See also**

Desktop Property Sheet

## View Selection Info

**Menu:** View|Selection Info

The View Selection Info command displays and hides the Selection Info palette. This palette contains information about the currently selected object, such as its entity type, dimensions, and location in the drawing.

The Selection Info palette contains two windows. The upper window shows the type and structure of the objects selected. If an object is a group containing other objects, this relationship will be displayed in outline form. A plus sign next to an item indicates that it contains other items that are not displayed. To open the outline, and see what objects are contained, click on the plus sign; click again to close the outline to see a higher level of the hierarchy.

The lower window displays precise data about whatever object is currently selected in the upper window. It shows the object's graphic type, its coordinate location in the drawing space, and its dimensions in current World units.

Pressing the Copy button copies this data to the Windows clipboard so that you can paste it into a word processor or text editor, such as Windows notepad. The text that you copy contains special markers so that programmers, or very adventurous users, can "parse" the data in their own programs.





Experienced Web users may notice that the markers are similar to those used in HTML.

## Insert Menu and Tools

The Insert menu provides access to tools that let you draw entities and commands that enable you to insert pictures and OLE objects into your drawing.

### [Construction Tools](#)

Create temporary construction lines.

### [Point](#)

Draw single-point marker entities in various shapes.

### [Line](#)

Draw lines, rectangles, polygons, and lines tangent to circles and arcs.

### [Double Line Tools](#)

Draw double line segments, rectangles, and polygons.

### [Circle/Ellipse](#)

Draw circles and ellipses.

### [Arc Tools](#)

Draw circular and elliptical arcs.

### [Curve Tools](#)

Draw Bezier curves, spline curves, and freehand sketches.

### [Text Tool](#)

Add single- or multiple-line text to the drawing.

### [Dimension Tools](#)

Draw dimension entities.

### [Insert Picture](#)

Insert images (bitmaps or metafiles) into the drawing.

### [Insert Object](#)

Insert OLE objects into the drawing.

## Construction Tools

**Menu:** Insert|Construction

The construction tools enable you to place temporary construction lines on your drawing. Construction lines are entities used for reference: You can define points in relation to constructions and use them to segment your drawing space.

[Horizontal Construction Line](#)

Draw a horizontal construction line.

[Vertical Construction Line](#)

Draw a vertical construction line.

 **Notes**

Construction lines exist on their own layer of the drawing, called “\$CONSTRUCTION”. You can clear this layer with the Clear|Constructions command. You can clear an individual construction with the Edit|Clear|Construction tool.

The properties of constructions cannot be accessed or changed.

## **Horizontal Construction Line**

**Menu:** Insert|Construction|Horizontal Line

To place a horizontal construction line in your drawing:

1. Activate the Horizontal Construction Line tool.
2. Define a point that you want the line to intersect. TotalCAD draws a horizontal construction line that intersects the point you have defined.

## **Vertical Construction Line**

**Menu:** Insert|Construction|Vertical Line

To place a vertical construction line in your drawing:

1. Activate the Vertical Construction Line tool.
2. Define a point that you want the line to intersect. TotalCAD draws a vertical construction line that intersects the point you have defined.

## Point

**Menu:** Insert|Point|Dot, Star, Square, Cross, or Circle    **Toolbars:** Insert

You can create five different types of marker points: dot, star, square, cross, or circle. Each type of point is created in the same way. Four of these points are essentially similar, except for their shape. The Dot point is different: It stays the same size no matter how close you zoom in or how far you zoom out. This is useful if you want to use it as a point of reference while working in different views of the drawing.

To place a point:

1. Activate the Point tools and select the type of point you want to create.
2. Define a point.

 **Note**

 **Tip**

Points cannot be scaled or rotated.



If you create a point of a particular shape and you subsequently want to change the shape, activate the Select tool, double-clicking on the point, then change its Point Type in the Point property sheet.

# Line

**Menu:** Insert\Line      **Toolbar:** Insert

The Line tools enable you to draw a single line segment, multiple connected segments (multilines), polygons, rectangles, perpendicular lines, parallel lines, and lines tangent to arcs.

## Single Line

Draw a single line segment.

## Multiline

Draw a series of connected line segments.

## Polygon

Draw a regular polygon with an arbitrary number of sides.

## Rectangle

Draw an orthogonal rectangle.

## Perpendicular Line

Draw a line perpendicular to an existing line from any point.

## Parallel Line

Draw a line parallel to an existing line, at a specific distance from the line.

## Tan Arc Pt

Draw a line tangent to an arc, touching the arc at the line's midpoint.

## Tan to Arc

Draw a line tangent to an arc, with the line's second endpoint touching the arc.

## Tan from Arc

Draw a line tangent to an arc, with the line's first endpoint touching the arc.

## Line 2 Arcs

Draw a line tangent to two arcs, with each endpoint of the line touching one of the arcs.



## Single Line

**Menu:** Insert|Line|Single

**Toolbars:** Insert

Use the Single Line tool to draw a line segment between two defined points:

1. Activate the Single Line tool.
2. Define the line segment's starting point.
3. Do one of the following:



Define the line segment's ending point.





Set the length and angle of the line segment in the Edit Bar, then press <Enter>.



## Multiline

**Menu:** Insert|Line|Multiline      **Toolbars:** Insert

The Multiline tool enables you to draw a series of connected line segments that comprise one entity. You can create a multiline by defining a series of points. Alternatively, you can set a starting point, then define each line segment in sequence by entering its length and angle in the Edit Bar, pressing <Enter> to complete each segment of the multiline, and pressing <Ctrl>+<F> to finish the multiline:

1. Activate the Multiline tool.
2. Define the first point of the multiline entity.
3. Do one of the following:
  -  If necessary, reset the Snap mode, then define a second point: TotalCAD draws a line segment between the two points.
  -  Specify the length and angle of the segment in the Edit Bar and press <Enter>.
4. Repeat step 3, adding connected segments to the multiline, until the multiline is finished.
5. To define the last point in the series, double-click, choose Finish from the Local Menu, or press <Ctrl>+<F> if you are working in the Edit Bar.

 **Tips**

 **Note**

Instead of choosing Finish, you can have TotalCAD close the multiline for you by choosing Close from the Local Menu. This draws a line segment from the last point of the multiline to the first point.

To undo the most recently drawn segment of the multiline, open the Local Menu and choose One Step Back.

Once a multiline is closed, TotalCAD treats it as a polygon, which means that it will be filled with the color and pattern of the current brush. You can control brush formatting through the Format|Properties dialog.



## Polygon

**Menu:** Insert|Line|Polygon      **Toolbars:** Insert

Use the Polygon tool to draw a regular polygon by defining points for the polygon's center and for one of its vertices:

1. Activate the Polygon tool.
2. Define a point for the center of the polygon.
3. Enter the number of sides for the new polygon in the Sides field on the Edit Bar.
4. Do one of the following:



Define a second point for one of the polygon's vertices.



Type values for the polygon's radius, and degree of rotation in the Edit Bar. The radius is the distance from the centerpoint to one of the vertices, and the degree of rotation is the angle of a line from the centerpoint to one vertex.



**Note**

You could create a regular polygon by defining the distance between the center and the midpoint of a line segment. TotalCAD 1.0 does not currently support this capability.





## Rectangle

**Menu:** Insert|Line|Rectangle    **Toolbars:** Insert

The Rectangle tool enables you to create a rectangle by defining two diagonally opposite corners. Rectangles created in this way are oriented orthogonally, although they can subsequently be rotated using the editing tools:

1. Activate the Rectangle tool.
2. Define a point for one corner of the rectangle.
3. Do one of the following:



Define a point for the diagonally opposite corner.



Specify the A (horizontal) and B (vertical) size of the rectangle in the Edit Bar, then press <Enter>.



**Tip**

If you use the Edit Bar method, the A size will be drawn upward from the starting point, and the B size will be drawn to the right of the starting point. To draw downward or to the left, use negative values.



## Perpendicular Line

**Menu:** Insert|Line|Perpendicular Line    **Toolbars:** Insert

Use the Perpendicular Line tool when you want to draw a line perpendicular to an existing line. The new line extends from a defined point to the existing line:

1. Activate the Perpendicular Line tool.
2. Click on an existing line segment to which you want to draw the perpendicular line.
3. Define the point from which you want the perpendicular line to originate.

If the point you define is beyond the end of the existing line segment, TotalCAD by default will draw the perpendicular line from this point to the point where the two lines would meet if the existing line segment were extended. To constrain the perpendicular line so that it must meet the existing line segment, right-click in the drawing area and choose Limit to Segment in the Local Menu.



## Parallel Line

**Menu:** Insert|Line|Parallel

**Toolbars:** Insert

Use the Parallel Line tool to draw a line parallel to an existing line.

1. Activate the Parallel Line tool.
2. Click on or near an existing line. Let's call this the *source line*. As you move the mouse cursor away from the source line, TotalCAD draws a line parallel to it.
3. Define a second point, which will be the second endpoint of the new line.

You can also select the source line, then specify the length of the parallel line and its distance from the source line in the Edit Bar. Press <Enter> or <Ctrl>+<F> to finish the parallel line.



Tip

To create a parallel line that is constrained to be the same length as the source line, right-click in the drawing area with the Parallel Line tool active, then switch on the Keep Length option in the Local Menu.



## Tan Arc Pt

**Menu:** Insert|Line|Tan Arc Pt    **Toolbars:** Insert

The Tan Arc Pt tool enables you to draw a line tangent to an arc or circle with the midpoint of the line centered at the point of tangency:

1. Activate the Tan Arc Pt tool.
2. Click on the arc or circle on which you want to draw the line. A line appears tangent to the arc at a point on the arc near the mouse click.
3. Move the mouse to adjust the length of the line. The line length changes in both directions so that the midpoint of the line remains at the point where the line and arc meet. Click to set the line length.
4. A dotted line appears, from the center of the arc or circle to the mouse position, perpendicular to the tangent line. Move the mouse to change the angle of the dotted line and the position where the tangent line touches the arc, then click to set the angle of the tangent line.

Alternatively, you can click on the circle or arc, then specify the length and angle of the tangent line in the Edit Bar, then press <Enter>.



## Tan to Arc

**Menu:** Insert|Line|Tan to Arc    **Toolbars:** Insert

Use the Tan To Arc tool to draw a line tangent to an arc or circle from a defined point.

1. Activate the Tan to Arc tool.
2. Define the point from which you want to draw the tangent line.
3. Click on the side of the arc or circle to which you want to draw the tangent line.

 **Tip**

To create a tangent line with a fixed length, define the first point of the line, then enter a length in the Edit Bar and check the lockdown box. Return to the drawing area and click on the circle or arc near the point of tangency. If the length you define is too long or too short this won't work—but you can return to the Edit Bar to adjust the length.





## Tan from Arc

**Menu:** Insert|Line|Tan from Arc **Toolbars:** Insert

Use the Tan from Arc tool to draw a line tangent from an arc or circle.

1. Activate the Tan from Arc tool.
2. Click on the side of the arc or circle from which you want to draw the tangent line.
3. Do one of the following:



Define the point to which you want to draw the tangent line.



Specify the length of the line in the Edit Bar, then press <Enter>.



## Line 2 Arcs

**Menu:** Insert|Line|Line 2 Arcs    **Toolbars:** Insert

Use the Line 2 Arcs tool to draw a line tangent to two arcs or circles:

1. Activate the Draw|Line|Line 2 Arcs tool.
2. Click on the side of the first arc or circle from which you want the line drawn tangent.
3. Click on a side of the second arc or circle.

 **Note**

The current Snap mode is irrelevant when you use this tool.

## Double Line Tools

**Menu:** Insert|Double Line

**Toolbars:** Insert

Use the double line tools to draw two parallel lines as a single entity. The Double Line tools are essentially similar to the Line tools, except that there are no Double Line tools for drawing lines tangent to arcs.

[Double Line Properties](#)

[Double Line Separation](#)

[Double Line Segment](#)

Draw a double line segment by defining two endpoints.

[Double Line Multiline](#)

Draw a series of connected double lines.

[Double Line Rectangle](#)

Draw a horizontal double line rectangle.

[Perpendicular Double Line](#)

Draw a double line perpendicular to an existing line.

## Double Line Properties

Double Lines have unique properties that you can set in the Double Line property sheet of the Edit| Properties dialog. You can access the default properties for double lines by right-clicking on any double line tool. You can access the properties of a specific double line entity by activating the Select tool and then double-clicking on the entity.

In the Double Line property sheet you can:



Set the distance between the two lines of the double line.



Determine whether the endpoints of the double line are open or closed. Each endpoint can be set individually.



Set the alignment of the double line. Alignment sets whether the nodes (reference points) of the double line are aligned along its “left” line, its “right” line, or along its center.

The alignment of the nodes determines how the double line aligns itself to the coordinate system, and to the grid and other objects when you use snap modes to position the double line.



**Tip**

To determine which is the left and which is the right side of a double line, imagine yourself standing on the starting point of the line, facing the endpoint. No matter which way you rotate the line, the right side remains on your right. This is similar to the terms “port” and “starboard” which define directions relative to the fore and aft of a ship. The critical thing is to know the direction in which the double line is drawn.

## Double Line Separation

The width of a double line is set only in its property sheet. If you stretch or scale a double line, its width always remains the same. If you want the width of the double line to change as you resize a selection that contains the double line, first select the double line and choose **Explode** from the **Format** menu. The first time you explode a double line, it will become a group of single line segments. (If the double line entity is itself contained within a group, you will need to explode it more than once.) Once the double line is exploded into two single lines, the distance between the two single lines will be scaled along with the selection that contains them.

 **See Also**

Double Line Property Sheet





## Double Line Segment

**Menu:** Insert|Double Line|Single

**Toolbars:** Insert

Use the Double Line Segment tool to draw a double line segment between two defined points.

1. Activate the Double Line Segment tool.
2. Define the double line segment's starting point.
3. Do one of the following:



Define the double line segment's ending point.



Specify the length and angle of the double line segment in the Edit Bar, then press <Enter>.



## Double Line Multiline

**Menu:** Insert|Double Line|Multiline

**Toolbars:** Insert

The Double Line Multiline tool enables you to draw a series of connected double line segments. You can create a multiline by defining a series of points. Alternatively, you can set a starting point, then define each double line segment in sequence by entering the length and angle of each segment in the Edit Bar, pressing <Enter> to complete each segment of the multiline.

1. Activate the Double Line Multiline tool.
2. Define the first point of the multiline entity.
3. Do one of the following:



If necessary, reset the Snap mode, then define a second point: TotalCAD draws a double line segment between the two points.



Specify the length and angle of the double line segment in the Edit Bar and press <Enter>.

4. Repeat step 3, adding connected double line segments to the multiline, until the multiline is finished.
5. To define the last point in the series, right-click in the drawing area and choose Finish from the Local Menu, or press <Ctrl>+<F> if you are working in the Edit Bar.



**Tips**

You can have TotalCAD close the multiline for you by choosing Close from the Local Menu. This draws a double line segment from the last point of the multiline to the first point, resulting in a double line polygon.

If you want to cancel the previous line segment, open the Local Menu and choose One Step Back.



## Double Line Rectangle

**Menu:** Insert|Double Line|Rectangle    **Toolbars:** Insert

This tool enables you to create a double line rectangle by defining two diagonally opposite corners. Rectangles created in this way are orthogonal, although they can subsequently be rotated using editing tools.

1. Activate the Double Line Rectangle tool.
2. Define the first point.
3. Do one of the following:



Define a point for the diagonally opposite corner.



Specify the vertical and horizontal dimensions of the rectangle in the Edit Bar, then press <Enter>.



**Note**

If you use the Edit Bar method, the A size will be drawn upward from the starting point, and the B size will be drawn to the right of the starting point. You can draw downward and to the left by entering negative values.



## Perpendicular Double Line

**Menu:** Insert|Double Line|Perpendicular Line    **Toolbars:** Insert

Use the Perpendicular Double Line tool when you want to draw a double line perpendicular to an existing line. The double line extends from a defined point to the existing line.

1. Activate the Perpendicular Line tool.
2. Click on an existing line segment to which you want to draw the perpendicular double line. (This can be one of the sides of another double line.)
3. Define the point from which you want the perpendicular line to originate.



**Tip**

If the point you define is beyond the end of the existing line segment, TotalCAD will normally draw the double perpendicular line from this point to where it would meet the existing line segment if the existing line segment were extended. If you want to constrain the perpendicular line so that it will not be drawn beyond the ends of the existing line segment, right-click in the drawing area to open the Local Menu, then choose Limit to Segment.

## Circle/Ellipse

**Menu:** Insert|Circle/Ellipse      **Toolbar:** Insert

The Circle/Ellipse menu item opens a submenu containing tools for drawing circles and ellipses. You can also access these tools from the Ellipse toolbar, and on the Ellipse flyout of the Insert toolbar.

### Center and Point

Draw a circle by specifying its centerpoint and radius.

### Double Point

Draw a circle by defining two opposite points on its perimeter.

### Circle Tan to Arc

Draw a circle tangent to another circle or arc.

### Circle Tan to Line

Draw a circle tangent to a line.

### Triple Point

Draw a circle by defining three points on the circle's perimeter.

### Ellipse

Draw an ellipse by defining its bounding rectangle.

### Rotated Ellipse

Draw an ellipse rotated at any angle.





## Center and Point

**Menu:** Insert|Circle/Ellipse|Center and Point      **Toolbar:** Insert

Use the Center and Point tool to create a circle by defining its center and a point on its perimeter.

1. Activate the Center and Point tool.
2. Define a point for the center of the circle.
3. Do one of the following:



Define a point on the perimeter of the circle.



Enter the radius of the circle in the Radius field, or the circumference of the circle in the Circumf. field on the Edit Bar, then press <Enter>.



## Double Point

**Menu:** Insert|Circle/Ellipse|Double Point

**Toolbar:** Insert

The Double Point tool lets you draw a circle by defining two opposite points on the circle's perimeter:

1. Activate the Double Point tool.
2. Define a point that will be on the perimeter of the circle.
3. Do one of the following:



Define a second point, 180 degrees from the first point on the perimeter of the circle.





Specify the diameter (or circumference) of the circle and the angle of the line in the Edit Bar, then press <Enter>.



## Circle Tan to Arc

**Menu:** Insert|Circle/Ellipse|Tan to Arc    **Toolbar:** Insert

Use the Circle Tan to Arc tool to draw a circle tangent to an arc or to another circle.

1. Activate the Circle Tan to Arc tool.
2. Click on the existing arc or circle to which you want the circle to be tangent.
3. Do one of the following:
  -  Define the centerpoint of the circle.
  -  Enter the radius or circumference of the circle in the Edit Bar, then *lock down* the field in which you typed the value, then return to the drawing area by clicking there (or press <Ctrl>+<D>). Click once more to continue to the next step.
4. Move the mouse along the perimeter of the arc to position the circle on the arc. If you move the mouse along the inside of the arc, the circle will move to the inside of the arc; if you move the mouse along the outside of the arc, the circle will move to the outside. Once the circle is in place, click the mouse to fix its position.

 **Note**

After clicking on the circle or arc (step 2), you can set the radius or circumference of the circle you are drawing in the Edit Bar and press <Enter>, finishing the entity. Be aware that if you use this method, you will not be able to reposition the circle on the arc using this tool, so make sure that you place it where you want it with the first click.



## Circle Tan to Line

**Menu:** Insert|Circle/Ellipse|Tan to Line   **Toolbar:** Insert

Use the Circle Tan to Line tool to draw a circle tangent to an existing line.

1. Activate the Circle Tan to Line tool.
2. Click on the line to which you want the circle to be tangent.
3. Do one of the following:



Define the centerpoint of the circle.



Enter the radius or circumference of the circle in the Edit Bar, then *lock down* the field in which you typed the value, then return to the drawing area by clicking there (or press <Ctrl>+<D>). Click once more to continue to the next step.

4. Move the mouse along the line to position the circle, then click (using any snap mode) to fix the circle's position.



**Note**

After clicking on the line (step 2), you can set the radius or circumference of the circle you are drawing in the Edit Bar and press <Enter>, finishing the entity. Be aware that if you use this method, you will not be able to reposition the circle on the line using this tool, so make sure that you place it where you want it with the first click.



## Triple Point

**Menu:** Insert|Circle/Ellipse|Triple Point   **Toolbar:** Insert

Use the Triple Point tool to draw a circle that passes through any three points.

1. Activate the Triple Point tool.
2. Define the first, then the second, then the third point through which the perimeter of the circle will pass.



## Ellipse

**Menu:** Insert|Circle/Ellipse|Ellipse

**Toolbar:** Insert

The Ellipse tool lets you create an ellipse by defining a bounding rectangle for the ellipse. The axes of the ellipse will be orthogonal; however, you can rotate the ellipse after you have created it using the editing tools.

1. Activate the Ellipse tool.
2. Define the first point or corner of the bounding rectangle.
3. Do one of the following:



Define the diagonally opposed corner of the bounding rectangle.



Specify the lengths of the ellipse's major and minor axes in the Edit Bar, then press <Enter>.





## Rotated Ellipse

**Menu:** Insert|Circle/Ellipse|Rotated Ellipse

**Toolbar:** Insert

The Rotated Ellipse tool enables you to create an ellipse tilted at any angle.

1. Activate the Rotated Ellipse tool.
2. Define the centerpoint of the ellipse.
3. Define a point to specify an angle and distance from the centerpoint. This creates an axis from the defined point, through the centerpoint, to an opposite point the same distance from the centerpoint. You can view this axis on the screen as you adjust the position of the point using the mouse or Coordinate Fields.
4. Define a second point to specify the length of the opposite axis. This axis will of course be perpendicular to the first axis.

Alternatively, you can define the centerpoint of the ellipse, specify the length and angle of the major axis and the length of the minor axis in the Edit Bar, then press <Enter>.

## Arc Tools

**Menu:** Insert|Arc

**Toolbars:** Insert

Drawing arcs is very similar to drawing circles. Generally, the arc tools work precisely the same way as the circle tools, except that they add one extra step: defining the start and end angles of the arc.

[Circumference and Length of Arcs](#)

[Combining Edit Bar and Mouse When Drawing Arcs](#)

[Center and Point Arc](#)

Draw a circular arc by specifying its centerpoint and a point on its circumference.

[Double Point Arc](#)

Draw a circular arc by defining the endpoints of its diameter.

[Arc Tan to Arc](#)

Draw a circular arc tangent to a circle or arc.

[Arc Tan to Line](#)

Draw a circular arc tangent to a line.

[Triple Point 1-2-3](#)

Draw a circular arc by defining (1) its starting point, (2) a point on its perimeter, and (3) its ending point.

[Elliptical Arc](#)

Draw an elliptical arc by defining its bounding rectangle.

[Rotated Elliptical Arc](#)

Draw an elliptical arc rotated at any angle .

 **Note**

Arcs are always drawn in a counterclockwise direction.

## **Circumference and Length of Arcs**

When you are drawing arcs using the Edit Bar, you will need to distinguish between the Circumf. field and the Length field.

The Circumf. field holds that value that represents the circumference of the arc circle, that is, the length of the circle that the arc would form if it were closed. This field and the Radius field are mutually dependent: Changing one changes the other. The Length field is the actual length of the arc, taking into account its starting and ending angles.

## Combining Edit Bar and Mouse When Drawing Arcs

Arcs are relatively complex entities, and it's likely that you'll want to use various combinations of the Edit Bar and the mouse to create them.



Keep in mind that if you specify values in the Edit Bar, you must lock the fields down before returning to the drawing area to finish the entity. If you don't, the values that you typed in the Edit Bar will change when you re-enter the drawing area.



## Center and Point Arc

**Menu:** Insert|Arc|Center and Point

**Toolbars:** Insert

Use the Center and Point tool to create an arc by defining its center, a point on its circumference, and its starting and ending angles.

1. Activate the Center and Point tool.
2. Define the centerpoint of the arc.
3. Define a point at a distance from the centerpoint to specify the radius of the arc. TotalCAD draws a circle with the specified radius and centerpoint. A dotted line appears, extending from the centerpoint of the circle to the cursor.
4. Move the cursor to change the the dotted line to the angle at which you want the arc to start, then click to set the arc's starting angle. Move the cursor counterclockwise to draw the arc, then click the mouse button to define the ending angle.

Alternatively, you can define a point for the center of the circle, enter the radius (or circumference) of the arc and its starting and ending angles (or length) in the Edit Bar, and then press <Enter>.

You can also combine the two approaches. For example, you could define the center and radius of the arc using the mouse, then specify the starting and ending angles in the Edit Bar.



## Double Point Arc

**Menu:** Insert|Arc|Double Point   **Toolbars:** Insert

The Double Point tool lets you draw an arc by defining the endpoints of its diameter.

1. Activate the Double Point tool.
2. Define the starting point of a line that will pass through the center of the arc.
3. Define the endpoint of this line. TotalCAD draws a circle that passes through the two defined points. A dotted line appears, extending from the centerpoint of the circle to the cursor.
4. Move the cursor to change the the dotted line to the angle at which you want the arc to start, then click to set the arc's starting angle. Move the cursor counterclockwise to draw the arc, then click the mouse button to define the ending angle.

Alternatively, you can define the first point of the arc's diameter, then specify all the quantities that define the arc in the Edit Bar.

You can also combine the two approaches. For example, you could define the center and radius of the arc using the mouse, then specify the starting and ending angles in the Edit Bar.





## Arc Tan to Arc

**Menu:** Insert|Arc|Tan to Arc      **Toolbars:** Insert

Use the Arc Tan to Arc tool to draw an arc tangent to an existing arc or circle.

1. Activate the Arc Tan to Arc tool.
2. Click on the existing arc or circle to which you want the arc to be tangent.
3. Do one of the following:



Define the centerpoint of the arc you are drawing.



Enter the radius or circumference of the circle in the Edit Bar, then *lock down* the field in which you typed the value, then return to the drawing area by clicking there (or press <Ctrl>+<D>). Click to continue to the next step.

4. Move the mouse along the perimeter of the arc to position the circle on the arc. If you move the mouse along the inside of the arc, the circle will move to the inside of the arc; if you move the mouse along the outside of the arc, the circle will move to the outside. Once the circle is in place, click the mouse to fix its position. A dotted line appears, extending from the centerpoint of the circle to the cursor.
5. Move the cursor to change the dotted line to the angle at which you want the arc to start, then click to set the arc's starting angle.
6. Move the cursor counterclockwise to draw the arc, then click the mouse button to define the ending angle.



**Note**



After clicking on the circle or arc (step 2), you can set the radius (or circumference) of the arc, as well as its starting and ending angles ( or length) in the Edit Bar,. then press <Enter> to finish the entity. Be aware that if you use this method, you will not be able to reposition the circle on the arc using this tool, so make sure that you place it where you want it with the first click.



## Arc Tan to Line

**Menu:** Insert|Arc|Tan to Line    **Toolbars:** Insert

Use the Arc Tan to Line tool to draw an arc tangent to an existing line.

1. Activate the Arc Tan to Line tool.
2. Click on the line to which you want the arc to be tangent.
3. Do one of the following:
  -  Define the centerpoint of the arc you are drawing.
  -  Enter the radius or circumference of the circle in the Edit Bar, then *lock down* the field in which you typed the value, then return to the drawing area by clicking there (or press <Ctrl>+<D>). Click to continue to the next step.
4. Move the mouse to either side of the line and define a point to position the arc on the line. A dotted line appears, extending from the centerpoint of the circle to the cursor.
5. Move the cursor to change the the dotted line to the angle at which you want the arc to start, then click to set the arc's starting angle.
6. Move the cursor counterclockwise to draw the arc, then click the mouse button to define the ending angle.

 **Note**

After clicking on the line (step 2), you can set the radius (or circumference) of the arc, as well as its starting and ending angles ( or length) in the Edit Bar, then press <Enter> to finish the entity. Be aware that if you use this method, you will not be able to reposition the circle on the line using this tool, so make sure that you place it where you want it with the first click.



## Triple Point 1-2-3

**Menu:** Insert|Arc|Triple Point 1-2-3      **Toolbars:** Insert

The Triple Point 1-2-3 tool lets you draw an arc by defining three points: a starting point, a middle point, and an ending point. The arc will touch all three of the points.

1. Activate the Triple Point 1-2-3 tool.
2. Define the starting point of the arc.
3. Define a point through which the arc will pass.
4. Define the endpoint of the arc.



## Elliptical Arc

**Menu:** Insert|Arc|Elliptical

**Toolbar:** Insert

The Elliptical Arc tool lets you create an elliptical arc by defining its bounding rectangle. The lengths of the minor and major axes of the arc are determined by the lengths of the sides of the bounding rectangle. The axes of the arc will be orthogonal; however, you can rotate the arc after you have created it using the editing tools:

1. Activate the Ellipse tool.
2. Define a point for the first corner of the bounding rectangle.
3. Define a point for the diagonally opposite corner of the bounding rectangle. TotalCAD draws an ellipse bounded by the rectangle that you have defined. A dotted line appears, extending from the centerpoint of the ellipse to the cursor.
4. Move the cursor to change the dotted line to the angle at which you want the arc to start, then click to set the arc's starting angle.
5. Move the cursor counterclockwise to draw the arc, then click the mouse button to define the ending angle.

Alternatively, you can define the starting point of the elliptical arc, then specify its major and minor axes and starting and ending angles in the Edit Bar. You can also combine use of the mouse and Edit Bar.



## Rotated Elliptical Arc

**Menu:** Insert|Arc|Rotated Elliptical

**Toolbar:** Insert

The Rotated Elliptical Arc tool enables you to create an elliptical arc tilted at any angle:

1. Activate the Rotated Elliptical Arc tool.
2. Define the centerpoint of the arc.
3. Define a point to specify an angle and distance from the centerpoint. This creates an axis from the defined point, through the centerpoint, to an opposite point the same distance from the centerpoint. This is the major axis of the elliptical arc.
4. Define a third point to specify the length of the minor axis (perpendicular to the major axis). TotalCAD draws an ellipse with the specified major and minor axes. A dotted line appears, extending from the centerpoint of the ellipse to the cursor.
5. Move the cursor to change the dotted line to the angle at which you want the arc to start, then click to set the arc's starting angle.
6. Move the cursor counterclockwise to draw the arc, then click the mouse button to define the ending angle.

You can also define the centerpoint of the ellipse, then specify the length and angle of the major axis and the length of the minor axis, as well as the starting and ending angles of the arc, in the Edit Bar.

## Curve Tools

**Menu:** Insert|Curve    **Toolbar:** Insert

[Curve Properties](#)

[Bezier Curve](#)

Connects two endpoints; gravitates toward control points along its path.

[Spline Curve](#)

Continuous curve that connects a series of points.

[Sketch Curve](#)

Freehand multiline drawn with mouse.



## Curve Properties

Curve's have special properties that can be set in the Curve property sheet of the Properties dialog. This property sheet lets you show or hide the frame of a Bezier or Spline curve, transform a Bezier curve into a spline curve, or vice versa. You can access the default properties of curves by property sheet by double-clicking on a curve with the Select tool, or by selecting the curve and then choosing Format|Properties.

 **Tip**

 **See Also**

You can close a Bezier or Spline curve by choosing Close from the Local Menu instead of Finish. This draws a curved line between the two endpoints. You can also close curves or break open closed curves using Node Edit mode, employing the same technique that you use on multilines and polygons.

[Breaking Polygons](#), [Closing Multilines](#), [Curve Property Sheet](#)



## Bezier Curve

**Menu:** Insert|Curve|Bezier

**Toolbar:** Insert

To create a Bezier curve, define points along the intended path of the curve. The curve will span the two endpoints and be pulled toward the intermediate points, which act as control points along its path.

1. Activate the Bezier Curve tool.
2. Define the first endpoint of the curve.
3. Continue to define points along the path of the curve.
4. To define the final point, right-click and choose Finish from the Local Menu. If you want to close the curve, choose Close instead of Finish.

If at any point you wish to undo the segment of the curve that you just drew, right-click in the drawing area and choose One Step Back from the local menu.

An alternative to defining points along the path of the curve is to specify the length and angles of the line segments between the points, which form the frame of the curve. You can do this in the Edit Bar, pressing <Enter> to complete each segment of the frame, then pressing <Ctrl>+<F> to complete the entire curve. This procedure is essentially similar to drawing a multiline.



## Spline Curve

**Menu:** Insert|Curve|Spline      **Toolbar:** Insert

To create a Spline curve, define points along the intended path of the curve. The curve will pass through the defined points.

1. Activate the Spline Curve tool.
2. Define the first endpoint of the curve.
3. Continue to define points along the path of the curve.
4. To define the final point choose Finish from the Local Menu. If you want to close the curve choose Close instead of Finish.

If at any point you wish to undo the segment of the curve that you just drew, right-click in the drawing area and choose One Step Back from the local menu.

An alternative to defining points along the path of the curve is to specify the length and angles of the line segments between the points, which form the frame of the curve. You can do this in the Edit Bar, pressing <Enter> to complete each segment of the frame, then pressing <Ctrl>+<F> to complete the entire curve. This procedure is essentially similar to drawing a multiline.



## **Sketch Curve**

**Menu:** Insert|Curve|Sketch      **Toolbar:** Insert

The Sketch tool is used for freehand drawing.

1. Activate the Sketch tool.
2. Click the mouse to define the starting point of the sketch.
3. Move the mouse to draw a freehand polyline.
4. Click the mouse to define the final endpoint of the sketch polyline.



## Text Tool

**Menu:** Insert|Text

**Toolbar:** Insert

The features for creating, formatting, and manipulating text in TotalCAD 1.0 are powerful; however, entering text and placing it in your drawing is simple and intuitive:

1. Activate the Text tool.
2. If you wish to set the font, size, and attributes of text prior to typing, display the [Text Format toolbar](#) (using View|Toolbars) and choose settings from the toolbar. You can use any TrueType font installed in your system. Alternatively, you can right-click on the text tool and set the text attributes in the [Text property sheet](#).
3. Click in the drawing area at the point where you want to place your text.
4. Type the text that you want to place, using the <Backspace> key to make corrections. Press <Shift>+<Enter> to add a new line of text. To finish the entity, press <Enter>, or right-click and choose Finish from the Local Menu.

[Text Properties](#)

[Text Height and Angle Options](#)

[Text Alignment Options](#)

 **Note**

For information about installing TrueType fonts, see your Windows documentation.



## Text Properties

The Text tool has a special set of properties that are accessible through the Text property sheet. You can access the default properties of the Text tool by right-clicking on the tool. You can access the properties of selected text by double-clicking on a selection. The Text property sheet has options for setting font, style, text height, and color.

 **Tip**

 **See Also**

To edit the contents of the text, select the text object and choose Format|Properties, then edit the text in the Attribute field of the General property sheet. To reformat the text, use the Text property sheet in the same dialog.

[Manipulating Text](#), [Formatting Text](#), [Text Property Sheet](#)

## Text Height and Angle Options

The Local Menu offers options for changing the height and angle of text “on the fly” while the Text tool is active.

If you right-click and choose Height from the Local Menu, a rectangle will appear. Move the mouse to position the lower-right corner of the box until it is the desired size. Click the mouse to set the text height, then continue entering text.

If you choose Angle from the Local Menu, a rectangle will appear. Move the mouse to rotate the rectangle, then click the mouse. You can now continue entering text into the angled text frame.

You can also change the text height and by typing values into the Height and Angle fields of the Edit Bar.

 **Note**

 **Tip**

 **See Also**

The editing and formatting tools provide other options for rotating and resizing text more precisely after it has been drawn.

To use Ortho while rotating text, hold down the <Shift> key.

[Manipulating Text](#), [Formatting Text](#), [Text Property Sheet](#)

## **Text Alignment Options**

The text alignment options of the Text tool give you precise control over how the text is aligned relative to the point where it is inserted in your drawing. To access these options, activate the text tool, right-click in the drawing area, then choose Align (the command will be followed by two capital letters that vary, depending on the current alignment setting).

The figures next to each alignment setting show you graphically how the text is positioned relative to the insertion point, which is represented by a star. There are three vertical positions (Top, Center, and Bottom), and three horizontal positions (Left, Middle, and Right). When you re-open the Local Menu with the Text tool active, the letters next to the command represent the current settings. TL, for example, means that the the insertion point will be at the top-left corner of the text.



# Dimension Tools

**Menu:** Insert|Dimension **Toolbars:** Insert

TotalCAD contains a variety of dimension tools that you can use to display the measurements of lines and angles. Dimensions generally consist of a dimension line or arc, extension lines extending to the object being measured, and dimension text. Leader dimensions consist of dimension text and a simple leader line. *Dimensioning* an entity means to display a dimension that measures or otherwise marks that entity.

By default, dimension text contains a measurement of the dimensioned object or coordinates in current World units. (Leader dimensions, which contain text labels, are the exception to this rule.) You can override the default text of a dimension by specifying the text in the Edit Bar prior to finishing the dimension.

[Manual, Segment, and Entity Dimensioning](#)

[Associative Dimensions](#)

[Setting Dimension Properties](#)

[Horizontal Dimension](#)

Draw a linear dimension showing horizontal distance.

[Vertical Dimension](#)

Draw a linear dimension showing vertical distance.

[Parallel Dimension](#)

Draw a linear dimension showing absolute distance between two points.

[Angular Dimension](#)

Draw a angular dimension showing the angle formed by two lines, by points on the perimeter of a circle or arc, or the angle formed by any two points relative to a defined vertex.

[Radius Dimension](#)

Draw a dimension indicating the radius of an arc or circle.

[Diameter Dimension](#)

Draw a dimension indicating the diameter of an arc or circle.

[Leader Dimension](#)

Draw a simple line leader with label text.

## Manual, Segment, and Entity Dimensioning

Horizontal, vertical, and parallel dimensions are collectively known as linear dimensions. When you create a linear dimension, you have three primary choices:



You can draw the dimension manually, using the Coordinate Fields, Edit Bar, or the mouse and snap modes.



You can associate the dimension with a line segment.



You can associate the dimension with an entire entity (such as a multiline or group).

To choose one of these three modes:

1. Activate the dimension tool that you want to use.
2. Right-click in the drawing area to open the Local Menu.
3. Do one of the following:



Click to place a check mark next to Segment Dimensioning to associate the dimension with a line segment.



Click to place a check mark next to Entity Dimensioning to associate the dimension with an entity.



If you want to create the dimension manually, uncheck both Segment Dimensioning and Entity Dimensioning.

Once you have selected a dimensioning mode, the mode is saved with the drawing and remains in effect for that drawing unless you explicitly change it.

For further information on drawing linear dimensions, read the following sections on the individual dimension tools.



**Tip**



**See Also**

If you generally prefer one of the dimensioning modes over the others, you can save it in a drawing template.

Horizontal Dimension, Parallel Dimension, Vertical Dimension

## Associative Dimensions

Dimensions can be *associated* to objects. This means that you can freely move, rotate, or resize the objects with which the dimension is associated, and the dimension will retain its position relative to the objects, and the dimension text will change automatically to reflect changes resulting from the objects being resized or moved (provided that you did not override the automatic text in the Edit Bar when you created the dimension).

If you want a dimension to be associated with an object, you first need to be sure that the Associative Dimension option is turned on. To do this, first right-click on any dimension tool to bring up the Properties dialog, which will contain the property sheets for default dimension properties. If there is no check mark in the Associative Dimension check box, then click in the box. Click OK to close the dialog.

Once this box is checked, then any dimension that is “connected” to an object will be associated with the object. You can accomplish this in the following ways:



Use Segment Dimensioning.



Use Entity Dimensioning.



Use any snap mode to snap the dimension to an existing object.



Use a dimension tool that automatically connects to an entity. If you dimension an angle on a circle, for example, the dimension will be associated with the circle. If you dimension an angle between two lines, the dimension will be associated with both lines.



[Tips](#)

To tell whether a dimension is associated with an entity, select the entity. Any dimensions associated with that entity will be colored blue.

If you want to turn off default associative dimensions, first right-click on a dimension tool to bring up the Properties dialog. Then, in the Format property sheet, uncheck the Associative Dimensions check box.

## Setting Dimension Properties

The Dimension property sheets provide a large number of options for controlling the format of dimensions. You can set the default properties of dimensions by right-clicking on a dimension tool and then choosing settings in the Dimension property sheets. These settings are saved in your drawing or template and remain in effect until you explicitly change them.

If you want to set the properties of a dimension or a set of dimensions, activate the Select tool, select the dimensions that you want to format, then double-click on the selection to bring up the Properties dialog.

 **See Also**

## Dimension Property Sheets





## Horizontal Dimension

**Menu:** Insert|Dimension|Horizontal

**Toolbars:** Insert

Use the Horizontal Dimension tool to display the horizontal length of an entity, or to display the Horizontal distance between any two points that you define in the drawing space:

1. Activate the Horizontal Dimension tool.
2. If desired, change the current dimensioning mode, as described in the [Manual, Segment, and Entity Dimensioning](#).
3. Do one of the following:
  - If Segment or Entity Dimensioning mode is active, click on the segment or entity that you want to dimension.
  - If you are dimensioning manually, define two points, using any snap mode or the Coordinate Fields. (You can also specify the length and angle of a line connecting the two points in the Edit Bar.)
4. Define a final point to determine the direction and length of the dimension's extension lines. (You can also specify this length directly in the Edit Bar.)

To override the default, automatic dimension text, type the text you want in the Edit Bar prior to finishing the dimension.



## Vertical Dimension

**Menu:** Insert|Dimension|Vertical

**Toolbars:** Insert

Use the Vertical Dimension tool to display the Vertical length of an entity, or to display the vertical distance between any two points that you define in the drawing space:

1. Activate the Vertical Dimension tool.
2. If desired, change the current dimensioning mode, as described in the [Manual, Segment, and Entity Dimensioning](#).
3. Do one of the following:
  - If Segment or Entity Dimensioning mode is active, click on the segment or entity that you want to dimension.
  - If you are dimensioning manually, define two points, using any snap mode or the Coordinate Fields. (You can also specify the length and angle of a line connecting the two points in the Edit Bar.)
4. Define a final point to determine the direction and length of the dimension's extension lines. (You can also specify this length directly in the Edit Bar.)

To override the default, automatic dimension text, type the text you want in the Edit Bar prior to finishing the dimension.





## Parallel Dimension

**Menu:** Insert|Dimension|Parallel

**Toolbars:** Insert

Use the Parallel Dimension tool to display the absolute length of an entity (the distance between the two points on the entity that are farthest apart).

1. Activate the Parallel Dimension tool.
2. If desired, change the current dimensioning mode, as described in [Manual, Segment, and Entity Dimensioning](#).
3. Do one of the following:
  -  If Segment or Entity Dimensioning mode is active, click on the segment or entity that you want to dimension.
  -  If you are dimensioning manually, define two points, using any snap mode or the Coordinate Fields. (You can also specify the length and angle of a line connecting the two points in the Edit Bar.)
4. Define a final point to determine the direction and length of the dimension's extension lines. (You can also specify this length directly in the Edit Bar.)

To override the default, automatic dimension text, type the text you want in the Edit Bar prior to finishing the dimension.

 **Note**

When dimensioning a circle or arc, you can position the parallel dimension at any angle.



## Angular Dimension

**Menu:** Insert|Dimension|Angular

**Toolbars:** Insert

Use this tool to dimension angles in your drawing. You can dimension:



The angle formed by any two lines.



The angle of an arc.



The angle formed by any two points on the perimeter of a circle.



A freeform angle that you create by defining a vertex node and two points.

[Dimensioning an Angle Formed by Two Lines](#)

[Dimensioning Angles on Circles](#)

[Dimensioning Angles on Arcs](#)

[Dimensioning Freeform Angles](#)

## **Dimensioning an Angle Formed by Two Lines**

To dimension the angle formed between two lines

1. Activate the Angular Dimension tool.
2. Click on the first line.
3. Click on the second line. If you now move the mouse cursor between the two lines, TotalCAD forms a dimension showing the inner angle. If you move the mouse cursor outside the two lines, a dimension will be formed showing the outer angle.
4. Click to specify the position of the dimension arc.

## **Dimensioning Angles on Circles**

To dimension the angle formed between any two points on the perimeter of a circle:

1. Activate the Angular Dimension tool.
2. Click on the circle that you want to dimension. A rubber-band line will appear, extending from the center of the circle to the location of the mouse cursor.
3. Define a second point, which forms an angle with the first point you defined, with the centerpoint of the circle as the angle's vertex.
4. Define a third point to specify the location of the dimension arc. If you specify a location between the first two points, you will dimension the inner angle; if you specify a location outside the two points, you will dimension the outer angle. You can locate the dimension arc inside or outside the circle.

## Dimensioning Angles on Arcs

To dimension the angle of an arc:

1. Activate the Angular Dimension tool.
2. Click on the arc that you want to dimension.
3. Click to specify the location of the dimension arc. You can dimension either the inner or outer arc, and you can place the dimension either inside or outside the arc that you are dimensioning.

 **Note**



When you use the Angular Dimension tool on an arc, TotalCAD automatically dimensions the angle formed by the endpoints of the arc. If you want to dimension the angle formed by any other two points on the perimeter of the arc, you can employ the Angle Node technique described in [Dimensioning Freeform Angles](#), using snap modes to define points at the center and on the perimeter of the arc.

## **Dimensioning Freeform Angles**

You can dimension the angle formed by any two points in the drawing space relative to a defined vertex, using the Angle Node option from the Local Menu:

1. Activate the Angular Dimension tool.
2. Right-click in the drawing area to display the Local Menu, then choose Angle Node.
3. Define a point, using any snap mode or the Coordinate Fields, for the vertex of the angle.
4. Define two more points to form an angle with the vertex node.
5. Define a point to position the dimension arc. You can measure either the inner or outer angle formed by the three points.



## Radius Dimension

**Menu:** Insert|Dimension|Radius

**Toolbars:** Insert

Use the Radius Dimension tool to dimension the radius of an arc or circle.

1. Activate the Radius Dimension tool.
2. Click on the arc or circle that you want to dimension. TotalCAD draws a dimension from the center of the circle or arc to its perimeter.
3. Define a second point to position the dimension on the perimeter of the circle, and to specify the position of the dimension text. The text can be positioned either inside or outside the circle or arc.

Alternatively, you can click on the circle or arc that you want to dimension, specify the text, angle, and length of the dimension in the Edit Bar, then press <Enter>.



## Diameter Dimension

**Menu:** Insert|Dimension|Diameter

**Toolbars:** Insert

Use the Diameter Dimension tool to dimension the Diameter of an arc or circle.

1. Activate the Diameter Dimension tool.
2. Click on the arc or circle that you want to dimension. TotalCAD draws a dimension through the center of the circle or arc to opposed points on its perimeter.
3. Define a second point to position the angle of the dimension, and to specify the position of the dimension text. The text can be positioned either inside or outside the circle or arc.

Alternatively, you can click on the circle or arc that you want to dimension, specify the text, angle, and length of the dimension in the edit bar, then press <Enter>.



## Leader Dimension

**Menu:** Insert|Dimension|Leader

**Toolbars:** Insert

Use the Leader tool to create dimension text attached to a simple line leader pointing to a location in your drawing. The Leader tool is similar to the Multiline tools in that it can contain an arbitrary number of connected line segments, which can be oriented at any angle.

1. Define the starting point of the leader (the end that is pointing toward an object).
2. Define a second point to specify the second endpoint of the line segment. TotalCAD will draw a line between the two points with an arrowhead on the first point.
3. If necessary, define more points, adding new line segments to the leader.
4. Type the text of the note in the field on the Edit Bar.
5. Complete the leader by double-clicking, or choosing Finish from the Local Menu.

You can also use the Edit Bar to create a leader. First define the starting point of the leader, then enter the length and angle of the line segment in the Edit Bar. Press <Enter> to add a new line to the dimension without finishing the entity. Press <Ctrl>+<F> to complete the leader.

 **Note**

This tool serves a function similar to the Arrow (Selection) tool used in many standard Windows applications.



## Insert Picture

**Menu:** Insert|Picture    **Toolbar:** Insert

Use this command to insert a picture into your drawing. The picture file can be of type WMF (Windows metafile), BMP (bitmap), or DIB (device-independent bitmap). Once the image is inserted, you can move it, copy it, or scale it. You cannot, however, rotate pictures or edit their contents. If you want to change the contents of the picture, you need to edit it in a bitmap editor, such as Windows Paint, then re-insert it into your TotalCAD drawing.

To insert a WMF or bitmap picture:

1. Choose Insert|Picture to display the Insert Picture dialog.
2. Choose the type of graphic file you want to import from the List Files of Type dropdown list.
3. If necessary, navigate to the directory where the graphic file you need is located.
4. Choose the file that you want to open from the File Name list box and click the OK button.



**Tip**



**See Also**

You can edit a bitmap using Windows Paint, or any other bitmap editor that is registered in your Windows system, by simply double-clicking on the bitmap.



Insert Object



## Insert Object

**Menu:** Insert|Object    **Toolbar:** Insert

The Insert Object command lets you insert OLE (Object Linking and Embedding) objects from other Windows applications into your TotalCAD drawing. You can edit the object from within TotalCAD using the features and commands of the Windows application in which the object was created (the *source* application). For example, you can insert a Microsoft Word application into your drawing and edit the text or graphics using Microsoft Word tools.

To insert an OLE object into your drawing, first choose Insert|Object to display the Insert Object dialog. You can then Choose one of two basic options by clicking the Create New or the Create From File radio button (Create New is the default). The following topics describe how to insert objects using these two options.

[Create New](#)

[Create From File](#)

[Embedded and Linked OLE Objects](#)



**See Also**

Edit Links, Edit Object, Paste Special

## **Create New**

The Create New option lets you embed a new OLE object from another application into your drawing.

1. Choose Insert|Object to display the Insert Object dialog.
2. If you want an icon representing the OLE object to be placed in your drawing, click the Icon check box.
3. Choose an OLE application from the dialog's scrolling list: Either double-click the application's name, or select it and press <Enter>. The object will appear in your TotalCAD drawing.

You can create and edit the OLE object using tools from the external application.

## Create From File

The Create From File option lets you either embed or link an OLE object into your drawing based on a file previously created in the source application. See [Embedded and Linked OLE Objects](#) for an explanation of the difference between embedding and linking.

1. Choose Insert|Object to display the Insert Object dialog.
2. Click the Create From File radio button. The appearance of the dialog will change.
3. If you want the OLE object to be displayed as an icon in your drawing, click the Icon check box.
4. Check the Link check box if you want the OLE object to be linked; otherwise it will be embedded.
5. Type the path and name of the file that you want to use in the text box, or click the Browse button and locate the file using the Open dialog.

An OLE object based on the file will be inserted into your TotalCAD drawing. To edit the object in TotalCAD using the application that created the file, double-click the object, or select the object and choose Edit|Object. To update or edit links to a linked object, use the Edit|Links dialog.

 **See Also**

[Edit Object](#), [Edit Links](#), [Paste Special](#)

## Embedded and Linked OLE Objects

OLE objects that you insert can be either *linked* or *embedded*. Note these important differences between embedded and linked OLE objects:



Embedded objects become an integral part of your TotalCAD file, and take up more space in the file than a linked object. Embedded objects will move with the file if you transfer it to another computer system. If you make subsequent changes to the file on which the embedded OLE object was based, these changes will *not* affect the embedded OLE object.



Linked objects are references to the file on which they were based, and they retain their connection to that file. If you make changes to the file on which a linked OLE object is based, the changes will also affect the linked OLE object.



**See Also**



**Warning**

[Edit Links](#)



If you move a TotalCAD file containing a linked object to another system, you will need to move any linked files along with the TotalCAD file. Linked OLE objects are referenced to their files by a specification of their drive and path. If you move the linked file to another drive or directory, Windows will attempt to find the linked file, but if it cannot locate the file, you will need to change the drive path and directory using the Edit|Links dialog.

## Format Menu and Tools

The Insert menu provides access to tools that let you draw entities and commands that enable you to insert pictures, files, and OLE objects into your drawing. The following table summarizes the contents of the Insert menu:

### Properties

Edit the properties of objects.

### Create Hatch

Draw a hatch pattern over selected objects.

### Create Group

Define all objects in the selection as a group.

### Explode

Break up an object into its constituent parts.

### Bring to Front and Send to Back

Move selected objects to the front or back of the object stack on their layer.

### Align Commands

Line up all selected objects on the left, right, top, bottom, vertical center, horizontal center, or along a line that you specify.



## Properties

**Menu:** Format|Properties      **Toolbar:** Modify

Properties are similar to *attributes*. The Properties command displays the Properties dialog. Use this dialog to set the properties of objects, or the default properties of drawing tools.

The Properties dialog contains a set of property sheets. The General, Pen, and Brush property sheets are available whenever you display the dialog. Other property sheets (Double Line, Curve, Point, Text, and the three dimension property sheets) are displayed only when the tool to which they apply is activated, or if an entity to which they apply is selected.

[General Property Sheet](#)

[Pen Property Sheet](#)

[Brush Property Sheet](#)

[Curve Property Sheet](#)

[Double Line Property Sheet](#)

[Dimension Property Sheets](#)

[Point Property Sheet](#)

[Text Property Sheet](#)

 **See Also**

Select Commands, Property Toolbar

## General Property Sheet

The General property sheet of the Properties dialog lets you:



Select the layer of the drawing on which objects are drawn.



View or edit an object's *text attribute*.



Select a style (a set of pre-defined property settings).



Save a set of property settings as a style, or edit the property settings of an existing style.

The property sheet contains the following controls:

*Attribute.* To give selected objects a text attribute, click in the Attribute text box and type in whatever information you want to record. You might, for example, want to record the part number or cost of a physical component represented in the drawing. The Attribute text field is used to hold the names of groups.

*Layer.* To set the layer property, which determines the layer on which entities are drawn, click on the Layer drop-down list box, then click on the desired layer. You can create new layers using the Layers dialog (Tools|Layers).

*Style.* A style is a set of saved property settings.

[Selecting, Editing, and Creating New Styles](#)

## Selecting, Editing, and Creating New Styles

To select an style that has already been defined, click the down arrow on the right side of the Style text box, then click to select the style from the drop-down list that appears. The property settings that define that style will now become the default property settings of the tool, the properties of which you are editing.

To edit an existing style:

1. Select the style from the drop-down list.
2. Make changes to settings in the Properties dialog, in any of the property sheets that are available in the dialog.
3. Click the OK button to close the dialog.

To create a new style:

1. If you like, select a style that you want to use as a basis for a new style. (If you skip this step, then you will use whatever style is currently selected.)
2. Select a style name in the text box and type a name for the new style.
3. Click the New button. This creates a new style that has all the property settings of the style you selected in step 1.
4. Make any changes you like to settings in the property sheets, then click the OK button. The property sheet settings will be saved in the new style, which will be available the next time you open the Properties dialog for the tool for which you are setting properties.



Notes

Styles are applicable only if you are setting the default properties of a drawing tool. You cannot apply a style to selected objects, nor can you define a style based on the properties of selected objects.

There are separate sets of styles for points, single-line entities, double-line entities, arcs, curves, dimensions, and text. Circle and arcs share the same set of styles. The set of styles available to you depends on the tool whose properties you are modifying.

## Pen Property Sheet

The Pen property sheet lets you set properties of the drawing pen, such as its pattern, color, and thickness.

The Pen property sheet contains the following controls:

*Pattern.* Scroll through this list to choose a pattern for the line. You can choose either a continuous line, or one of many dot-and-dash patterns.

*Alignment.* Click this box if you want the pattern to align itself so that the corners of rectangles and polygons are always solid.

*Color.* Click in this box to choose a color for the drawing pen. If you select By Layer, the pen color will be set to the default color of the object's layer. This color is set in the Layers property sheet.

*Dash Scale.* You can modify the factor in this box to set the scale of the pen's dot-and-dash pattern. For example, if you set the scale to .5 (instead of 1, the default), the dots and dashes will be half the default scale, and the pattern will therefore be repeated twice as often.

*Width Scaling.* This setting determines whether the width of the line will be scaled, or remain the same size, when you zoom in and out of the drawing. If you select Device, the width will be scaled relative to your computer screen, and will remain the same size at any magnification. If you choose World, then the width will be scaled with your drawing, and will change size as you zoom in and out. This setting also applies to the size of the elements in the pen's dot-and-dash pattern.

*Width.* Click in this box to choose a width for the pen, or type a custom width directly into the box.

 **Note**

 **See Also**



You can create new colors using the Color Palette property sheet (Tools|Color Palette).

Layers Property Sheet

## Brush Property Sheet

The Brush property sheet lets you set fill properties of objects, as well as the properties of hatches. Fills appear only on closed entities, such as circles and polygons.

The Brush property sheet contains the following controls:

*Pattern.* To change the fill pattern, scroll through the Brush Style list box and click to select the pattern that you want. The name of the pattern will be displayed just above the Brush Style list box.

*Color.* To change the color of the fill, click on the Color drop-down list box, then scroll through the list and click to select the color that you want. Choose By Pen if you want the Brush color to match the Pen color. Choose By Layer if you want the color to match the default color of the object's layer (set in the Layers property sheet).

*Hatch.* Use the Hatch controls to control the scale and angle of the fill pattern. The default scale of the pattern is 1, so increasing it to 2 doubles the size of the pattern elements, and decreasing the scale to .5 halves the size of the components. Use the Angle spin button to change the angle of the hatch pattern, rotating it inside the filled entity (or type a custom value directly into the text box). If you check the Cross Hatch check box, the pattern will be drawn across the entity a second time, in a direction perpendicular to the first pattern. This gives you many more choices of hatch patterns.

 **Note**

 **See Also**

You can create new colors using the Color Palette property sheet (Tools|Color Palette).

[Color Palette Property Sheet](#), [Property Toolbar](#)

## Curve Property Sheet

The Curve property sheet is visible only when a Bezier or spline curve entity is selected, or when the Bezier or spline curve tool is active and no entities are selected. This property sheet does not apply to sketch curves.

The Curve property sheet contains the following controls:

*The number of segments between adjacent control points.* Curves are composed of a number of miniscule line segments drawn between control points. Spline curves intersect the control points; Bezier curves gravitate toward the control points but do not intersect them. Use this spin control to set the number of line segments drawn between each control points. Large numbers will round the contours of the curve; small numbers will flatten out the contours.

*Show Frame.* Check the Show Frame check box to display a multiline frame connecting the curve's control points. This provides visual feedback about how the curve is drawn.

*Curve Type.* Use the Curve type option buttons to set whether the curve is drawn as a Bezier or spline curve.

 **See Also**

## Curve Tools

## Double Line Property Sheet

The Double Line property sheet is visible only when a double line entity is selected, or when a double line tool is active and no entities are selected.

The Double Line property sheet contains the following controls:

*Separation.* To set the distance between the two lines of the double line, specify the distance in the Separation text box.

*Reference.* Use the Reference option buttons to determine the alignment of the double line nodes. These points determine how the double line will align itself to coordinate positions when you draw or position the line. You can set the double line nodes to align to its left line, its right line, or to a centerpoint exactly between the two lines. (You can edit these nodes directly in Node Edit mode).

*End Caps.* Click in the Start and End check boxes to determine whether the startpoint and endpoint of double lines are open or closed.

 **See Also**



Double Line Tools, Node Edit with Lines and Polygons

## Dimension Properties

Dimensions consist of three basic components:



The dimension lines, with arrows pointing to either end of the dimension.



Extension lines, which connect the dimension line to the object being dimensioned.



Dimension text, normally displaying the distance being dimensioned in World units.

Extension lines have optional line segments that continue the extension outward beyond the dimension line (extensions to the extension).

A dimension can have interior dimension lines or two exterior dimension lines.

The [Format property sheet](#) contains options that let you format dimension arrowheads, and text position.



**See Also**

## Dimension Tools

## Format Property Sheet

The Format property sheet gives you control over the shape of dimension arrows and the position of text relative to the dimension line.

The Arrowhead group contains the following controls:

*1st and 2nd.* Click in these drop-down boxes to choose dimension arrow shapes. The arrowheads can have angles of either 30 or 45 degrees; they can be filled, open, or closed. If you like, you can choose None (no arrow at all), or Tick (a simple angled line).

*Size.* Click in this control to choose a size for the length of the arrowhead, or type in a custom size.

*Tick Extension.* If you set the arrowhead to Tick in the 1st and 2nd boxes, you can specify the length of the tick's extension line in the Tick Extension control.

The Text Position group contains the following controls:

*Font.* Click in this box to choose a TrueType font for the dimension text. All TrueType fonts installed on your system will be available.

*Height.* Click in this text box to choose a text height, or type in a custom height. You can set the height in either World or Paper units, depending on the current mode.

*Position.* Choose Outside if you want the text placed outside of the dimension line, Above if you want the text "above" the dimension line ("above" is relative to the orientation of the text, not relative to your screen), or On Line if you want to place the text directly on the dimension line. This position can be manually adjusted (see the next control).

*On Line Adjustment.* Type a number into this text box to specify the distance of the text above or below the dimension line. A value of zero (the default value) will place the text on the same level as the dimension line. Higher values move the text above the line ("above" is relative to the orientation of the text); lower (negative) values move the text below the line.

*Force Text Horizontal.* Check this box if you want the dimension text to be horizontal no matter how the dimension is rotated.

*Force Dimension Text Inside Extension Lines.* Click this box if you want to keep dimension text between the extension lines, regardless of the distance between the extension lines. If you leave this box unchecked, TotalCAD will decide whether to place dimension text inside or outside the extension lines.

## Point Property Sheet

The point property is included in the properties dialog only when a point entity is selected, or when a point tool is active. It includes settings for changing the size of a point, its shape, and its color.

The Point property sheet contains the following controls:

*Size.* Type a value in this box to set the width and height of the point. This value affects all types of points except the dot, which is always displayed at a minimal size. You can set the size in either World or Paper units, depending on the current mode. (Check the icon at the upper-left corner of the ruler to see whether TotalCAD is showing World or Paper units.)

*Point Type.* Click in an option button to select the type of point (dot, star, square, cross, or circle).

*Color.* Click in this box and scroll to choose a color for the point from the color palette.

## Text Property Sheet

The Text property sheet is included in the Properties dialog only when a text entity is selected, or when the text tool is active. It includes options for setting font, style, height, color, text effects, and text justification.

The Text property sheet contains the following controls:

*Font.* Click on this box and scroll to select a TrueType font for the text. All the TrueType fonts that are installed in your system will be available.

*Height.* Click in this box and scroll to choose a text height, or type a custom text height directly into the box. You can set the height in either World or Paper units, depending on the current mode. (Note that text can be measured in points only if you are using Paper units.)

*Style.* Click in this box to choose a text style (Normal, Italic, Bold, or Bold Italic).

*Color.* Click in this box and scroll to select a color for the text from the color palette.

*Effects.* Check the options in this box to turn on any of these text effects: Underline, Strikethrough, Text Box (a simple border around the text), Hidden (used for notes that you don't wish to display), and All Caps.

*Justification.* Click an option button to choose whether the text is left-, center-, or right-justified. When you are creating text, left-justified text is typed to the right of the insertion point; right-justified text is typed to the left of the insertion point, and center-justified text is centered on the insertion point.

 **Note**

 **Tip**

If you change the justification of existing text, the change will have an effect only if the existing text has multiple lines.

You can get more precise control of the placement of text relative to the insertion point by using the Align feature. You can access this feature by right-clicking in the drawing area while the Text tool is active, and then choosing the Align... command from the Local menu. The Align feature can only be applied to text as it is being drawn; you can't use it to format existing text.





## Create Hatch

**Menu:** Format|Create Hatch    **Toolbar:** Entity Format

The Create Hatch command applies a hatch pattern to selected entities. To apply the hatch:

1. If you wish to change the hatch pattern or other hatch settings, right-click on the Hatch tool to bring up the Properties dialog, then click the Brush tab to bring the Brush property sheet to the front.
2. Use the Brush property sheet to format the hatch pattern. Formatting a hatch pattern is essentially the same as formatting a fill pattern, so you can refer to the [Brush Property sheet](#) topic for a description of the property sheet options.
3. Use the Select tools to select the object or objects that you want to hatch.
4. Click the Hatch tool (or choose Format|Hatch) to apply the hatch pattern to the selection.

 **Note**

 **See Also**

A hatch pattern, unlike a fill pattern, is an object independent of the objects to which it has been applied. You can select the hatch and manipulate it independently.

## Brush Property Sheet



## Create Group

**Menu:** Format|Create Group

Use the Create Group command to combine any currently selected drawing objects into a group.

1. Use any selection method to select a set of drawing objects. You can select drawing entities or groups.
2. Click the Create Group tool or choose Format|Create Group from the menu.

 **See Also**

Groups, Explode



## Explode

**Menu:** Format|Explode

Use the Explode command to break up any object into its constituent parts. You can explode an object or group of objects by selecting them, then choosing Format|Explode.

When you explode an object, you break it down to the next level of constituents. For example, you might explode a group that contains a group, a line segment, and some text. After exploding the outer group, you could then explode its constituents: the group into its parts, and the text into polylines.

[Tips](#)

[See Also](#)

You can use Explode to break hatch patterns into constituent entities, and to break text into polylines.

You can also explode double-line entities into groups of single-line entities. This is useful if you would like to scale the width of double lines along with an object that contains the double lines.

Create Group, Groups



## Bring to Front and Send to Back

In a TotalCAD drawing, objects on each layer are stacked in the order that you create them, from back to front. The object at the front of the stack is visible on top of all other objects on its layer. The object furthest to the back of the stack is overlapped by all other objects on its layer.

TotalCAD provides two commands, [Bring to Front](#) and [Send to Back](#), that will change the order of the entities in the stack, bringing the selection to the front, or sending it to the back.

 **Warning**

 **Tip**

 **See Also**

The Bring to Front and Send to Back commands will not function as you expect if the objects are on different layers and the layers have different redraw order values. The redraw order value of each layer is set in the Layers property sheet. Layers with higher redraw order values will always be in front of layers with lower redraw order values. If the objects are on different layers and the layers have equal redraw order values, then Bring to Front and Send to Back will work across layers.

If you want to systematically place some categories of objects in front of others, use the redraw order property of layers.

Layers Property Sheet

## **Bring to Front**

**Menu:** Format|Bring to Front

Use the Bring to Front command to bring any currently selected entities to the front of the entity stack. This command is especially useful when you are dealing with overlapping filled entities, or bitmaps.

1. Select the object or objects, using any selection method, that you want to bring to the front of the entity stack.
2. Choose the Bring to Front command.

## Send to Back

**Menu:** Format|Send to Back

Use the Send to Back command to send any currently selected entities to the back of the entity stack. This command is especially useful when you are dealing with overlapping filled entities, or bitmaps.

1. Select the object or objects, using any selection method, that you want to send to the back of the entity stack.
2. Choose the Send to Back command.

 **Note**

Sending an object to back has no effect on its layer, and changing its layer has no effect on its position in the entity stack.

## Align Commands

Use the Align commands to line up any currently selected objects. With all the Align commands except Along Line, the objects are lined up relative to the bounding box of the selection. The *bounding box* is a virtual rectangle that completely contains all of the selected objects. (Because TotalCAD draws a frame around the objects you have selected, you can see the extents of the bounding box.)

### [Align Bottom](#)

Along the bottom of the bounding box.

### [Align Middle](#)

Along a virtual horizontal line passing through the vertical center of the bounding box.

### [Align Top](#)

Along the top of the bounding box.

### [Align Left](#)

Along the left edge of the bounding box.

### [Align Center](#)

Along a virtual vertical line passing through the horizontal center of the bounding box.

### [Align Right](#)

Along the right edge of the bounding box.





## Align Bottom

**Menu:** Format|Align Bottom    **Toolbar:** Entity Format

Use Align Bottom command to align the bottom edges of the currently selected objects along the bottom of the selection's bounding box.

1. Select all the objects that you want to align, using any selection method. TotalCAD will create a rectangular frame around the selection.
2. Choose the Align Bottom command from the menu or toolbar.



## **Align Middle**

**Menu:** Format|Align Middle      **Toolbar:** Entity Format

Use the Align Middle command to center the selected objects vertically in the selection's bounding box.

1. Select all the objects that you want to align, using any selection method. TotalCAD will create a rectangular frame around the selection.
2. Choose the Align Middle command from the menu or toolbar.



## **Align Top**

**Menu:** Format|Align Top

**Toolbar:** Entity Format

Use the Align Top command to align the top edges of the currently selected objects along the top of the selection's bounding box.

1. Select all the objects that you want to align, using any selection method. TotalCAD will create a rectangular frame around the selection.
2. Choose the Align Top command from the menu or toolbar.



## **Align Left**

**Menu:** Format|Align Left

**Toolbar:** Entity Format

Use the Align Left command to align the left edges of the currently selected objects along the left edge of the selection's bounding box.

1. Select all the objects that you want to align, using any selection method. TotalCAD will create a rectangular frame around the selection.
2. Choose the Align Left command from the menu or toolbar.



## **Align Center**

**Menu:** Format|Align Center      **Toolbar:** Entity Format

Use the Align Center command to center the currently selected objects at the horizontal center of the selection's bounding box.

1. Select all the objects that you want to align, using any selection method. TotalCAD will create a rectangular frame around the selection.
2. Choose the Align Center command from the menu or toolbar.



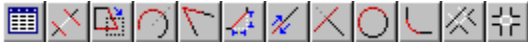
## **Align Right**

**Menu:** Format|Align Right      **Toolbar:** Entity Format

Use the Align Right command to align the right edges of the currently selected objects along the right edge of the selection's bounding box.

1. Select all the objects that you want to align, using any selection method. TotalCAD will create a rectangular frame around the selection.
2. Choose the Align Right command from the menu or toolbar.

## Modify Menu and Tools



The tools on the Modify menu are conventionally called *CAD editing* tools. These tools provide a variety of special editing functions for modifying the shape of entities.

### Object Trim

Defines entities as a cutting edge, then trims objects in and around the cutting edge.

### Stretch

Stretches an entity composed of line segments by moving a selected set of vertices.

### Split

Splits an entity into two entities.

### Meet 2 Lines

Shrinks or extends two lines or double lines so that two of their endpoints form a clean intersection.

### Chamfer

Joins two line segments or double line segments with a flattened corner.

### Line Length

Changes the length of a line while retaining its connection with other lines in a polygon or multiline.

### Shrink/Extend Line

Changes the length of a line so that it precisely meets another entity.

### Arc Complement

Reverses the starting and ending angles of an arc.

### Fillet

Connects two lines, either single or double lines, with a smoothly fitted arc.

### T-Meet 2 Double Lines

Forms a clean T intersection of two double lines.

### Intersect 2 Double Lines

Forms a clean X intersection of two double lines.



Note

There is practical distinction between the tools on the Modify menu and those on the Edit menu. When you use the Edit menu tools, you must first select the object or objects that you want to edit. The Modify menu tools are similar to the drawing tools, in that you first activate the tool, then follow a series of steps to complete the edit. When you finish the edit, the tool remains active, so that you can either begin a second editing operation of the same type, or switch to another tool.





## Object Trim

**Menu:** Modify|Object Trim      **Toolbar:** Modify

Object Trim is a powerful editing tool that lets you define an entity or set of entities as a *cutting edge*. You can then cut out any parts of the drawing in or around the cutting edge tool, leaving only the parts of the drawing that you want to keep. You can trim any part of the drawing that touches the cutting edge. To use Object Trim:

1. Draw a new entity (or a set of entities) over an existing entity in such a way that the new entity defines areas of the existing entity that you want to remove. You might, for example, want to use a circle to remove the ends of intersecting lines that fall outside of the circle.
2. Activate the Object Trim tool.
3. Click on the new entity to define it as a cutting edge. You can continue to add entities to the cutting edge by holding down the <Shift> key and clicking on additional entities.
4. Click on parts of the drawing in or around the cutting edge to remove it from the drawing. TotalCAD will not allow you to remove the last remaining part of the drawing touching the cutting edge.

**Warning**

Object trim will not work with certain complex entities, specifically curves, dimension, and text. To use the tool on these types of object (as well as on groups), you must first select the object that you want to trim, then use the Format|Explode command to break the entity into simple sub-entities. In the case of text, you must explode the entity twice (first into groups of multilines, then into multilines).



## Stretch

**Menu:** Modify|Stretch   **Toolbar:** Modify

Use the Stretch tool to move groups of nodes on any object or set of objects. This has the effect of stretching (or shrinking) the object by relocating the selected set of nodes. To move the nodes you use a vector line, similar to the vector line used in the Vector Copy command.

1. Choose the Stretch tool.
2. Click to define the first corner of the selection rectangle. Drag the mouse over the drawing so that the rectangle encloses all of the nodes (vertices) that you want to move, then click to complete the selection rectangle. A small rectangle will appear at each of the nodes that you have selected.
3. Define the first point of a vector line, either by clicking the mouse (using any snap mode), or by entering the point's coordinates in the status bar.
4. Specify the length and angle of the vector. You can do this either by directly entering the length and angle in the edit bar, or by defining the vector line's second point. You can define the second point using either the mouse (with any snap mode), or by entering the point's coordinates in the status bar.

All of the selected nodes will move, from their original locations, the length and angle defined by the vector line. The objects in which the nodes were selected will stretch, shrink, or move accordingly.

**Note**

**See Also**

If you select all of the nodes on any entity, the entire entity will simply be moved (as with the Vector Copy command). This is also the effect if you select any node on a circle or arc.

Node Edit



## Split

**Menu:** Modify|Split      **Toolbar:** Modify

Use the Split tool to divide an entity into two separate entities. This tool works on any single-line or double-line entity, as well as on circles, and arcs.

[Splitting Line and Double Line Entities](#)

[Splitting Arcs and Circles](#)

 **Note**

 **See Also**

You can also use this tool to split curves and other complex entities, but you need to explode the entity first.

Explode



## Splitting Line and Double Line Entities

You can use the Split tool to split entities into two parts.

1. Activate the Split tool.
2. Click on the line or double-line that you want to split: Let's call this the *target*. TurboCad draws a perpendicular line intersecting the target.
3. Move the mouse cursor until the perpendicular line is at the point where you want to split the target.
4. Click to split the target.

 **Note**

If the target is a closed entity (a rectangle or polygon), then you will need to split the entity in two places to divide it into two entities.

## **Splitting Arcs and Circles**

To divide an arc into two arc segments:

1. Choose the Split tool.
2. Click on the arc that you want to split. A dotted line appears, connecting the cursor to the centerpoint of the arc.
3. Click to define the angle at which to split the arc.

Splitting a circle is the same as splitting an arc, except that at step 3 you need to define a starting and an ending angle (breaking the circle in two places).



## Meet 2 Lines

**Menu:** Modify|Meet 2 Lines      **Toolbar:** Modify

Use the Meet 2 Lines function to shrink or extend two lines or double lines so that two of their endpoints meet. This function automatically cleans up the corner formed by two double lines.

1. Activate the Meet 2 Lines tool.
2. Click on the first, then the second line or double line.

When the two lines are crossed, the two lines will be shrunk in the direction of the mouse clicks. In other words, the two parts of the line that you click on will be preserved; the parts of the lines on the opposite side of the intersection will be trimmed.



## Chamfer

**Menu:** Modify|Chamfer **Toolbar:** Modify

Use the Chamfer tool to join two line segments or double line segments with a flattened corner. You can also use the Chamfer Multiline option to chamfer all the corners of a multiline or polygon.

When you activate a chamfer tool, the values you can use to calculate the chamfer appear in the edit bar.

The Chamfer tool draws a chamfer between points on each line calculated by their distance from the point of intersection on the first (Distance A) and second (Distance B) lines clicked:

1. Activate the Chamfer tool.
2. Enter values for distances in the Dist. A and Dist. B fields of the Edit Bar. Dist. A is the distance on the first line clicked. Dist. B is the distance on the second line clicked. Return to the drawing area by clicking there or pressing <Ctrl>+<D>.
3. Do one of the following:
  -  Click the first line, then the second line.
  -  Right-click in the drawing area to open the local menu, then choose Multiline to chamfer all the corners of an multiline or polygon.



## Line Length

**Menu:** Modify|Line Length      **Toolbar:** Modify

Use the Line Length tool to to extend or shrink a line segment from either endpoint. If the line segment is part of a multiline or polygon, as you change the length of the line segment, it retains its relationship to other lines within a multiline or polygon. This means that lines connected to the line endpoint you are moving will also change angle and length so that they remain connected to the endpoint.

1. Activate the Line Length tool.
2. Click on a line segment at a position near the endpoint that you want to modify. The segment now extends from the endpoint furthest from the mouse to the mouse cursor.

3. You can now set the line length in one of three ways:



Enter an absolute length for the line in the Length field of the edit bar, then press <Enter>.



Enter the amount that you want to lengthen the line in the Delta L field of the edit bar (enter a negative value to shorten the line) then press <Enter>.



Drag the mouse to lengthen or shorten the line, then click when it is the length you want.



## Shrink/Extend Line

**Menu:** Modify|Shrink/Extend Line

**Toolbar:** Modify

Use the Shrink/Extend to Object editing tool to edit the length of a line so that it precisely meets another entity. You can either extend the line, increasing its length until it meets the other entity; or you can shrink the line, decreasing its length until it just meets the other entity. To use this tool:

1. Activate the Shrink/Extend to Object tool.
2. Click on the line or double line that you want to shrink or extend.
3. Click on the entity to which you want to shrink or extend the line. TotalCAD will adjust the length of the line or double line until it precisely meets the second entity.



## Arc Complement

**Menu:** Modify|Arc Complement **Toolbar:** Modify

Use the Complement tool to reverse the starting and ending angles of an arc. If, for example, the arc is drawn from 0 degrees to 90 degrees, the complement of the arc will be drawn from 90 degrees to 0 degrees.

To change an arc to its complement, first choose the Arc Complement tool, and then click on the arc that you want to modify.





## Fillet

**Menu:** Modify|Fillet      **Toolbar:** Modify

Use the Fillet tool to connect two lines, either single or double lines, with a smoothly fitted arc. The result is a rounded corner. The Multiline option of this tool allows you to fillet all of the corners of a multiline or polygon simultaneously.

TotalCAD inserts an arc of a specified radius between two intersecting lines, making the arc tangent to both lines. The start and end angles for the arc are determined by the points at which the arc is tangent to the two lines. TotalCAD then trims or extends the lines to form the smooth corner.

1. Activate the Modify|Fillet tool.
2. Enter the radius for the filleted arc in the Radius text box on the edit bar and press <Enter>. (If the edit bar is not visible, display it using View|Toolbars.)
3. Do one of the following:



Click the first line, then the second line.



Right-click in the drawing area to open the local menu, then choose Multiline to fillet all the corners of an multiline or polygon.

### Note

If the radius specified is too large for the arc to fit between the two lines, or if you specify a radius of zero, TotalCAD will be unable to perform the fillet, and will display an error message.



## T-Meet 2 Double Lines

**Menu:** Modify|Length    **Toolbar:** Modify

Use the T-Meet 2 Double Lines tool to form a clean T intersection of two double lines. The first line clicked will extend or shrink to form the stem of the T.

1. Click on the line that will form the stem of the T.
2. Click on the line that will form the top of the T.



## Intersect 2 Double Lines

**Menu:** Modify|Length    **Toolbar:** Modify

Use the Intersect 2 Double Lines tool to clean up the intersection of two double lines.

1. Activate the Intersect 2 Double Lines function.
2. Click on the first, then the second of two intersecting double lines.

## **Modes Menu and Tools**

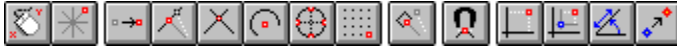
### Snap Tools

Set the current snap mode.

### Coordinate System

Set the current coordinate system.

## Snap Tools



**Menu:** Modes|Snaps    **Toolbar:** Snap

### Combining Snap Modes

#### No Snap

Turn off all snap modes; point is defined at the location of the mouse click.

#### Snap to Vertex

Snap to the nearest end of a line segment, endpoint of an arc, or corner of a polygon or multiline.

#### Snap to Near Point

Snap to the nearest point on any entity.

#### Snap to Midpoint

Snap to the midpoint of a line segment.

#### Snap to Arc Center

Snap to the center of an arc or circle.

#### Snap to Quadrant Point

Snap to the nearest quadrant point (at 0, 90, 180, or 270 degrees) on the circumference of a circle or arc.

#### Snap to Intersection

Snap to the nearest intersection of two line segments.

#### Snap to Grid

Snap to the nearest grid point or intersection of grid lines.

#### Ortho

Constrain the angles of lines; the default settings constrain them to 0, 90, 180, or 270 degrees. (You can change these settings in the Angle property sheet.)

#### Magnetic Point

Increase visual feedback; the rubber band line you are currently drawing will gravitate toward the snap point.

## Combining Snap Modes

Some snap modes can be combined with one another, and some are mutually exclusive. No Snap turns off all other snap modes. Near Point snap (which snaps to the nearest point on any object) also turns off all other snap modes. (Ortho and Magnetic point can stay on no matter which snap modes are active.)

The remaining snap modes all snap to a particular node (nodes are key points that define the object geometrically), and they are not mutually exclusive. You can, for example, have Snap to Grid and Snap to Vertex on simultaneously. If you then click the mouse while a grid point and a line segment are both within the snap aperture, your point will snap to either an endpoint of the line segment, or to the grid point, whichever is closest to the mouse cursor.



The snap modes that can be used at the same time are all in the same section of the Modes|Snaps submenu.





## No Snap

**Menu:** Modes|Snap|No Snap    **Toolbar:** Snap

Activating No Snap will turn off all snap modes except Ortho. With snap modes turned off, you define points by simply clicking at the point that you want to define in the drawing area, or by entering the point's coordinates on the status bar.

 **Note**

 **See Also**

If Ortho mode is on, the second point of a line will snap to a point to constrain the angle of the line, even if No Snap mode is active.

Ortho



## Snap to Vertex

**Menu:** Modes|Vertex    **Toolbar:** Snap

When Snap to Vertex mode is active, the point you are defining will snap to the nearest vertex on an entity within the snap aperture. A vertex in TotalCAD can be:



The endpoint of any line segment (either a simple line or a segment of a multiline).



A corner of a polygon.



The endpoint of an arc.



**Note**

When snapping to a double line, your point will be defined at the endpoint of either of the single line segments that comprise the double line, not to the double line's reference point.



## **Snap to Nearest on Graphic**

**Menu:** Modes|Snap|Nearest On Graphic

**Toolbar:** Snap

When Snap to Near Point mode is active, a point will be defined at the nearest point on an object within the snap aperture.



## **Snap to Midpoint**

**Menu:** Modes|Snap|Middle Point

**Toolbar:** Snap

When Snap to Midpoint mode is active, the point you are defining will snap to the midpoint of a line segment. A line segment can be any part of an entity, such as one of the lines in a double line, or a segment of a multiline or polygon.



## **Snap to Arc Center**

**Menu:** Modes|Snaps|Arc Center

**Toolbar:** Snap

When Snap to Arc Center is active, the point you are defining will snap to the center of an arc or circle that you indicate. To indicate the arc or circle, click on or near its perimeter.





## **Snap to Quadrant Point**

**Menu:** Modes|Snaps|Quadrant Point    **Toolbar:** Snap

When Snap to Quadrant Point is active, the point you are defining will snap to the nearest quadrant point. Quadrant points are at 0 degrees, 90 degrees, 180 degrees, and 270 degrees on the circumference of an arc or circle.



## **Snap to Intersection**

**Menu:** Modes|Snaps|Intersection

**Toolbar:** Snap

When Snap to Intersection is active, the point you are defining snaps to the intersection of two lines. Both lines must be located within the snap aperture; but the intersection need not be located within the snap aperture.



## Snap to Grid

**Menu:** Modes|Snaps|Grid

**Toolbar:** Snap

When Snap to Grid is active, the point you are defining snaps to the nearest grid point, or to the nearest intersection of grid lines, if the grid is being displayed in lines rather than points or crosses.



**See Also**

## Grid Property Sheet



## Ortho

**Menu:** Modes|Snaps|Ortho      **Toolbar:** Snap

Ortho mode constrains the angle of the entity you are drawing. By default, ortho mode will constrain the angle of the entity so that it will be parallel to either the X or Y axis of the drawing. You can change these default settings in the Angle property sheet.

 **See Also**

Angle Property Sheet



## Magnetic Point

**Menu:** Modes|Snaps|Show Magnetic Point      **Toolbar:** Snap

Magnetic Point provides visual feedback about the location of snap points. When this mode is active, as you move the mouse cursor within a critical distance from the snap point, the rubber band line will be attracted to the snap point, showing you its exact location. Unless speed is a consideration on your computer, or unless you are working on a very large drawing, it is probably useful to have this mode on all the time.

 **See Also**

## Snap Aperture



## Coordinate System

**Menu:** Modes|Coord System    **Toolbar:** Snap

The coordinate system menu commands enable you to change between absolute, relative, and polar coordinates, and to relocate the origin of the relative and polar coordinate systems.:

### [Coordinates: Absolute](#)

Change to absolute Cartesian coordinates, in which you define points by their distance from a fixed (absolute) point of origin.

### [Coordinates: Relative](#)

Change to the relative coordinate system, in which you define points by their X and Y distance from the Relative Origin point.

### [Coordinates: Polar](#)

Change to the polar coordinate system, in which you define points by their distance and angle from the Relative Origin point.

### [Relocate Origin](#)

Change the position of the Relative Origin point.

 **See Also**

## Coordinate Systems



## **Coordinates: Absolute**

**Menu:** Modes|Coord System|Absolute    **Toolbar:** Snap

Use the absolute coordinate system to work in absolute Cartesian coordinates. When working in absolute coordinates, you define points by their X and Y distances from the Absolute Origin. This fixed point, absolute (0,0) is located by default at the lower-left corner of the drawing page (although the page can be moved using the Page Setup dialog).

For example, to draw a horizontal line two units long starting at (2,2), you would define the line's second point at (4,2).

 **See Also**

## Coordinate Systems



## Coordinates: Relative

**Menu:** Modes|Coord System|Relative    **Toolbar:** Snap

Use the relative coordinate system to work in relative Cartesian coordinates. When you work in relative coordinates, you define the location of points relative to the Relative Origin (relative (0,0)). To define a point relative to the current position, you can enter X and Y displacement values in the Coordinate Fields. You can enter either positive or negative values. The displacement values specify the distance of the new point from the Relative Origin.

 **Note**

 **See Also**

The Relative Origin moves whenever you define a new point. In most cases, the Relative Origin will be the last point you defined. If you are drawing a circle or arc, however, the Relative Origin will be at the center of the circle or arc.

[Coordinate Systems](#), [Relocate Origin](#), [Program Setup Dialog](#)



## Coordinates: Polar

**Menu:** Modes|Coord System|Polar      **Toolbar:** Snap

Use the polar coordinate system when you want to define points by their distance and angle from the Relative Origin Point. When you start a new drawing, this point is located at the same position as the Absolute Origin Point, at absolute (0,0). You can move the Relative Origin Point using the Relocate Origin command.

Let's say, for example, that you have defined the first point of a line in any coordinate system, and you want to draw a line exactly 3.5 units long at a 45 degree angle from the first point. You can accomplish this by following these steps:

1. If you are in another coordinate system, switch to polar coordinates.
2. Type 3.5 into the length text box on the status bar.
3. Type 45 into the angle text box on the status bar, then press <Enter>.

 **See Also**



## Coordinate Systems



## Relocate Origin

**Menu:** Modes|Coord System|Relocate Origin    **Toolbar:** Snap

Use the Relocate Origin command to manually relocate the Relative Origin point. When you start a new drawing, this point is located at the lower-left corner of the page, and appears as a red dot. You can relocate the point using any coordinate system; that is, you can specify its position in absolute, polar, or relative coordinates. To relocate the point of origin:

1. Choose Relocate Origin from the menu.
2. Define a point for the new origin by clicking the mouse in the drawing space or entering values in the status bar (the type of values you enter in the status bar depend on the currently active coordinate system).

**See Also**

## Coordinate Systems

## Tools Menu and Dialogs

The following table summarizes the commands accessible from the Tools menu. Only the major menu items are shown. The indented items on the menu are shortcuts that go directly to individual property sheets in the Program Setup and Drawing Setup dialogs. These property sheets will be described in detail in the following sections.

### [Program Setup Dialog](#)

Display the Program Setup dialog, in which you can set options that are saved when you exit TotalCAD.

### [Drawing Setup Dialog](#)

Display the Drawing Setup dialog, in which you can set options that are saved with your drawing.

## Program Setup Dialog

**Menu:** Tools|Program Setup

The Program Setup dialog contains five property sheets for controlling TotalCAD program settings:

### [General Property Sheet](#)

Specify template, read-only file access, user information, snap aperture, zoom factor, save, and advanced settings.

### [Desktop Property Sheet](#)

Display and hide user-interface controls, control desktop colors and paper display.

### [Groups Property Sheet](#)

Set options for naming groups.

### [File Location Property Sheet](#)

Specify directories for autosave, backup, and template files.

### [Color Palette Property Sheet](#)





Add new colors to the palette, modify existing colors, and delete colors from the palette.

You can access each of these property sheets directly from the Tools menu, which gives you the convenience of opening the Program Setup dialog with the correct property sheet in front using a single command. The options that you set in this dialog are saved when you exit TotalCAD, and will remain in effect the next time you start the program.

## General Property Sheet

**Menu:** Tools|Program Setup

In the General Property sheet, you can set general program options and user information. The controls on this property sheet enable you to:

-  Set the default template used when you start a new drawing, and specify whether TotalCAD will prompt you for a template when you start a new drawing.
-  Specify whether drawings are opened as read only files.
-  Read and update user information.
-  Set the zoom factor and snap aperture.

The General property sheet contains the following controls:

*Default Template for New Files.* If you do not specify a template when you start a new drawing, TotalCAD will use a default template. To change the default template, click the arrow on the right side of the drop-down list and choose a template (or type a template name directly into the box).

*Prompt for Template Name.* Check this box if you want TotalCAD to prompt you for a template whenever you start a new drawing. If you want to use the default template without being prompted, clear this check box.

*Open All Files as Read Only.* There may be times when you want to look at drawing files without changing them, or prevent them from being inadvertently changed by someone else. Check this box to prevent anyone using TotalCAD from changing existing files. To restore the ability to change files, clear the check box.

*User Information.* The User Information text fields contain the user and company name.

*Snap Aperture.* The *snap aperture* is a circular area around the mouse cursor. TotalCAD will detect and snap to a potential snap point only if it is within the snap aperture. (Which points qualify as potential snap points depends on the current snap mode.) You can set the radius of the snap aperture using the snap aperture control.

*Zoom Factor.* The zoom factor setting controls how much you zoom in or out of the drawing when you use the Zoom In (<Gray +>) or Zoom Out (<Gray ->) command. The default zoom step setting is 2, which is a factor, meaning that if you zoom in one step, objects will be twice as big; if you zoom out one step, objects will be half as big, and the area of the drawing shown in the view will double. You can set the zoom step to any number between 1.01 and 2000.

[Save Options](#)

[Advanced Settings](#)

 **Tip**

 **See Also**

You can view the Snap Aperture as you work by choosing [View|Aperture](#).

[Save As Dialog](#), [Open Dialog](#), [New](#), [Snap Aperture](#), [Snap Modes](#), [Snap Tools](#), [View Crosshairs](#), [Zoom In](#),  
[Zoom Out](#), [Zoom Commands](#)



## Save Options

The General property sheet includes several controls that allow you to control aspects of how TotalCAD saves drawing files.

*Prompt for summary information.* If this box is checked, TotalCAD will display the Summary Information dialog, in which you can record information about your drawing, the first time that you save any drawing to disk, and whenever you use Save As to save a new copy of the file.

*Make backup copies.* If this box is checked, TotalCAD will automatically make backup copies of your drawings whenever you save them. You can choose either the On First Save or On Every Save option button to determine how often TotalCAD saves your backup files. Backup files share the same file name as the drawing file, but have the file extension .BAK instead of .TCW. By default, backup files are saved in the same directory as your primary drawing files. You can specify another directory in the File Location property sheet of this dialog.

*Save drawings every.* If this box is checked, TotalCAD will make autosave copies (.ASV files) of your document automatically at regular intervals. You can set the duration of the intervals in minutes using the Minutes control just below the check box.

*Save desktop on exit.* If this box is checked, TotalCAD will save the desktop settings when you exit the program. This means that TotalCAD will “remember” which toolbars and other controls you have displayed, as well as their positions on the desktop, and will display them exactly the same way the next time you start the program.

 **See Also**

[Save](#), [Save All](#), [Save As Dialog](#), [Desktop Property Sheet](#)

## Advanced Settings

*Fixed Relative Origin.* Check this box if you want to fix the Relative Origin in place while you are in relative or polar coordinates. You will have to relocate the Relative Origin manually using the Modes|Relocate Origin command.





 **See Also**

## Coordinate Systems

## Desktop Property Sheet

**Menu:** Tools|Desktop

The Desktop property sheet contains controls that let you:

-  Display and hide toolbars and other interface controls.
-  Display and hide the paper margins and background.
-  Set the color of the paper and background.
-  Set the zoom factor.

The Desktop property sheet contains the following controls:

[Toolbar Controls](#)

[Desktop Controls](#)

[Paper Margins and Background](#)

[Desktop Colors](#)

 **Note**

 **See Also**

Many of the functions available on the Views property sheet are also available from the View menu.

[View Toolbars](#), [View Grid](#), [View Crosshairs](#), [View Rulers](#), [View Edit Bar](#), [View Symbol Library](#), [View Selection Info](#), [Zoom In](#), [Zoom Out](#), [Zoom Commands](#)

## Toolbar Controls

*Toolbars.* The Toolbars scrolling list displays the names of all the toolbars available in TotalCAD. Toolbars that are currently displayed are checked. To display a hidden toolbar, scroll through the list and click on the check box next to the name of the toolbar, displaying the check. To hide a currently displayed toolbar, click on the box, removing the check.

*Color Buttons.* Check this box for color buttons, clear it for grayscale buttons.

*Large Buttons.* Check this box for large buttons, clear it for standard size buttons.

*Show Tooltips.* Check this box to display tooltips (the yellow boxes that appear under the button if you hold the mouse cursor over them); clear the box to suppress display of tooltips.





This function is also available in the View Toolbars dialog.

## Desktop Controls

The Views property sheet contains a number of check boxes that control the display a variety of user interface controls and palettes, specifically:

- Edit Bar
- Status Bar
- Scroll Bars
- Rulers


Checking a box displays the corresponding control; clearing the box hides the control.




It's a good idea to switch off controls that you don't use, because this increases the size of your drawing area.

## Paper Margins and Background

The Paper Display control group lets you display or hide the paper margins, and set whether the background behind the paper is colored or white:

 *Margins*. Check this box to display page margins; clear the box to hide page margins.

 *Background*. Check this box to display a background behind the paper; clear the Background check box to display a white background.

 [Tip](#)

You can switch off the paper until you're ready to print, then use the Fit feature of the Page Setup Dialog to place your drawing on the page.

## Desktop Colors

The Desktop Color control group contain buttons for customizing the color of the paper, background, and floating palettes. Click in the drop down box and Select Paper, Background, or Floater. Click the Modify button to change the color using the Color Picker.

 [See Also](#)

[Using the Color Picker](#)

## Groups Property Sheet

**Menu:** Tools|Groups

The Groups property sheet lets you control whether and how TotalCAD generates names for groups , and set whether TotalCAD prompts for group names.

[Groups Options](#)

[Automatic Group Numbering](#)

 **Tip**

 **See Also**





The name of a group is displayed on the General property sheet of the Properties dialog. To view this dialog, you can simply double-click on a group.


Create Group, Properties

## Groups Options

The following controls let you determine how TotalCAD handles the naming of groups:

 **Generate group names.** If this option is checked, TotalCAD automatically generates a name whenever you create a group.

 **Group name prefix.** If the Generate Group Names option is on, you can type a string (a series of characters) into the Group Name Prefix text box, and TotalCAD will use this string to automatically generate group names. All the characters that you type into the box are used literally, except the '@' character, which is a placeholder for a group number that TotalCAD generates automatically. You can type the '@' character in the name at the place where you want the automatic number to appear. For an explanation of how this works, see [Automatic Group Numbering](#).

 **Prompt for name.** Check this option if you want TotalCAD to prompt you for a name every time you create a new group.

 **Note**

 **Tip**

 **See Also**

If you turn off both the Prompt for Name and Generate Group Names options, TotalCAD will not name groups.

You can subsequently name or rename a group by editing its properties. To do this:

1. Select the group that you want to rename.
2. Double-click on the group to bring up the Properties dialog.
3. Type a name for the group into the Attribute text field, then press <Enter> to close the dialog.

Create Group, Explode, Properties

## Automatic Group Numbering

When TotalCAD creates a group name, it automatically gives the group a number, which increases by 1 whenever you add a new group to the drawing. This ensures that all groups have unique names. By default, the names that TotalCAD generates are “Group 1”, “Group 2”, and so forth.

You can change the default that TotalCAD uses for automatically naming groups in the Group Name Prefix text box. The names that you type in this text box have two parts: the name itself (any series of characters), and the ‘@’ character, which is a placeholder for the automatic number. You can place the ‘@’ character anywhere in the name, to specify where TotalCAD will place the automatic number.

For example, if you type “Part @” in the Group Name Prefix text box, then create the first group in a drawing, the group will automatically be named “Part 1”, and the next group you create will be named “Part 2”. If you type “#@ gear” into the Group Name Prefix text box, then create the first group in a drawing, the group will be named “#1 gear”, and the next group you create will be named “#2 gear”.

 **Note**

 **See Also**

The first '@' that you include in the name is a placeholder for the automatic number. Any '@' characters that occur after the first one will be part of the literal name.



Create Group, Groups Options

## File Location Property Sheet

**Menu:** Tools|File Location

The File Location property sheet allows you to specify the directories where TotalCAD places these different types of program files:

*Autosave* (.ASV). These are program files that TotalCAD saves automatically, if this option is turned on in the Save property sheet. These will allow you to restore work that would otherwise be lost due to a system crash.

*Backup* (.BAK). These are copies of drawing files that TotalCAD saves, if you check the Make Backup Copies option in the Save property sheet.

*Templates* (.TCT). Files in which you save settings as well as symbol libraries that you can start with as the basis of a new drawing.

To specify a directory in which to place a type of file:

1. In the list box, select the file type for which you want to specify a directory.
2. Click the modify button. TotalCAD will display a standard Windows Open dialog.
3. Use the standard Windows dialog to navigate to the drive and directory where you want to place the files. Click Open (or OK if you are using Windows 3.1) to accept the directory.



**Tip**

If you have two disk drives (or access to a network drive), it is wise to have your backup files automatically saved to a second drive. If the drive that contains your drawing files is damaged, you can recover your backup files from the second drive.

## Color Palette Property Sheet

**Menu:** Tools|Colors

The Color Palette dialog displays all the colors that are currently available. These colors and their names are displayed in a scrolling list, and also in a palette control to the right of the scrolling list. The Color Palette dialog lets you add new colors to the palette, modify existing colors, and delete colors from the list.

[Adding a New Color](#)

[Modifying an Existing Color](#)

[Deleting a Color](#)

[Picking Colors with the Mouse](#)

[Setting RGB Values](#)

[Setting Hue, Saturation, and Value](#)

## Adding a New Color

To add a new color:

1. Type a name for the color in the Color text box.
2. Click the New button, which displays the Color Picker dialog.
3. Use the Color Picker dialog to define a new color, then click the OK button.

The new color that you defined will appear in the scrolling list and palette controls. This color will now be available for formatting the pen and brush colors of objects.

 **See Also**

[Using the Color Picker](#)

## Modifying an Existing Color

To modify an existing color:

1. Select the color that you want to modify in the scrolling list or in the palette control.
2. Click the Modify button, which displays the Color Picker dialog.
3. Use the Color Picker dialog to redefine the color, then click the OK button.

Notice that the color is now changed in the scrolling list and palette controls. The modified color will now be available for formatting the pen and brush colors of objects.

 **See Also**

## Using the Color Picker



## Deleting a Color

To delete a color that you no longer need to use in a drawing:

1. Select the color in the scrolling list or in the palette control.
2. Click the Delete button.



Objects that have already been colored in a color that you delete will still have the deleted color.

## **Using the Color Picker**

The color picker lets you specify colors that you can add to your drawing's color palette, or use to replace an existing color in the palette. The dialog contains a color wheel, which displays a spectrum of available colors, and a scrollbar with which you can control the overall brightness of the colors on the wheel. It also contains controls that let you define a color by directly entering its RGB (Red, Green, Blue) values, or by specifying its Hue, Saturation, and Value, which correspond to positions on the color wheel and scrollbar.

A lower color display box to the left of the color wheel gives you visual feedback about the color you have chosen. If you are modifying an existing color, the original color will be displayed in the upper color display box.

## **Picking Colors with the Mouse**

To pick a color with the mouse:

1. Set the overall brightness of the color wheel using the scroll bar to the right of the wheel.
2. Click at the position on the wheel that is the color and shade you want.

An example of the color you have picked will appear in the color display box to the left of the color wheel.

## Setting RGB Values

Screen colors can be defined by a combination of Red, Green, and Blue (RGB) values. Each of these values can be set to an integer from 0 to 255. For example, an RGB setting of (0,0,0) is black; an RGB setting of (255,255,255) is white. A setting of (255,0,0) is “pure” red, and a setting of (0,255,255) is Cyan. To define a color by its RGB values, enter the values in the R, G, and B controls on the dialog. Notice that the color in the lower color display box changes to correspond to the values you have entered, and the position of the color picker on the color wheel changes as well.

## **Setting Hue, Saturation, and Value**

Hue and Saturation settings correspond to physical positions on the color wheel. Hue represents an angular position on the wheel measure in degrees (corresponding to the color spectrum), and can have a value from 0 to 360. Saturation represents a radial position on the wheel (corresponding to the amount of "grayness" in the color), and can have values from 0 to 255. Value controls the overall brightness on the color wheel, so changing the value in this control is the same as using the scroll bar to the right of the color wheel.

## Drawing Setup Dialog

**Menu:** Tools|Drawing Setup

The Drawing Setup dialog includes five property sheets containing settings that are saved in a drawing or template file.

### [Grid Property Sheet](#)

Choose the grid type and other grid options.

### [Symbol Libraries Property Sheet](#)

View and maintain symbol libraries in current drawing or template.

### [Units and Scale Property Sheet](#)

Set the unit of measure, the drawing scale, and related options.

### [Angle Property Sheet](#)

Choose direction, units, and base angle for angular measurements.

### [Layers Property Sheet](#)

Create new layers, modify existing layers, and delete layers.

## Grid Property Sheet

**Menu:** Tools|Grid

The grid is a matrix, displayed as a pattern of dots, crosses, or lines, that divides your drawing geometrically into Cartesian or polar coordinate locations.

The Grid property sheet allows you to customize the grid. The following topics describe the controls of the Grid property sheet.

[Spacing](#)

[Divisions](#)

[Frequency](#)

[Colors](#)

[Show Grid Checkbox](#)



## Grid Spacing

The *spacing* of the grid determines how far apart the grids or lines of the grid are spaced from one another. For example, if spacing is set to one foot, points on the grid will be spaced a foot apart. Grid spacing is set in either World or Paper units, whichever is currently being used.



Grid spacing applies to every grid point, regardless of whether it is a minor or major grid point, and without regard to the frequency of the grid.

## Grid Frequency

The Frequency settings lets you alternate visible grid lines with invisible grid lines. For example, if you set the frequency 2, then every other grid line will be displayed. If you set frequency to 4, then every fourth grid line will be displayed. You will still be able to snap to the invisible grid lines.

 **Warning**

Don't confuse this with whether the grid itself is visible. If you make the grid invisible using `View|Grid`, then the whole grid will be invisible.

## **Divisions**

Divisions determines the number of minor grid divisions between major grid divisions. You can set the number of divisions between each major grid point using the spin button control near the option button.

## Color

Click the buttons in the Color group to set choose a new color for the major and minor grid. Click the Major or Minor button, then choose a color using the Color Picker.

 [See Also](#)

[Using the Color Picker](#)

## **Show Grid Checkbox**

The Grid property sheet also contains these controls:

*Hide grid when finer than...* You can type a value in this box to control how fine a grid can be drawn on the screen. When the grid becomes finer than this value, it will not be drawn. Note that this value applies to distances on your computer screen, which are not related to World or Paper units.

*Show grid.* Check this box to display the grid. This is the same as choosing View|Grid from the menu.



## Symbol Libraries Property Sheet

**Menu:** Tools|Symbol Libraries

TotalCAD 1.0 symbol libraries are actually just Windows folders that contain TotalCAD drawings. This dialog associates descriptive names with folders that you want to use as symbol libraries. Once you create names for the symbol libraries in your drawing, you can easily switch between the named symbol libraries using the Symbol Library palette. These libraries will be saved with your drawing, or in a template if you save your drawing as a template.

[Adding a New Symbol Library](#)

[Deleting a Symbol Library Name](#)

[Changing the Symbol Library Association](#)

 **See Also**

[View Symbol Library](#)

## **Adding a New Symbol Library**

The Add button lets you associate a descriptive name with a symbol library.

1. Type a name for the library in the Name text box.
2. Click the Add button. TotalCAD will display the standard Windows Open dialog. Use the dialog to choose the folder containing the library, then click OK to return to the Drawing Setup dialog.

When you select a library, its name, as well as its file name and path, appear in the property sheet's list box.

## Deleting a Symbol Library Name

To delete a symbol library's descriptive name from the list, select its name in the list box, then click the Delete button.



Deleting a library name removes the library name from your drawing, but has no effect on the library's folder or the drawing file contained in the folder.

## **Changing the Symbol Library Association**

To associate a descriptive name with a different symbol library:

1. Select the symbol library name that you want to change by clicking on it in the Name list box.
2. Click the Modify button. TotalCAD will display the standard Open dialog. Use the dialog to navigate to directory with which to associate the symbol library name, then click OK to return to the Drawing Options dialog.

## Units and Scale Property Sheet

**Menu:** Tools|Units and Scale

The Units and Size property sheet contains options for setting the drawing scale and for controlling the display of linear units.

The following sections describe the controls available on this property sheet.

[Measuring System](#)

[Scale](#)

[Units in Work Space](#)

[Use Paper Space Coordinates](#)



**Tip**

TotalCAD's built-in templates and wizards can help you set up your drawing space when you start a new drawing. This property sheet is for cases when you want to set up your units and scale manually.



## **Measuring System**

The Measuring System control group contains two general settings, English and Metric. Choosing either of these settings will select different sets of options to choose from in the other controls of this property sheet.

## Scale

*Scale* is the ratio of measurements on the page (Paper units) to measurements in the object being modelled (World units). TotalCAD lets you set the drawing scale using one of three standard systems: Architectural, Metric, and Absolute. If Metric is selected in the Measuring System group, then Absolute will be the only choice available. The following table shows some sample drawing scales expressed in each of these systems:

<b>System</b>	<b>Example</b>
Architectural	$3/16'' = 1'-0''$
Engineering	$1'' = 200'$
Absolute	1:500

Use the Type drop down box to choose a system, then use the Scale box to choose a scale, or type in a custom scale.

## Units in Work Space

Use the controls in this group to specify a unit of measure for your drawing:

*World Units.* Click in this drop-down list to choose a unit of measure. The choices that appear in the list will depend on whether English or Metric is selected in the Measuring System group.

*Show All Units.* Check this box if you want the World Units drop-down list to display all available units, regardless of whether English or Metric is selected in the Measuring System group.

*Format.* Click in this box to choose a format for the display of numbers. The choices available in this box will be limited to Decimal and Scientific if Metric is chosen in the Measuring System group. The following table shows examples of the available number formats:

<b>Format</b>	<b>Example</b>
Decimal	3.25
Scientific	1E+01 mm
Fractional	3 3/16
Architectural (Feet in Fractional)	1'-5 11/16"
Engineering (Feet in Decimal)	1"= 200'

*Precision.* Use this spin control to specify the precision of decimal numbers (the number of places to the right of the decimal point that TotalCAD displays). You can either click on the arrows, or type a number directly into the box.

## Use Paper Space Coordinates

Check the Use Paper Space Coordinates box if you want the Ruler, Grid, and the coordinate system to be in Paper, rather than World, units. This is useful if you want to precisely arrange your drawing on a page, or if you want to precisely place or scale an element of your drawing, such as text, in Paper space.



Checking and unchecking this box is exactly the same as clicking the Paper/World button in the upper-left corner of the Ruler.

## Angle Property Sheet

**Menu:** Tools|Angle

The Angle property sheet gives you precise control over the measurement and display of angles, as well as control over the Ortho angle. You can change the system and direction of measurement, the base angle, and the precision of measurement.

[Angular System](#)

[Direction](#)

[Ortho Angle](#)

## **Angular System**

Use the list box to choose a system appropriate to your drawing. You can choose to display angular measurements in degrees, degrees minutes and seconds, grads, radians, or the surveyor system. In addition, you have the following two options:

*Base Angle.* Type a value into this text box to set the position of the base angle. Setting the base angle to 90 degrees, for example, would cause TotalCAD to start measuring angles from the top quadrant point of a circle or arc, rather than from the right quadrant point.

*Precision.* Use this control to set the precision of the display of decimal numbers. If you set Precision to 2, for example, measurements will be displayed to an accuracy of two digits to the right of the decimal point.

## **Direction**

Use these option buttons to choose whether to measure angular distances in a clockwise or counterclockwise direction.



## **Ortho Angle**

Use these controls to customize the the Ortho angle. Ortho constrains the angles of lines during drawing and editing operations.

*Base Angle.* Type a value into this text box to set the position of the ortho base angle, which is the angle from which the ortho angle is measured. Changing this angle can be helpful if you want the ortho angle to be measured in relation to an object that is not oriented orthogonally relative to the drawing space.

*Step Angle.* The step angle determines the angle to which Ortho constrains lines. If you set the step angle to 15, for example, the line will be constrained to angles 15 degrees apart. TotalCAD's default sets the step angle to 90 degrees.

## Layers Property Sheet

**Menu:** Tools|Layers

The Layers property sheet allows you to create new layers, set the current drawing layer, set the properties of layers, and delete layers.

Layers are virtual levels in a drawing, analogous to acetate sheets (overlays) in traditional drafting. Layers give you the advantage of sorting objects in your drawing by the type of object they are modelling, by order of creation, or by any principal that suits the way you work. You can then place each category of object on a separate layer of the drawing, so that you can work on one layer of objects at a time.

Layers can be visible or not visible, editable or not editable. If you want to view only a subset of the objects in your drawing, set the layers you want to see as visible, and set all other layers to invisible. Setting layers to non-editable makes it possible to protect objects on those layers from unintended change while someone is working on another layer of the drawing.

Layers can have their own default color and line styles. If the color and line-style properties of objects on a layer are set to By Layer, they will have the color and line style assigned to the layer.

[Creating a New Layer](#)

[Setting the Current Drawing Layer](#)

[Setting Layer Properties](#)

[Deleting a Layer](#)

 **Note**

 **See Also**

If you create a group composed of objects on different layers of the drawing, the group will reside on the current layer (the layer that is current at the time the group is created). If you subsequently explode the group, each of the constituent objects will be on its original layer.

Format Property Sheet

## **Creating a New Layer**

To create a new layer, type a name for the layer in the Name text box, then click the New button. The new layer name will appear in the Layers dialog list box.

## **Setting the Current Drawing Layer**

The current drawing layer is the layer on which any new objects that you create or insert will reside. To set the current drawing layer, select the name of the layer from the Layers dialog list box, then click the OK.

## Setting Layer Properties

You can set the properties of a layer using the controls in the Properties area of the Layers dialog. Any changes you make apply to the layer that is currently selected in the Layers dialog list box.

*Visible.* Check this box to make the layer visible and uncheck it to make the layer invisible.

*Editable.* If this box is checked you can draw, insert, and edit objects on the layer. If it is not checked, then you can view the layer, but you can't change it.

*Color.* Use this pull-down box to select a color for the layer. If objects on the layer have their pen or brush color property set to By Layer, then the pen or brush of the objects will have the color specified here.

*Line Style.* Use this pull-down box to select a line style for the layer. If objects on the layer have their line style property set to By Layer, then the objects will have the line style specified here.

*Order.* Use this text box to assign an *order number* to the layer selected in the Layers dialog list box. Order numbers determine the order in which layers are drawn (which corresponds to the order in the Layers list box). Objects drawn on layers with higher order numbers will overlap objects drawn on layers with lower order numbers. The order numbers of layers are all set to 0 initially. Layers that have the same order number are sorted in alphanumeric order.

After you have made changes to the properties of the selected layer, click the Modify button to apply the changes.

 **Note**

 **See Also**

The Bring to Front and Send to Back commands (on the Format menu) affect the redraw order of objects within a single layer, or across layers if the layers have the same order number. An object on a layer with a higher order number will always overlap an object at the front of a layer with a lower order number.



[Bring to Front](#), [Format Property Sheet](#), [Send to Back](#)

## **Deleting a Layer**

To delete a layer, you must first clear any objects on the layer. Once the layer is cleared of objects, you can delete it in the Layers dialog by selecting its name in the list box and clicking the Delete button.

## Windows Menu

### Tile

Arrange windows so that they are equal-size, side-by-side, with no overlap.

### Cascade

Arrange windows so that they overlap with title bars visible.

### Arrange Icons

Line up minimized windows along bottom at bottom

### Close All

Close all open windows.

### Changing the Active Window

Lower part of menu lists open drawings; choosing a drawing makes it the active window.

## Tile

**Menu:** Window|Tile

Use the Tile command to arrange all the open drawing windows into a mosaic. The windows are resized so they all the same size and fit entirely within the application window.



**Tip**

This is a convenient arrangement when you want to compare the contents of two documents or drag and drop objects between documents.

## **Cascade**

**Menu:** Window|Cascade

Use the Cascade command to resize and arrange all the drawing windows so they overlap with their title bars showing. Because you can see a portion of each window, you can easily activate any window by clicking it. Only the topmost window is completely visible.

## Arrange Icons

**Menu:** Window|Arrange Icons

When you minimize a drawing window, Windows shrinks it to an icon. When there is not an open drawing window, you can see any minimized drawings. Use Arrange Icons to line up these iconized windows along the bottom of the application window.



The general shape and appearance of the icon will vary, depending on whether you are using Windows 95 or Windows 3.1.



## Close All

**Menu:** Window|Close All

Use Close All to close every open drawing window.

 **Note**

TotalCAD will warn you if an open drawing has unsaved changes. Click the Yes button in the warning dialog to save the changes; click the No button to discard the changes. Click Cancel return to TotalCAD without closing the drawing windows.

## **Changing the Active Window**

TotalCAD lists all of the open drawings in the lower half of the Windows menu. The currently active drawing will have a check mark. When you choose a drawing from the menu, TotalCAD makes it the active window. This is the same as clicking in window.

The zoom percentage box shows how far in or out of your drawing the current view is zoomed. To zoom to 100%, choose View|Zoom|Printed Size.



