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Getting Started

Congratulations! You are now the owner of TurboCAD v4 or TurboCAD v4 Professional, the world's leading value CAD (Computer Aided Design) software. TurboCAD offers superb ease of use, enhanced by Microsoft Office compatibility, extensive file format capability and all the tools you need to create professional CAD drawings.

Subtopics

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System Requirements

TurboCAD v4 Professional and TurboCAD v4 minimum requirements:
- 486 DX2/66 or better CPU
- 16 MB RAM (32 MB recommended)
- 30 MB of free hard disk space
- VGA graphics card or better
- CD-ROM drive
- Windows 95 or Windows NT 4
- Mouse (Intellimouse supported)
- Internet connection recommended

Note: TurboCAD v4 is fully Windows 95 compliant, and is Microsoft Office compatible.

Registration

You can register your copy of TurboCAD over the Internet or by phoning the toll free number. Please click on the Registration button on the opening screen for more details. You are obliged, and indeed compelled, to register your copy TurboCAD. Advantages of registering included access to extensive on-line support databases, and program updates.

Installing TurboCAD

These installation instructions apply to TurboCAD v4 Professional and TurbCAD v4 only. TurboCAD 3D must be installed separately.

Subtopics

Installing TurboCAD in Windows 95
IMSI Program Folder

Installing TurboCAD in Windows 95

1. Insert the CD-ROM into your CD-ROM drive.
2. Click the Start button, then click Run from the popup menu.
3. Type d:\setup in the dialog Open field, substituting for d: the drive letter of your CD-ROM drive.
4. Follow the instructions on screen to complete the installation.

IMSI Program Folder

After you have installed TurboCAD, the Setup program creates a program folder. If you chose the default settings, TurboCAD is installed in the C:\IMSI\TCW40 folder. This folder contains several subfolders that in turn contain TurboCAD's program files and related files such as templates, sample drawings, and symbols.

The Setup program also creates a program group containing the TurboCAD application icon, as well as shortcuts to the Help, Readme, and an Uninstall program.

In Windows 95, the program group is accessed through the Start menu.

Before you start the program, please read the Readme file, which contains the latest information on TurboCAD.

If at any time you want to Uninstall TurboCAD (remove TurboCAD and all related program files from your system), select the Uninstall icon.

Getting the Answers You Need

TurboCAD comes with a combination of print, online help, and Web access, that can help you learn to use the program and provide answers to all of your questions as you delve deeper into TurboCAD's many features. If you are new to CAD programs, we recommend that you start using the documentation as follows:

1. Read Quick Tour (Chapter 2 of the manual), and work through the "mini-tutorial" contained in that chapter.
2. Read the TurboCAD Guide to Learning CAD included with the package. This documentation contains extensive examples using TurboCAD and covers the most important concepts of CAD.

After you are familiar with TurboCAD's basic features, read the manual for more in-depth information, and use the online help system for fast assistance with TurboCAD features while you are drawing.

Subtopics

Running the Quick Tour

Using the Help System

Context-Sensitive Help

Running the Quick Tour

TurboCAD's electronic demo provides self running illustrations that demonstrate   various features in TurboCAD. To start the Quick Tour demonstration, make sure that the TurboCAD CD is in the drive and the click the button on the opening screen. Follow the instructions on screen.

Using the Help System

TurboCAD's provides several different methods for getting help. Access a complete reference to TurboCAD's features, by choosing Help|TurboCAD Help Topics. To see a complete list of keyboard shortcuts, choose Help|Keyboard. To access the Online Knowledge Base over the Internet, select Help| Online Technical Support.

Click here for information on 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 TurboCAD interface. To view the help topic on a tool or menu item, just place the mouse cursor over the tool, or highlight the menu item, then press <F1>.

Or, you can click on the Help tool, then click on the tool you want information about.

Symbols and Conventions

This documentation uses the following symbols and conventions:

▪ The names of keys are enclosed in angle brackets, and a plus (+) sign is used in key combinations. For example, <Ctrl>+<Y> means to press the Y key while holding down the Control key.

▪ Menu items are referred to by a *menu path,* in which the steps through the menu system are separated by a pipe (|). For example, File|Open refers to the Open menu item, located on the File menu. If there is a submenu off of the menu item, then there will be one more pipe. For example, Modes|Snaps|Vertex means the Vertex command in the Snaps submenu of the Modes menu.

▪ Windows dialog boxes can contain multiple sheets accessible by clicking on tabs. This documentation will refer to those sheets as *property sheets.* When referring to a control located in a dialog, the documentation will also refer to the specific property sheet where the control is located.

▪ User entry is shown in boldface type. For example, you might see instructions like this: "Type **1.24 ft** into the Length field on the edit bar."

Subtopics

Commands and Tools

Objects and Entities

Commands and Tools

The terms *command* and *tool* are used in the documentation to describe distinct types of TurboCAD functions.

Command. Commands are TurboCAD functions that take some action and complete the action. Select All would be an example of a command, because it immediately selects all of the objects on the screen and returns TurboCAD to its previous state, expecting no further action on the part of the user.

Tool. When you select a tool, you access a special set of functions that are available as long as the tool is *active*. Some tools remain active until you explicitly switch to another tool. For example, you can activate the Single Line tool and continue to draw lines until you switch to another tool. Other tools, such as Zoom Window, are deactivated as soon as you finish using them.

Tip: Think of a command like this: You tell TurboCAD what to do, and TurboCAD does it. Think of a tool like this: You ask TurboCAD for the means to perform an action, TurboCAD provides that means, then you perform the action.

Note: Occasionally the term "tool" can also be used simply to mean a tool button on the toolbar, or any function accessible using a tool button.

Objects and Entities

The term *object* is a general term used in this documentation to mean anything that can be displayed in a TurboCAD drawing. The term *entity* is used in a narrower sense, meaning a drawing object created with TurboCAD drawing tools. Entities include single lines, multilines, arcs, circles, polygons, and so forth, as well as construction lines and circles. Entities can be combined into groups and blocks, which are also referred to as objects. Other types of TurboCAD objects include bitmaps, metafiles, OLE objects, and symbols.

Contacting Technical Support

There are several options for contacting IMSI technical support.

Subtopics

[Electronic Support](#)

[Standard 60 day Telephone Support](#)

[Premium Support](#)

[Other Support Options](#)

Electronic Support

Free electronic support is available 24 hours a day on the IMSI Web Site. Several support options are available including, the TurboCAD Support Forum, Frequently Asked Questions, and the TurboCAD Knowledge Base.

Web Site	www.turbocad.com
E-mail	support@imsisoft.com
TurboCAD	cadsupport@imsisoft.com
IMSI Web Site	www.imsisoft.com
Fax	415-257-3565

Click on the support icon in the top frame of the web site to access the various support options.

Link via the Web to a dynamic help file available via the TurboCAD Inference Knowledge Base accessible from the Support Web page. Go to the HELP drop down menu and click on Online Technical Support.

Standard 60 day Telephone Support

No charge support, via a toll call, is available from IMSI for 60 days beginning with your first call to a support technician. 415-257-3000 x 3.

In the United States, hours are 8 a.m. - 5p.m. Pacific Time, Monday through Friday, excluding holidays.

Premium Support

For an additional amount you can receive unlimited phone support and two-years worth of product upgrades. The insert placed in the box contains additional information on support contracts.

Other Support Options

IMSI also offers support via a 900 number after the initial 60 Day free-tech support. Calls are $2.00 a minute and the first minute is free. (US Only) See insert for more details.

Upgrade and Support options can be purchased on an annual basis.

Please call for pricing.	800-833-8082
Product Information or replacement disks:	800-833-8082
Sales/Customer Service:	800-833-8082
Technical Support:	415-257-3000x3
Online Store:	www.imsisoft.com

Quick Tour

The following sections provide a brief introduction to the TurboCAD environment. For further help with getting started, read the Quick Tour in the manual or run the online tutorial.

Subtopics

Starting TurboCAD
New TurboCAD Drawing
TurboCAD Desktop

Starting TurboCAD

When you install TurboCAD in Windows 95 or Windows NT 4.0, TurboCAD is installed in the IMSI TurboCAD folder and can be accessed via the Windows Start|Programs menu. To start TurboCAD:

1. Click the Start button, then choose Programs to open the Programs submenu.
2. Click IMSI TurboCAD to open the IMSI TurboCAD submenu.
3. Click TurboCAD v4 Professional, or TurboCAD v4, depending on your purchase.

Subtopics

Start Page Options
Start Page Control Options

Start Page Options

The TurboCAD Start Page will appear. Several options are available.

New. Click the New button to start a new TurboCAD drawing.

Open. Click the Open button to work with an existing TurboCAD drawing.

Open Recently Used. Once you start using TurboCAD, the most recently used drawings will appear in the black area along the left side of the Start Page. Click the icon for any available drawing to open it. The drawings that appear will be the same ones listed in the File menu.

Update. Click the Update button to download any updated version of TurboCAD via the Internet. To use this option, you must have previously registered your purchase of TurboCAD. See Registration.

Quick Tour. Click the Quick Tour button to run an introductory TurboCAD tutorial.

Registration. Click the Registration button to automatically register your copy of TurboCAD via the Internet.

Go To Home Page. Click the IMSI logo to connect to the TurboCAD home page on the World Wide Web.

Note: To use the Update, Registration or IMSI Web Page options in the Start Page, you must have an account with an Internet Service Provider (ISP) and Internet access software installed in your Windows operating system.

Start Page Control Options

The drawing board icon in the upper left corner of the Start Page is the button for the Control Menu Box. The standard Control Menu Box items are available (Restore, Move, Minimize, and Close). The following additional commands are available:

Options. Two options are available. You may shut off the clicking sound that identifies each button in the Start Page, and you may disable the Start Page. If you disable it, TurboCAD will bypass the default opening screen, taking you straight to the "Tip of the Day" or "New" dialog.

About TurboCAD. This item displays copyright and version information, the same as the menu item Help| About TurboCAD.

Tip: To restore the Start Page, use the Run command in the Windows start menu to run FRONTOPT.EXE, found in the IMSI/TCW40/PROGRAM folder.

New TurboCAD Drawing

When you select the New Option from the Opening dialog you will be presented with a choice of different ways to create a new drawing.

1. Click the New button on the opening screen.

   TurboCAD will then display the New dialog asking what method you wish to use to create a new drawing.

2. Click the New from Template option to access the list of available templates.

TurboCAD will then display the New from Template dialog. This is where you choose a standard template to use as the basis of your drawing.

3. Select Normal, then click OK to close the dialog.

TurboCAD Desktop

Once you have chosen the Normal template you will see the TurboCAD desktop with a new, blank drawing.

Here are some brief descriptions of the components on your screen:

Subtopics

Menu Bar
Toolbars
Symbol Library Palette
Scroll bars
Paper
Ruler
World/Paper Button
Edit Bar
Coordinate Fields
Status Bar Message

Menu Bar

Click on the menu names on the menu bar to access TurboCAD's commands.

Toolbars

Toolbars provide a convenient alternative to the menu system. The default screen displays four toolbars: the Standard and Properties toolbars along the top of the screen, the View toolbar on the left, and the Snaps toolbar on the right. Clicking on any of these buttons (tools) on the toolbars will activate a TurboCAD command.

Note: The exact combination of toolbars will depend on the screen resolution in use when you start TurboCAD for the first time, and may differ slightly from the desktop described here.

Tip: Notice that if you place the mouse cursor over a tool and leave it there briefly, a small yellow rectangle called a tooltip will appear beneath the tool, displaying its name.

Symbol Library Palette

The large open area along the left side of the screen is used to display symbols, predrawn objects available for repeated use. Three other palettes (Blocks, Selection Info, and Measurement Info) may also be displayed on the desktop.

Scroll bars

Your drawing can be larger (much larger!) than what you see on the screen. 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.

Paper

The Paper, the large white rectangle in the middle of the screen, shows you how your drawing will be laid out on the page when printed. You don't have to draw on the Paper; you can draw anywhere in the drawing space and later fit the drawing to a sheet of paper. You can also turn off the paper sheet shown. The pattern of horizontal and vertical lines on the Paper is the grid, which marks exact locations in the drawing.

Ruler

The ruler gives you feedback about the size and position of objects in your drawing. In the Normal template, the unit of measure is inches. The units of measure may be set to inches, feet, or to common metric units. For example, you may be drawing a picture of a table that is five feet long, while the representation of 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

World/Paper Button

This button, in the upper-left corner of the ruler, lets you change the ruler units between World Units and Paper Units.

Edit Bar

Use the Edit Bar to draw and edit using numeric values such as length and angle. For example, you could tell TurboCAD to draw a line exactly two feet long, at a 45 degree angle.

Coordinate Fields

Use the Coordinate Fields at the bottom of the screen to specify a point in the drawing space by its exact coordinates.

Status Bar Message

This message area provides a brief message describing the currently active tool, or instructions about how to perform the next step in a drawing or editing process.

File Management and Printing

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

TurboCAD'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.

Subtopics

[Supported File Types](#)
[Printing](#)
[File Menu Commands](#)

Supported File Types

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

TurboCAD also supports TurboCAD 3D .MDL files. You can save a drawing as an .MDL file in TurboCAD, open the file in TurboCAD 3D, and then use the features of TurboCAD 3D to add a third dimension to your drawing. Or you can read an .MDL file in TurboCAD, if you want to edit a three-dimensional drawing using the more sophisticated drawing and editing tools available in TurboCAD.

To open files created in TurboCAD 2.0 or 3.0 for Windows, you can use the standard .TCW file type. Keep in mind, however, that once you save .TCW files in TurboCAD, they can no longer be read by an older version of TurboCAD. You can, however, save in TurboCAD 3.0 format by clicking the Setup button in the Save dialog. You can also open TurboCAD for DOS files that have the .TCD extension.

TurboCAD can open TurboCAD 2.0 symbol library (.SLW) files. See the Appendix at the end of this documentation for instructions on converting .SLW files.

The following table describes the file types supported by both TurboCAD v4 and Professional:

File Type	Extension	Comment
TurboCAD v4	.TCW	Default file type for TurboCAD v4.
TurboCAD 3D	.MDL	File type of TurboCAD 3D; TurboCAD v4 can open and save files of this type.
TurboCAD Template	.TCT	Use predefined or custom templates as basis for new drawings.
TurboCAD 3.0	.TCW	You can open and save files of this type.
TurboCAD 2.0	.TCW	You can open, but not save, files as this type.
TurboCAD DOS	.TCD	You can open, but not save, files as this type.
TurboCAD Symbols	.SLW	TurboCAD v4 and TurboCAD v2 symbol libraries.
AutoCAD DWG	.DWG	AutoCAD drawing files (though Release 13).
AutoCAD DXF	.DXF	Data eXchange Format (though DXF R13). The most common CAD file exchange type.
TurboCAD ASCII	.TCX	TurboCAD drawing saved as ASCII text.
Windows Metafile	.WMF	Windows vector image format.
Drawing Web Format	.DWF	Format for displaying drawings on the World Wide Web.
Floor Plan Plus	.FP3	IMSI FloorPlan Plus 3.0 format.
HPGL	.PLT	A commonly used CAD file type (Hewlett Packard Graphic Language).

The following table describes the additional file types supported by TurboCAD v4 Professional:

File Type	Extension	Comment
AutoCAD DWG	.DWG	AutoCAD (though Release 13) drawing files with

		embedded 3D ACIS technology.
PostScript	.EPS	Popular vector format for publishing graphics.
ACIS 3D	.SAT	Spatial Technology Inc., 3D Solid Modeling file format.

Note: TurboCAD v4 can open and save TurboCAD 3D .MDL files. The appearance of the drawing files in the programs will vary slightly, because of the programs' different features and capabilities.

Warning: If you open a file created in a previous version of TurboCAD, then save the file in TurboCAD v4, the file type will be updated to TurboCAD v4. You will not be able to re-open the files in the earlier version of TurboCAD, unless you specifically check the TCW30 option in the Setup of the Save As dialog.

See Also: Open, Save As Dialog

Printing

TurboCAD 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. TurboCAD also provides a full range of advanced printing options for experienced users.

Note: Under Windows, plotters are treated as printers.

Subtopics

[Printer Paper and the Drawing Sheet](#)
[Simple Printing](#)
[Tiled Printing](#)

Printer Paper and the Drawing Sheet

Before dealing with TurboCAD'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 adjustable 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

TurboCAD'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. 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. Also assume 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. Choose a printer in the Name pull-down field, then click the Properties button to bring up the property sheets for your printer. Choose your paper size and paper orientation, the 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.

Note: The exact appearance of the print dialog will depend on the printer you use.

Tiled Printing

TurboCAD 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.

Tip: When printing on multiple tiled sheets, it is usually best to 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.

See Also: Page Setup Dialog, Print Dialog

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. TurboCAD 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 Drawing 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 Options|Units and Scale 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.

Note: 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.

Subtopics

Tiled Printing by Setting Rows and Columns

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. TurboCAD will automatically adjust the dimensions of the drawing sheet to accommodate changes in the number of rows and columns.

File Menu Commands

The following table summarizes the commands and dialogs accessible from the File menu:

Command	Function
New	Start a new TurboCAD drawing.
Open	Open a drawing in one of several file formats.
Trace	Analyzes a bit-map image to create a vector drawing.
Close	Close the current drawing.
Save	Save the current drawing to disk.
Save As	Save the current drawing under a new name.
Save All	Save all open drawings.
Page Setup Wizard	Step-by-step guide to setting page size and drawing scale.
Summary Info Dialog	View or edit document information about the current drawing.
Page Setup	Choose options for printing your drawing.
Print	Print the current drawing.
Send	Send current drawing over a network using Microsoft Mail.
Recent File List	Re-open one of the four most recently used files.
Exit	Exit TurboCAD

New

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

Use the New command to start a new TurboCAD drawing.

Subtopics

New from Page Setup Wizard
New from Template Dialog
New from Scanned Image
New from Scratch
Cancel

New from Page Setup Wizard

The Page Setup provides an intelligent method of setting up a new drawing. It will make intelligent choices for you in regards to scale, hatch scale, text size, grid size and much more. The behavior is very similar to the options found in Options|Units and Scale except that each question is presented in a logical sequence.

See Also: Page Setup Wizard, Units and Scale

New from Template Dialog

Templates are a special type of drawing file that provide 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 block libraries.

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

The Create from Template dialog contains these additional controls:

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). The default template is highlighted when this dialog opens.

Large Icons, List, Details. Click these boxes if you want TurboCAD to display the templates in the associated format.

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

Tips: You can change the default template folder using the File Locations property sheet of the Program Setup dialog (Options|File Locations).

See Also: Program Setup Dialog, Save As Dialog

New from Scanned Image

This excellent new feature of TurboCAD v4 allows you to scan a drawing from paper and then trace it into TurboCAD vector format.

See Also: Trace, and the TurboCAD Guide to Learning CAD.

New from Scratch

This option allows you to create a new blank drawing, with the default settings stored in the Tcw40.INI file.

Cancel

This option will simply close the dialog returning you to TurboCAD.

Open

The Open command displays the Open dialog, which allows you to open an existing TurboCAD drawing, or a CAD drawing in any of several different standard formats.

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 the standard Windows controls to navigate to the folder 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 TurboCAD drawing files (.TCW) will be displayed.
3. Select the file that you want to open, then press <Enter>.

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. Only TurboCAD (.TCW) files can have previews.

Setup. Some of the file formats supported by TurboCAD have import options. Click the Setup button to access a dialog of options for these file types:
- AutoCAD DWG
- AutoCAD DXF
- ACIS SAT
- TurboCAD 3D MDL
- Windows WMF
- IMSI Floor Plan 3 FP3
- Hewlett-Packard Graphics Language (HP-GL) .PLT

The specific import options vary with the file type being imported. Specific settings for the various setup dialogs depend on how the file was originally created in its originating program.

See Also: Save As Dialog, Supported File Types

Trace

Trace analyzes a raster (bit-map) image, such as created by a scanner or a paint program, and converts it into TurboCAD's vector format.

Trace offers many advantages over importing a bit-mapped image using the Insert|Picture command. The image can be reversed, specific portions rotated, and fills separated from outlines. The imported objects can be scaled, edited, and in all other ways considered part of the TurboCAD drawing.

To trace a raster image, select Trace from the File menu. Press Browse, and select the file using a standard Windows dialog. Once the image is selected, two command options are available to control how the raster-to-vector conversion will occur.

Invert. The selected file will appear in the display area. If the file appears as white lines on a black background, click the Invert setting.

Settings. Raster-to-vector conversion process is an inexact science. The program starts with an image composed of only dots, and must generate from it a drawing composed of geometric objects. Most images will still need to be modified slightly after the conversion. Trace contains various mathematical algorithms; the Settings commands will select the right "method of attack" depending on the intended use for the converted image.

The options, found in the Settings list box, are:
- Architecture, for floor plans and other construction drawings.
- Cadastral, for survey documents.
- Contour, for contour maps.
- Exploded View, for technical illustrations.
- Logotype, for scanned logos and text.
- Mechanical, for engineering drawings.
- Outline, to generate an outline from an image.

After selecting the intended output method, click Trace to start the tracing process.

Subtopics

Trace Graphic Window Menu Options and Tools

Trace Graphic Window Menu Options and Tools

When you click on Trace, the Trace Graphic Window appears. Trace analyzes the drawing, and finishes by displaying a vector-lines overlay on top of the original bit-mapped image. You may choose to edit the drawing to delete most or all raster entities, or save as-is to preserve all raster entities as part of the drawing.

The Graphic Window has one menu item, File. Two options are available, Settings and Exit. Settings allows you to choose between millimeters and inches as the measurement scale. Exit closes the Trace program and returns operation to TurboCAD.

When the bit-map image has been traced, you may manipulate it using the tools that appear on the toolbar just beneath the File menu. The Select tool is always active in Trace, except when the program is responding to a tool selection.

Note: There is no local menu available when using the Select tool in Trace.

Subtopics

Save Exit (Trace Graphic Window)

Undo (Trace Graphic Window)

Delete (Trace Graphic Window)

Properties (Trace Graphic Window)

Show Vectors (Trace Graphic Window)

Show Raster (Trace Graphic Window)

Zoom In (Trace Graphic Window)

Zoom Out (Trace Graphic Window)

Zoom All (Trace Graphic Window)

Orthogonal Adjustment

Horizontal Adjustment

Vertical Adjustment

Vertical/Horizontal Adjustment

Text (Trace Graphic Window)

Save Exit (Trace Graphic Window)

The Save Exit tool is used to both save the newly created vector drawing, and to exit from the program. There is no separate option to save and continue using Trace. This tool works in Trace in the same manner as the File|Exit command works in TurboCAD.

If TurboCAD is open, the drawing will automatically appear on screen, and the file will be saved in the default Trace folder as RAS000##.TCW. You will have to use the Save procedure in TurboCAD to give the file a more meaningful name.

Undo (Trace Graphic Window)

The Undo tool will reverse the last operation.

Delete (Trace Graphic Window)

After Trace creates a vector drawing from your raster (bit-mapped, scanned) image, you will probably need to clean up the drawing. Select entities you wish to delete, using the <Shift> key or window selection to select more than one entity. Click the Delete key to delete all selected entities.

Properties (Trace Graphic Window)

A click on the Properties tool will access a dialog containing information on both the raster and vector entities in the drawing. The information appears in a standard windows expanding list; click the (+) and (-) items to expand or collapse the list.

Show Vectors (Trace Graphic Window)

As you clean up a traced vector image, you can choose to display only the vector entities, only the raster entities, or both. When the Show Vectors tool is active, all vector entities will be visible.

Tip: Turn Show Vectors on, and Show Raster off, when you want to use Trace's adjustment tools to clean up horizontal, vertical and orthogonal alignment in the drawing.

Show Raster (Trace Graphic Window)

When the Show Raster tool is on, all the original bit-mapped (raster) entities in the drawing will be visible. Both the Show Raster and Show Vector tools may be active at the same time.

Tip: Depending on the style of drawing, and its intended purpose, you may want to include some or all the original raster entities in the converted drawing. Use the Show Raster tool to examine the drawing with and without raster objects, to see which entities should remain and which should be deleted.

Zoom In (Trace Graphic Window)

Use the Zoom In tool to gain a close-up view of any portion of the image. The cursor will change to a magnifying glass with a (+) sign. Draw a selection window around the portion of the drawing to magnify.

Zoom Out (Trace Graphic Window)

Zoom Out returns to the previous zoom. It does not redraw to any particular percentage of zoom.

Zoom All (Trace Graphic Window)

Zoom All redraws the Trace screen to show the complete drawing, no matter what the current zoom factor.

Orthogonal Adjustment

When you first trace a bit-mapped image, the vector lines created are not always aligned correctly. Orthogonal Adjustment is the first of four tools that can modify the alignment of vector entities in the drawing. This tool will modify the alignment of selected objects so they are orthogonal -- aligned to true vertical and horizontal.

Select the lines to be adjusted, either by drawing a selection window or <Shift-Click>. Press the Orthogonal Alignment tool. An alignment dialog will appear.

Maximum Angle. This setting determines how far from orthogonal a line can be and still be modified by the command. To change the Maximum Angle value, click on the value and type a new number, or click one of the arrows for a higher or lower value.

Maximum Distance. This is the maximum distance lines can be shortened or lengthened when adjusted. This feature is found only in TurboCAD v4 Professional.

Only Selected Entities. If this item is selected, Trace will rotate only selected lines. If not selected, Trace will adjust all lines in the drawing. If no items are selected in the drawing, this item will be grayed out.

Baseline. If this item is selected, you can choose a line in the drawing to serve as the baseline, or reference. All rotated lines will be rotated horizontally or vertically using the baseline as the standard of reference. If this item is not selected, the horizontal plane of the drawing (0 degrees on the coordinate plane) will be used as the standard for adjustment.

To finish, press Start. All selected lines (or all lines, if no selection was made) will be rotated to true vertical or true horizontal if they fall within the parameters of the Maximum Angle setting.

Horizontal Adjustment

Horizontal Adjustment rotates the entire image so that a selected line becomes horizontal. To use, click the Horizontal Adjustment tool, and select the line that you want to become horizontal. The entire drawing will rotate so that the selected line is horizontal.

Vertical Adjustment

Vertical Adjustment rotates the entire image so that a selected line becomes vertical. To use, click the Vertical Adjustment tool, and select the line that you want to become vertical. The entire drawing will rotate so that the selected line is vertical.

Vertical/Horizontal Adjustment

Vertical/Horizontal Adjustment rotates the entire image so that a specified line becomes the new vertical or horizontal plane of the drawing. This tool is used to draw a line on the screen, using a click/drag sequence. Most of the time, you will use this tool to draw a line along an existing line in the drawing which you want as the horizontal or vertical standard for the drawing.

Text (Trace Graphic Window)

Any text in the original raster file will be converted to adjoining line segments. In TurboCAD v4 Professional you can add text to the drawing, to replace the converted lines.

When you click the Text tool, the cursor changes to a text insertion tool. Click in the drawing to set the text insertion point. A text placement dialog will then appear.

The current font and text size will appear at the bottom of the dialog. To change either setting, click the Font button. When the settings are correct, type the required text and click the OK button. The text will be inserted into the drawing.

Close

Menu: File|Close

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

See Also: Close All

Save 💾

The Save command saves the current drawing to disk, using the folder 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. If you have checked Prompt for Summary Info in the General property sheet of the Program Setup dialog (Options|Program Setup), the Summary Info dialog will also appear.

Tip: To create a new copy of the file, use File|Save As.

See Also: Save As Dialog, Summary Info 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 folder that you specify.

To save a drawing in a specific folder and under a specific file name:

1. Use the Save In box to navigate to the folder 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 TurboCAD (.TCW) file.
4. Click the Save button, or press <Enter>, to save the file.

If you have checked Prompt for Summary Info in the General property sheet of the Program Setup dialog, the Summary Info dialog will appear.

Tip: Other graphics programs are most likely to be able to read your TurboCAD drawing if it is saved as a .DXF file. This has become the standard file exchange type for CAD drawings. All TurboCAD entities translate well into the DXF format.

Subtopics

Save Setup

Save Setup

Special options for saving TurboCAD drawings are available from the Setup button in the Save As dialog.

TCW 3.0 file: Select this option to save a drawing in TurboCAD 3.0 format, if you need to share your work with a TurboCAD 3.0 user.

Optimizations: One of the two options, Faster Load Time and Smaller File Size, will always be used. The default is Faster Load time.

Full Precision: Check this option to enable the maximum depth of mathematical calculations performed to generate and save entities. If Full Precision is off, entities will be calculated and saved using only four decimal points of precision. This results in faster execution and smaller files, but compromises accuracy. The default is Full Precision.

Note: The lack of full accuracy can affect the ability of the program to correctly display and edit objects at extremely small scales. Unless you are using a slow machine, and know you do not need the extreme accuracy possible in TurboCAD, you should leave Full Precision on.

See Also: Save, Save All, Summary Info Dialog, 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, TurboCAD will prompt you for a folder and file name for those files.

If you have checked Prompt for Summary Info in the General property sheet of the Program Setup dialog (Options|Program Setup), the Summary Info dialog will appear once for each previously unsaved drawing.

See Also: Save, Save As Dialog

Page Setup Wizard

Menu: File|Page Setup Wizard

The Page Setup Wizard is a step-by-step guide to setting page size and drawing scale. Use it as an intelligent option to setting the Drawing Units (Options|Units and Scale), Paper Size (File|Page Setup), and Scale (Options|Units and Scale) dialogs separately. Advantages of the Wizard include automatic text and grid sizing according to the settings specified.

See Also: Page Setup Dialog, Program and Drawing Setup, Units and Scale

Summary Info Dialog

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.

Note: If a drawing is going to be used as a symbol, then Title should be a short description of the symbol, and Subject should be a more detailed description.

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 TurboCAD 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 the bitmap will use the same amount of space no matter how complex the drawing is.

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. Symbol previews are generally stored using this option.

Page Setup Dialog

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.

Tip: After you have selected options in these property sheets, you have two basic choices. The first choice is to make the changes in your drawing by clicking the OK button at the bottom of the dialog. The second choice is to print the drawing from the dialog, without making the changes in your drawing, by clicking the Print button. If you click the OK button, you can then print the drawing by choosing File|Print or clicking the Print button on the Standard toolbar.

Subtopics

Paper Property Sheet
Layout Property Sheet

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.

See Also: Print Dialog, Printer Paper and the Drawing Sheet, Layout Property Sheet

Subtopics

Printer Paper
Drawing Sheet Size
Options (Printing Only)
Page Layout Scale
Keep Centered

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 (which varies, depending on the printer driver you are using).
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. This dialog is specific to your printer driver, so check your printer or printer driver documentation for details.

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 they can be pasted against a backing. You may need to increase your margins, to make room for the crop marks.

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 and circles.

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. The drawing will be centered as it is fit onto 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 Options 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. It may be necessary to turn this option off, then on again, for the desired effect.

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 units, and relocating the paper in the drawing area.

Subtopics

[Margins](#)
[Rows and Columns](#)
[Origin](#)

Margins

Margins, the dotted rectangle inside the edges of the paper, let you know whether you are drawing inside or outside the printable area of your drawing.

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 Options|Units and Scale property sheet, but you can enter values in other units as well.

Tip: The Get Margins from Printer option will set the minimum margins for your printer and page size automatically.

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, TurboCAD will automatically increase the number of rows and columns to accommodate the change in the drawing sheet size.

Origin

The Origin box lets you move the paper to a different location in the drawing. You do this by specifying the location of the lower-left corner of the paper in absolute coordinates: Type the horizontal coordinate into the X box and the vertical coordinate into the Y box.

Tip: 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.

See Also: Coordinate Systems, Units and Scale Property Sheet

Print Dialog

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.

See Also: Page Setup Dialog, Printing, Views of the Drawing Space

Subtopics

Printer
Print Range
Print Source

Printer

The Printer control group lets you select any printer that is available on your system and access its properties. Click in the Name drop-down box to select any available printer. Click on the Properties button to access the current printer's property sheets. Click on the Page Setup button to go to the Page Setup dialog. Click on the Default Printer button to select the Windows default printer.

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.

Tip: This is useful for printing tiled drawings if you only want to print specific pages.

All. Select this option if you want to print all of the current drawing. If you have chosen 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.

Pages. 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. TurboCAD 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 Source

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

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

Send

The Send command is a standard Microsoft Office function. If you are running on a network and Microsoft Mail is installed on your system, 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, TurboCAD displays the four most recently saved drawings. These can be drawings that are currently open, 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, TurboCAD will make it active; if the drawing is not opened, TurboCAD will open the drawing.

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

Exit

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

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 TurboCAD drawing.

In addition to the features available from the Edit menu and toolbars, TurboCAD 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*.

Subtopics

Editing with the Mouse and Local Menu
Edit Menu Commands

Editing with the Mouse and Local Menu

TurboCAD 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.

Subtopics

Select Edit

Node Edit

Manipulating Text

Select Edit

Select Edit is the editing mode an object is in when you first select it with the Select Edit tool. 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.

See Also: Geometric and Cosmetic Select Modes

Tip: You can get information about selected objects by choosing View|Selection Info.

Subtopics

Geometric and Cosmetic Select Modes
Advantages of Geometric Select Mode
Moving the Selection's Reference Point and Rotation Handle
Moving a Selection
Scaling a Selection
Copying While Moving or Scaling
Rubber Stamp
Rotating a Selection

Geometric and Cosmetic Select Modes

TurboCAD offers two distinct selection modes: Geometric Select mode and Cosmetic Select mode. You can switch between the two modes by clicking the Geometric Select Edit mode check box, located in the General property sheet of the Options|Program Setup dialog.

In order to understand the difference between these selection modes, you must first realize that every TurboCAD entity has two separate sets of extents: *geometric extents* and *cosmetic extents*. Geometric extents are calculated based on the distance between the specific vertices that define the entity geometrically (for example, the three corners of a triangle, or the two endpoints of a line segment). Cosmetic extents are calculated based on distances between the outermost points on the object that will be drawn on the screen or on the printer. In many cases, the geometric and cosmetic extents will be exactly the same. This is true, for example, if you measure the extents of a single line segment drawn with the minimum (default) pen width. If, however, you are working with a double-line entity, or with an entity that is drawn with a pen that has greater than the default width, the geometric and cosmetic extents will differ.

Advantages of Geometric Select Mode

Other drawing packages use cosmetic selection only. What is the advantage of geometrically accurate selection? There are two main advantages:

▪   When you scale using geometric extents, it is easy to see exactly what is going to be scaled. If, for example, you scale a double-line entity, TurboCAD *does not* scale the width of the double line, just as it does not re-scale the width of the pen. Geometric Select mode makes it visually obvious that this is the case.

▪   When you scale an orthogonally oriented linear entity using geometric extents, you get accurate results. If, for example, you scale a 1-inch horizontal line segment 200 percent along the X axis, the line segment will always be 2 inches long. If you were to use cosmetic extents, the results would vary, depending on the width of the pen.

The disadvantage of geometric selection is that if you snap the endpoint of a wide line to another object, the cosmetic end of the line will overlap the object. You can work around this problem by selecting a line using cosmetic extents and using a snap mode to position the cosmetic end of the line.

Tips: If you want to scale the width of a double line as you scale an entity, you need to first turn the double line into two single lines. To do this, select the entity, then choose Explode from the Format menu. If the double line is contained within a group or within a block insert, you will have to choose Explode more than once, because the Explode command breaks an object down into its constituent parts one step at a time.

Another way to adjust the width of a double line, or to change the width of the pen, is to select the entity and double-click on it, bringing up its properties sheets. You can change the width of a double line in the Double Line property sheet, or the width of the pen in the Pen property sheet.

See Also: Explode, Properties Dialog

Moving the Selection's Reference Point and Rotation Handle

There are two ways to move the reference point of a selection to any position in the drawing area. The first way is to 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>.

The second way, new in TurboCAD v4, is to press <D> on the keyboard when an object is selected. This temporarily places the reference point in the cursor. Click at the desired new location, or use a snap mode, or type the coordinates of the point into the Coordinate Fields and press <Enter>.

It is possible to move the rotation handle in a similar fashion, by pressing <Ctrl> and then clicking the rotation handle. You can then place it using snaps or coordinate entry.

To restore the reference point to its original location at the center of the selection, press the <Ctrl> key while clicking on the reference point, then right click and chose Default Reference Point.

Tip: 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.

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).

Subtopics

"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 a 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 <Shift-Tab> to activate the Coordinate Fields, then type the coordinates of the desired location in the Coordinate Fields and press <Enter>.

Tip: 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.

See Also: Moving the Selection's Reference Point and Rotation Handle

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 TurboCAD drawing into another application. You may also use this method to create symbols or blocks by dragging into the symbol or block palette.

Warning: The behavior of snap modes is slightly unusual when you are moving an object using OLE drag and drop. If TurboCAD 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 TurboCAD cannot   find a snap point, then the object will simply be dropped at the point of release.

See Also: 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 its 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, press <Tab> to move to the Edit Bar, and then type 2 into the XScale field. Press <Tab> again and type 1.5 into the YScale field. Press <Enter> to finish.

Note: If the object you are scaling contains double lines or lines drawn with a pen width greater than 0, the results of scaling a selection will vary, depending on whether you are using geometric or cosmetic selection. For geometrically accurate results, use the default Geometric Select mode.

See Also: Geometric and Cosmetic Select Modes

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 selection handles.

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.

Tip: You can turn on the Make a Copy option temporarily by holding down the <Ctrl> key after you start to copy or scale the selection.

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. 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>.

Note: 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 Rotation Field on 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, press <Tab> to switch TurboCAD's focus to 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: The positive direction of rotation is counterclockwise by default. You can reverse the direction in the Angle property sheet of the Drawing Setup dialog (Options|Angle).

Tip: 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.

Tip: The rotation handle can be moved by pressing <Ctrl> and clicking on the handle.

See Also: 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, converting them into multilines.

▪   Delete segments of multilines.

▪   Close multilines, converting them into polygons.

▪   Close the endpoints of double lines.

▪   Reshape Bezier and spline curves by adding, deleting, and moving nodes and their tangent points 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, converting them into circles.

▪   Divide circles, arcs, and ellipses into arc segments of equal angular length.

There are three ways to activate Node Edit mode. You may 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 squares. The number and placement of the nodes depends on the type of object selected.

You may also select the Node Edit tool from the Standard toolbar; it is on a fly-out toolbar with the Select Edit tool.(Or use Menu: Edit|Node Edit.) The tool looks just like the Select tool, except for a dot inside the arrow. Click on the intended object, and it will be in Node Edit mode.

The third way to active Node Edit is to press the <Ctrl> key before clicking on an object with the Select Edit tool.

To deselect an entity in Node Edit mode, click anywhere in the drawing area other than on the selected object.

Note: You can only edit one entity at a time in Node Edit mode. 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 and block inserts can be nested, you may need to explode the selection more than once in order to isolate a single entity.

Subtopics

Node Edit with Linear Entities

Node Edit with Curves and Sketched Lines

Node Edit with Circles, Ellipses, and Arcs

Node Edit of Dimensions

Node Edit with Linear Entities

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 or deleting a line or double-line segment.

Subtopics

Adding a Node to a Line or Double Line Segment

Deleting a Node from a Line Segment

Moving a Node

Dividing a Line or Double Line Segment

Deleting a Line or Double Line Segment

Breaking Polygons

Closing Multilines

Closing Endpoints of Double Lines

Editing Double Line Cut Lines and Gaps

Adding a Node to a Line or Double Line Segment

To add a node to a single or double line segment on a line, 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.

Note: 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 or double line segment on a line, multiline, or polygon:

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, TurboCAD will keep the polygon closed by connecting the opposite endpoints of the line segments that were connected at the deleted node.

Note: 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.

See Also: Closing Multilines, Breaking Polygons

Moving a Node

You can reshape a single line or double line segment on a line, 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 by either using the mouse and any snap mode, or by typing the coordinates of the point in the Coordinate Fields and pressing <Enter>.

Note: You can also simply drag the node to any position you want.

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. TurboCAD displays the Divide Segment By dialog.
5. Enter the number of parts into which you want to divide the line into the dialog, and then press <Enter> or click the OK button.

Deleting a Line or Double Line Segment

The Segment Delete command from the local menu lets you delete a line segment. 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 delete.
4. Click the right mouse button to open the local menu and choose Segment Delete. The segment is deleted.

Note: If you delete a line segment that is part of a closed figure, the figure will no longer be closed.

Breaking Polygons

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

1. Select Node Edit from the Standard toolbar.
2. Click on the polygon.
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.

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

Closing Multilines

Node Edit allows you to close a multiline, changing 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.

TurboCAD 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, TurboCAD will simply close the polygon geometrically, since it is unnecessary to draw a line between the two points.

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

Warning: If you simply snap the endpoints of a multiline together you have something that looks exactly like a polygon; however, TurboCAD 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.

TurboCAD will close the double line with a flat end.

Editing Double Line Cut Lines and Gaps

Cut lines 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 cut lines in the first line, to even 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 very small 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 node points (spline curves gravitate toward the points, while Bezier curves pass through the node points, and gravitate toward separate control points).

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.

See Also: Adding a Node to a Line or Double-Line Segment, Breaking Polygons, Closing Multilines, Deleting a Node from a Line Segment, Moving a Node

Subtopics

Adding a Node to a Curve

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 part 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.

Note: 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.

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.

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.

Subtopics

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

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 is a blue square. You can manipulate this *radius 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 two nodes or directly enter values 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 represented as a red and a green square at the ends of dotted lines. You can manipulate these nodes to change the starting and ending angles of the entity.

2. You can 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.
2. Click the right mouse button to open the local menu, and choose Arc Divide. TurboCAD will display the Divide Arc By dialog.
3. Enter the number of parts into which you want to divide the arc, 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 dimensions. These are the nodes that are associated with the dimension text, so that changing the position of the nodes will cause automatic dimension text to be updated.

Note: If a dimension is associated with an entity, then moving the node attached to the entity will break the association between the entity and the dimension.

Manipulating Text

TurboCAD has a rich set of features for manipulating text, based on the principle that you can edit the text either as:

- TrueType font
- a group of polygons
- individual polygons (each character becomes one polygon)

When you first create text, it is a TrueType font. You can select the text, stretch it, and rotate it. When text is in this state you can edit it by changing the Attribute field of the General property sheet of 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 Text Format toolbar or the Text property sheet of the Properties dialog.

New in TurboCAD v4 is the ability to set the status of the text as Scalable, Standard, or Flexible.

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 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 Linear Entities, Select Edit, Text Property Sheet

Edit Menu Commands

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

Command	Function(s)
Undo	Reverse previous action.
Redo	Reverses the effect of the previous Undo command.
Cut	Remove selected objects and place them on the Windows clipboard.
Copy	Copy selected objects to the clipboard.
Paste	Paste objects into the drawing from the clipboard.
Paste Special	Paste objects from the clipboard as data or as OLE objects.
Clear	Remove selection, all objects, a single construction, or all constructions from the drawing.
Select	Activate the Select tool.
Node Edit	Activate the Node Edit Select tool.
Select All	Select all objects in the active drawing.
Select By	Select by query (Professional only), entity type, color, layer, or attribute. Also provides tool for selection using a free-form "fence."
Copy Entities Tools	Create multiple copies of objects along lines, around arcs, in two-dimensional arrays, and more.
Links	Make changes to linked OLE objects.
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. Undo is also available on the Local Menu.

See Also: Redo

Redo

Menu: Edit|Redo   **Hotkey:** <Ctrl>+<Y>   **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. Redo is also available on the Local Menu.

See Also: Undo

Cut

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

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

See Also: Cut, Paste, Paste Special, Copy Entities Tools

Paste

The Paste command pastes into your drawing any data that has been copied or cut into the clipboard. You can paste TurboCAD drawing objects, as well as text, bitmap, or Windows metafile data that has been placed in the clipboard from 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 TurboCAD, 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 TurboCAD 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 TurboCAD text object, which you can treat exactly like text created with the TurboCAD 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: There are only two types of data that can be inserted into TurboCAD as an editable TurboCAD drawing object. These are text data, which is inserted as a TurboCAD text object, and actual TurboCAD drawing entities that have been copied onto the clipboard from TurboCAD drawings. All other clipboard objects will be pasted either as OLE objects or as pictures (either bitmaps or metafiles).

Tip: 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.

See Also: 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: Selection, All, Construction, and All Constructions. You can reverse any clear with the Undo command. The following table summarizes the Clear commands.

Command	Function
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 constructions.
Clear All Constructions	Clear all construction entities in the drawing.

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, Cut

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 and circles. 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 entities (construction lines and circles) from your drawing.

Select Commands

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

TurboCAD provides a large set of selection options, summarized in the following table.

Command	Function
Select	Activate the selection tool.
Node Edit Command	Activate the Node Edit Select tool.
Select All	Select all objects in the drawing.
Select By Commands	Access a submenu that lets you select objects by query (Professional Only), entity type, color, layer, attribute, or with a free-form "fence."

Select

Menu: Edit|Select   **Toolbar:** Standard

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 near its border, 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>, choosing Cancel from the local menu, or clicking another drawing tool.

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.

Subtopics

Selection by Clicking
Selection within Rectangle

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 deselect 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.

Four function-key selection options are also available. <F6> will select the first object created in the drawing. Continuing to press <F6> will select additional objects in the order they were created. <F7> will select the last object drawn; continuing to press <F7> will select additional objects in reverse order. <Shift-F6> will add an additional object to the selection; in order of creation after the selected object. <Shift-F7> will add an additional object to the select, selecting the object drawn just previous to the selected object.

Node Edit Command

Menu: Edit|Node Edit   **Hotkey:** <Ctrl>+click   **Toolbar:** Standard

Use the Node Edit tool to select objects that will be edited using Node Edit commands and tools.

Select All

Menu: Edit|Select All   **Hotkey:** <Ctrl>+<A>

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, summarized in the following table:

Command	Action
Select by Query Dialog	Brings up a dialog that allows for multiple selection criteria.
Select by Entity Type Dialog	Brings up a dialog that lets you select objects by their type.
Select by Color Dialog	Brings up a dialog that lets you select objects by their color.
Select by Layer Dialog	Brings up a dialog that lets you select all the object on a layer or subset of layers.
Select by Attribute Dialog	Brings up a dialog that lets you select objects that share the same text in their Attribute property.
Select by Fence	Activates a tool that lets you select a set of objects by drawing a polygonal "fence"

Each of the selection methods can be cumulative; that is, you could first select by color, then add other objects selected by attribute to the total items selected.

Select by Query Dialog

Menu: Edit|Select By|Query

Select By Query allows for complex selection of entity variables. It is especially useful in large complicated drawings where color, layer, and entity type are all used to represent objects.

The command opens a dialog where you may build a query, step by step.

Subtopics

Query Procedural Language and Tools

Creating a Query Statement

Parse Query

Query Procedural Language and Tools

To build a query, you assemble a query statement. The query statement can be typed directly into the dialog, or generated using the available tools. A query statement is a logic statement, created using expressions, criteria, and variables.

Expressions. The query language uses the math symbols = (Equals), < (Less Than), > (Greater Than), <> (Not Equal), and parenthesis, plus the logic expressions AND, OR and NOT. Buttons are available for the logic expressions and the parentheses; the math symbols are generated when criteria are selected (or you may type them in directly).

Criteria. All the "Select By" options are available as criteria for a query: All, Selection, Field, Layer, Entity Type, Pen Color, Brush Color, and Attribute. Click the Criteria button to select these options.

Variables. Variables are the specific pieces of information that define criteria. For example, Yellow could be a variable that defines the criteria Color; "0" is a variable that defines the Criteria Layer. Variables are chosen along with criteria, or may be typed in directly.

Creating a Query Statement

Follow the steps below to create a sample query statement where Layer = "0" AND Color <> Yellow.

1. From the menu, select Edit|Select By|Query.
2. Click the Criteria button, then click on By Layer. The Select By Field dialog appears.
3. Click on the text box list arrow for Value. All available layer names will be listed. Select "0" and click OK.
4. Use the right arrow key on the keyboard, or your mouse, to move the blinking text insertion cursor to the end of the query statement. Press <Spacebar> once.
5. Click the AND button. Press <Spacebar> once.
6. Click Criteria and select By PenColor.
7. Click the Relation drop-down and select the <> (not equal) relationship.
8. Click the text box list arrow for Value. All available colors will be listed. Select Yellow from the list and click on OK.

Parse Query

The Parse Query command will test the syntax of the query statement. If there is an error in logic or format, a "Syntax Error" message will appear. If the query statement makes sense, a "Query was parsed successfully" statement will appear.

Note: The Parse Query command only tests to see if a query statement makes sense. It does not test to see if the statement will actually select entities or objects in the drawing.

Select by Entity Type Dialog

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 Color Dialog

Menu: Edit|Select By|Color   **Hotkey:** <Ctrl>+<K>

The Select By color command opens a dialog that lets you select all the objects in the drawing by specifying the color.

Note: Objects inside a group or block will not be found using Select By Color. The Explode command must be used to reduce groups or blocks to their basic elements before a By Color search would find any part of these objects.

Select by Layer Dialog

Menu: Edit|Select By|Layer   **Hotkey:** <Ctrl>+<L>

The Select By Layer command selects all objects that exist on a specific layer or on a specific set of layers.

You can use this dialog to select objects that exist on a particular layer or combination of layers, using common Windows list box selection techniques. For example, if you wish to select all objects on layers Floor and Furniture (assuming that you have created these layers in your drawing), you would hold down the <Ctrl> key while clicking on Floor and Furniture, highlighting both items. If you wish to add objects of these types to a current selection, click 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.

See Also: Layers Property Sheet

Select by Attribute Dialog

Menu: Edit|Select By|Attribute**Hotkey:** <Ctrl>+<I>

The Select by Attribute dialog lets you select all objects that share the same text in their Attribute property, which you can set for each object in the General property sheet of the Format|Properties dialog.

To use this dialog, type text in the Attribute text box that matches the contents of the attribute text box of a set of entities, then click OK to select the entities. Because the attribute property contains group and block names, you can use this dialog to select all groups or blocks that share the same name. The dialog also contains the following options:

Case Sensitive. Place a check mark in this box if you want the search to distinguish between uppercase and lowercase letters. Entering Wall with this checked would select objects that have the attribute "Wall", but not "wall" or "WALL".

All Entities. Select this option if you want to select all entities that share the specified text attribute.

Group Names Only. Select this option if you want to select only groups that share the specified text attribute.

Block Names Only. Select this option if you want to select only block inserts that share the specified text attribute.

Text Entities Only. Select this option if you want to select only text entities that share the specified text attribute.

Note: The text attribute of a text entity contains the same text that the entity displays.

You can also select objects that share attribute text that begins with the same characters by using the * symbol. The * symbol stands for any terminating characters in the text. For example, objects with text attributes "Wall 1", "Wall 2", and "Wall 3" will all be selected if you type Wall* into the Attribute text box. This is convenient for selecting all groups and blocks that belong to the same general category. You might want to keep this in mind when you are choosing names for groups and blocks.

Note: All characters that follow the * symbol will be ignored. If you type Wall * with Window, for example, you will select objects with the attribute "Wall 1", and "Wall 2", as well as "Wall 1 with Window" and "Wall 2 with Window".

See Also: Properties Dialog

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. This fence works exactly like the Irregular Polygon drawing tool.

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: Like the Select tool, the Select by Fence tool responds to the state of the Open Window Mode option. You can change this option 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.

See Also: Selection within Rectangle

Subtopics

Using Snap Modes
Constraining Angles with Ortho
Setting Snap Modes and Ortho

Copy Entities Tools

Menu: Edit|Copy Entities

TurboCAD'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. The following table summarizes the entity copying tools:

Command	Function
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.
Linear Fit Copy	Make copies along a line, specifying the total distance into which all the copies fit.
Radial Fit 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.
Array Fit 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: 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>.

Note: If you do not change the rotation or scale of the copies that you create, TurboCAD 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, TurboCAD will make 20 copies of the original. If you do not specify a new scale or rotation, TurboCAD will make 19 copies, using the original as the 20th copy.

Linear Copy

Menu: Edit|Copy Entities|Linear   **Toolbar:** Standard, Copy fly-out

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:** Standard, Copy fly-out

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 center point 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 the 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. TurboCAD will choose appropriate 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

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 center points 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.

Linear Fit 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 Fit 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 not 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.

Radial Fit Copy

Menu: Edit|Copy Entities|Fit Radial   **Toolbar:** Standard, Copy fly-out

1. Select the object or objects that you want to copy.
2. Choose the Radial Fit Copy tool.
3. Define a point for the center point 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 the 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.

TurboCAD 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.

Array Fit Copy

Use Array Fit 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 XBound and YBound 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 XBound and YBound 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

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.
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. (The line that appears in the drawing space looks and behaves like an angular construction line.)
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.

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

Note: To mirror text you must first make sure the text is "flexible." You can change the setting of selected text in the Text sheet of the Properties dialog.

Vector 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 the angle and distance of the vector directly in the Edit Bar and press <Enter>.

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 TurboCAD drawing with the Insert|Object command. You can edit an OLE object within TurboCAD using the Windows application in which the object was created (the *source* application). For example, you can insert a Windows Paint object into your drawing and edit it within TurboCAD using the Windows Paint 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 to 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|Object

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.

Note: 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. For a bit-mapped image, a menu item entitled Linked Image Object will appear, with options that allow you to edit, open or convert the image.

See Also: Embedded and Linked OLE Objects, Insert Object

Drawing Environment

The View menu provides control over two aspects of the drawing environment. Using View menu commands, you may:

- Change the view of your drawing.
- Modify your desktop by displaying and hiding toolbars and other interface controls.

Note: There are several new view features in TurboCAD v4, when compared with previous versions. The new features include docking palettes, on-the-fly measurement commands, grid magnification options, insert viewport, and aerial view. The experienced TurboCAD user will find many productivity improvements in these new features.

Subtopics

Views of the Drawing Space

Zooming

Panning

Inserting Named Views with Insert Viewport

Aerial View

Grid Magnification and Set Origin

Setting Up the Desktop

View Menu Commands

Views of the Drawing Space

The TurboCAD drawing space is your creative surface. It is a plane on which you place drawing entities and other objects. A view is the area of the drawing space that appears on your screen. As your drawing becomes larger and more complex, it's important 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. TurboCAD offers a variety of options for panning and zooming that get you quickly to the exact view that you need.

If you arrive at a view that you want to return to later, you can save it as a Named View, then return to that view at any time. If you need to see a close-up of two parts of the drawing at the same time, or you need to see both a close-up and a broad overview, use either Aerial View or Insert Viewport (in the Insert Menu).

Note: Aerial View is especially useful if you do not have a large monitor.

Tip: Another powerful way to control your view of the drawing is by selectively viewing layers. You can control the viewability of layers through the Options|Layers property sheet.

See Also: Layers Property Sheet, Named View Dialog

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, which are discussed in Zoom 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 two. You can set the zoom factor in the Options|Program Setup dialog.

Notes: When you zoom in and out using the plus and minus keys of the numeric keypad, 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, TurboCAD simplifies the representation of very small objects on the screen to increase redraw speed. You can set the size at which this occurs using the Suppress Display... control on the Options|Display property sheet.

Tip: 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.

See Also: Display Property Sheet, Program Setup Dialog, Zoom Commands

Panning

Panning means to move the view to another location across the plane of the drawing. TurboCAD provides several different options for panning, which are discussed in Pan Commands .

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 scroll bar arrow buttons to scroll one step, click in the body of the scroll bar to scroll about one third of a screen, or drag the "thumb" of the scroll bar 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.

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

Inserting Named Views with Insert Viewport

As mentioned earlier, it is possible to name specific views, then adjust the screen to that particular view. New in TurboCAD v4 is the ability to display a named view on the drawing area, using the Insert Viewport command. This feature allows you to see the exact contents of a named view without dedicating the entire screen to it.

Using Insert Viewport requires that one or more named views have already been created. Instructions on using Insert Viewport are found in Insert Menu Commands .

Tip: Named views are accessible from the Views toolbar.

Aerial View

Aerial View is a window you open to provide a second view of the drawing. You may use Aerial View to have a close-up view of one small part of the drawing, or of the entire drawing.

You can use the Aerial View to adjust the view in the drawing screen, without changing the Aerial View. Five drawing tools and a cursor-based tool make it easy to maintain two views of your work, adjusting it as necessary.

Grid Magnification and Set Origin

New in TurboCAD v4's View menu are commands to modify the grid without returning to Drawing Setup. You may double or halve the current distance between grid lines, toggle the display of the grid, or change the origin of the grid.

Tip: The grid magnification and set origin features are also available as a toolbar.

Setting Up the Desktop

In TurboCAD 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 TurboCAD desktop.

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

See Also: Program Setup Dialog, TurboCAD Desktop

Subtopics

Displaying and Hiding Controls

Docking Toolbars

Docking Palettes

Displaying and Hiding Controls

The View menu provides a number of commands for displaying and hiding interface controls such as the Edit Bar, Status Bar, Aerial View, 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 TurboCAD toolbars through the Options|Toolbars and Menus property sheet, which is also accessible through the View|Toolbars command.

Tip: It's generally a good idea to set your preferred settings in the Options|Toolbars and Menus property sheet, and then use the View menu to show and hide individual controls as you work.

Tip: Many of these viewing commands are now available on the view toolbar.

See Also: Desktop Property Sheet

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 four places on the TurboCAD 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 bottom of the desktop. When you dock a toolbar on the upper or lower sections 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 be contained in their own window, which acts as a floating palette. A floating palette is always visible in front of your drawing.

Note: The Edit Bar and the toolbars that contain text fields (Property and Text Format), cannot be positioned vertically.

Docking Palettes

New in TurboCAD v4 are docking palettes for symbol libraries, blocks, selection information, and measurement.

You may toggle a docking palette between being docked and floating over the drawing area by double-clicking on the margin just inside the boundaries of the palette window. You can also drag the name tab at the bottom of the palette into the center of the drawing area to float the palette. You can re-dock a floating palette by dragging it to the side of the screen. Docking palettes can attach to either side of the desktop, but not the top or bottom.

View Menu Commands

The following table summarizes the commands you can access from the View menu:

Command	Function
Redraw	Manually refresh a drawing that has changed.
Zoom	Access tools and commands for moving your view closer to or further away from an area of the drawing.
Aerial View	Navigate zoomed views by use of a small overall view.
Pan Commands	Pan to Point and Vector Pan let you either pick a point in the drawing and move it to the center of the screen, or pick a point in the drawing and move it to a particular location on the screen.
Named View	Save your current view under a name, or move to a previously named view.
Create View	Use selection rectangle to pick an area on the screen to save as a named view.
Previous View	Return to the previous view of the drawing.
Grid	Access commands to display, reorient and resize the grid.
Cursor	Set cursor style and aperture display.
Margins	Display or hide the Margins.
Background	Display or hide the Background.
Rulers	Display or hide the Rulers.
View Status Bar	Display or hide the Status Bar.
View Edit Bar	Display or hide the Edit Bar.
Toolbars	Display the Desktop property sheet in which you can display or hide toolbars.
View Symbol Library	Display or hide the Symbol Library palette.
Blocks	Display or hide the Blocks palette.
View Selection Info	Display or hide the palette of information about the currently selected object.
View Measurement Info	Display or hide the measurement data palette.

Redraw

In most cases TurboCAD 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

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:

Command	Action
Zoom In	Move one step into the drawing.
Zoom Out	Move one step further from the drawing.
Zoom Window	Activate the Zoom Window tool to 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

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 Options|Program Setup dialog.

Note: 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 <+> key on the numeric keypad, you will zoom in to an area defined so that the mouse cursor is kept in the same relative position within the view rectangle.

See Also: Program Setup Dialog

Zoom Out

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 Options|Program Setup dialog.

Note: 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 <-> key on the numeric keypad, you will zoom out to an area defined so that the mouse cursor is kept in the same relative position within the view rectangle.

See Also: Program Setup Dialog

Zoom Window

The Zoom Window tool enables you to select a rectangle area of the screen and zoom in to a view of 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 encompasses the area that you want to view.
4. Click to specify the opposite corner of the window. TurboCAD 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, TurboCAD chooses the closest view based on the zoom rectangle you define.

View Extents

View Extents 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

Full View sets the view to the full drawing page.

View Printed Size

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.

Aerial View Command

Aerial View provides a second viewport of the drawing area. It may display any part of the drawing you select. Only one aerial view can be open on the desktop.

To create an aerial view:

1. Select Aerial View from the view menu or the view toolbar.
2. Click to specify one corner of the aerial view window.
3. Move the mouse cursor to the point where you want to place the diagonally opposite corner of the window, so that the window encompasses the area that you want to view.
4. Click to specify the opposite corner of the window. TurboCAD will immediately create the aerial view window, displaying the view you selected. You can also drag the mouse to create the aerial view window, in which case your view will be created immediately after you release the mouse button.

You may move or resize the aerial view window as you would any other window: Drag the title bar to move the window; drag a border or corner to resize the window.

Subtopics

Aerial View Tools

Aerial View Tools

Five tools appear in a toolbar just beneath the Aerial View's title bar. A sixth tool is cursor-based. Note that the five tool icons are in two groups. The first three affect the view inside the aerial view window; the last two are used inside the aerial view window, but affect the drawing.

Aerial Zoom In. Move one step into the aerial view.

Aerial Zoom Out. Move one step back from the aerial view.

Aerial Zoom Extents. Zoom to an aerial view containing all objects in the drawing.

Aerial Zoom Window. Select a new view *for the drawing* window with a selection rectangle in the aerial view.

Aerial Pan View. Select a new drawing view at the same scale moving the red rectangle that appears inside the aerial view window.

Aerial Cursor Pan. Adjust aerial view with cursor. Place the cursor just inside a border of the aerial view window. After a pause, the cursor will change to a small solid arrow and start panning the drawing in the direction the arrow points. Move the cursor away from the edge to end panning.

Note: You can roll up or roll down the Aerial View window   by clicking the roll-up button (similar to the minimize button in most Windows applications) on the title bar. You can also right-click the Title bar to turn MultiPicker option ON, which enables the Aerial View Window to be used across multiple drawings.

Pan Commands

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

Command	Function
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 crosshairs 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

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.

Named View Dialog

Once you have used the zoom and pan features to arrive at exactly the view of your drawing that you want, you won't want to repeat all that work to get back to the same view later. You can use the Named View dialog to save a view so that you can return to it at any time. Named views are saved with your drawing and remain available until you explicitly delete them.

See Also: Create View

To save a view under a name:

1. Use the zoom and pan commands to arrive at the view you want to name.
2. Choose View|Named View from the menu to display the Views dialog.
3. Click in the Name text box and type a name for the view. The name can be up to 128 characters in length and can include spaces.
4. Click the New button to add the name to the list of named views.

Tip: Give the view a descriptive name that will help you remember it, such as "Door Leading to Garden." If you use names like "View 1" and "View 2," you'll quickly lose track of which is which.

Subtopics

Returning to a Named View
Deleting a Named View
Modifying a Named View

Returning to a Named View

To return to a view that you have previously saved under a name:

1. Choose View|Named View from the menu, or the View toolbar, to display the Views dialog.
2. Select a view in the Name list box, then click the Go To button to go to that view. The view will change while the dialog box is open, so you can check the screen to see if the view is correct without leaving the dialog.
3. When you have found the view you want, click the Close button to leave the dialog.

To immediately go to a named view and close the dialog, double-click on the name of the view.

Deleting a Named View

To delete a named view that you no longer need:

1. Choose View|Named View from the menu to display the Views dialog.
2. Select a view in the Name list box, then click the Delete button to delete the view. Click the Close button to leave the dialog.

Modifying a Named View

The modify button of the Named View dialog allows you to replace a view that you have selected in the list. To do this:

1.   Use the zoom and pan tools to go to a view with which you want to replace a currently named view.
2.   Choose View|Named View to display the Views dialog.
3.   In the list box, select the named view that you want to replace.
4.   Click the Modify button. Your current view will replace the named view that you selected.

Create View

The Create View tool enables you to select a rectangular area of the screen and save this area as a named view. You can use this view as you would any other named view created with the Named View tool.

To use the Create View tool:

1. Choose Create View from the View menu.
2. Click to define one corner of the rectangular area that you want to save as a named view.
3. Move the mouse cursor to the point where you want to place the diagonally opposite corner of the rectangle and click: The Create Named View dialog will appear.
4. Type a name for the new view in the text box, then click OK to save the named view and close the dialog.

See Also: Named View Dialog

Previous 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 without using Aerial View.

View Grid Options

The View Grid commands control the display of the drawing grid. The grid is a pattern of dots, crosses, or lines that divides your drawing space geometrically into Cartesian, isometric 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 Options|Grid property sheet.

The following table summarizes the grid commands and tools:

Command	Function
Display	Toggles the grid display on/off.
Size * 2	Doubles the distance between grid points.
Size / 2	Halves the distance between grid points.
Set Origin	Changes the location from which the grid originates.

Tip: Use Set Origin to have the grid line up with a corner or other strategic point on an object in your drawing, then set the grid distance to a logical interval for the project.

Note: 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, the local menu, or the keyboard equivalent <G>.

See Also: Coordinate Systems, Grid Property Sheet, Snap Modes, Using Single Entry Keyboard Equivalents

Cursor View Commands

Menu: View|Cursor

Two cursor view options are available: Aperture and Crosshairs. Both function as toggles.

Subtopics

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

View Aperture

When selected, Aperture displays the snap aperture, a circular area surrounding the cursor. An entity (or grid point) must be inside the circle in order for a snap command to find the entity or grid point. The aperture is always active; this command merely toggles the view of the aperture.

Tip: You can set the radius of the snap aperture in the Options|Program Setup dialog.

View Crosshairs

Menu: View|Cursor|Crosshairs

Crosshairs changes the drawing cursor from a small cross to one that stretches across the entire drawing screen. By extending the crosshairs, you can often more clearly identify how the current location of the cursor corresponds to the ruler and to other objects in the drawing.

See Also: Program Setup Dialog

View Margins

The View Margins command is a toggle that displays and hides the Margins. Margins, the dotted rectangle inside the edges of the paper, let you know whether you are drawing inside or outside the printable area of your drawing.

See Also: Drawing Setup Dialog

View Background

The View Background command is a toggle that displays and hides the Background.

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 Paper/World 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 Options| Drawing Setup dialog. The rulers will change to reflect the current scale and display.

See Also: Drawing Setup Dialog

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. At printed size, the drawing's zoom percentage is 100.

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 menu bar, or at the bottom of the screen. You can also place the Edit Bar over the drawing space, where it will float over the drawing as a palette.

Tip: Although you can draw in TurboCAD 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 visible at all times.

Note: The fields in the Edit Bar change dynamically, depending on the currently active drawing or editing tool. The number of fields varies, 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.

See Also: Drawing with the Edit Bar

View Toolbars

Menu: View|Toolbars

The View|Toolbars brings up the Options|Toolbars and Menu Property Sheet. You may add or remove toolbars from the desktop. In TurboCAD v4 Professional you can also change their appearance, or even create custom toolbars.

View Symbol Library

Menu: View|Symbol Library

The View Symbol Library command displays the Symbol Library palette. The Symbol Library palette shows the symbols available in any library you have opened.

Symbols are pre-drawn objects available for repeated use. In TurboCAD, symbols can be fully independent TurboCAD (or other CAD) drawings. You can draw an object that you intend to reuse, save it as a drawing, then pick it from the symbol library palette when working on future projects. Any folder in Windows containing vector drawings can be recognized by the symbol library palette.

TurboCAD will recognize an individual drawing file as a symbol, and can display it in the symbol library palette. TurboCAD also allows you to create a symbol library file, containing several related objects, and can display the contents in the symbol library palette.

To identify a symbol in the palette, move the cursor over it. A tool tip will identify the symbol by name. To see the symbol's path name, right click on the symbol's icon.

New in TurboCAD v4 is the ability to dock the Symbol Library palette the way you would a toolbar. To switch between a floating palette and a docked palette, double-click on the margin that separates the outside boundaries of the palette from the interior (raised) sections, or drag this area to the center of the screen.

The title and description of the symbol is determined by information entered into the Summary Info dialog when the symbol is first created. The Title field of the Summary Info dialog determines the text that appears under the icon. The Subject field determines the icon description.

When the Symbol Library palette is open, you can:
- Place a symbol by dragging and dropping it onto the drawing.
- Turn a selected part of your drawing into a symbol, using drag and drop.
- Change to another open symbol library using the drop-down list.
- Open a new symbol library using the Open button on the palette.
- Change your view of the Symbol Library.

See Also: Symbol Libraries Property Sheet, Summary Info Dialog

Subtopics

Placing a Symbol in a Drawing

Symbol Management Options

Naming New Symbols

Storing New Symbols

Turning a Selection into a Symbol

Changing to Another Symbol Library

Changing Your View of a Symbol Library

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. You may need to click the symbol after you drop it into the drawing, to restore the focus of the program on the drawing area, if you intend to scale or move it.

The symbols that IMSI provides with TurboCAD 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 just a TurboCAD drawing, and it won't be a group unless the person who created the symbol saved it as a group.

Symbol Management Options

Any TurboCAD drawing can be a symbol. To create your own symbol library, all you 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.

Before you start saving drawings or drawing parts as symbols, you need to plan your file management strategy. There are two fundamental decisions to make: How to name the symbols, and where to store them.

Naming New Symbols

Every new symbol must be given a name. You can have TurboCAD generate a name, or you may assign one. The Auto-Naming property sheet (Options|Auto-Naming) is used to set which method you prefer.

Auto-naming controls the naming process for Groups, Blocks, and Symbols. For symbol libraries, three command settings need to be considered.

Generate Symbol Name. If this item is checked, TurboCAD will automatically assign a name to the symbol. The prefix listed in the "Symbol Name Prefix" text box will be used as the first part of the symbol name, followed by a sequence number in place of the @ symbol (0 for the first symbol, 1 for the second symbol, and so on). If this item is not checked, TurboCAD will prompt you to provide a name for the symbol.

Prompt for Library Locations. If this selection is checked, TurboCAD will give you the opportunity to specify a folder for the symbols you have created. You may save each symbol as soon as you create it (by clicking the Save Symbol tool in the Symbol Library palette), or wait until you close the active drawing (to save all the new symbols at once). If Prompt for Library Locations is not checked, the symbols you create during a drawing session will be saved to the default (pre-selected) folder.

Symbol Name Prefix. If Generate Symbol Name is selected, each symbol you save will be named with the prefix listed here, followed by a sequence number in place of the @ symbol.

Storing New Symbols

If the Options|Auto-Naming|Prompt for Library Location option is checked, a special Save As dialog for symbols will appear when symbols are saved.

The options in the Save As dialog work as follows:

Library File (*.SLW). A TurboCAD Library file is a drawing saved with the .SLW extension instead of the .TCW extension. TurboCAD will display the contents of an .SLW file in the Symbol Library palette just as it will the contents of any other windows folder containing drawing files. The advantage of using .SLW files is that many symbols can be saved to disk under one unifying name.

Library Folder. Use this option to select any Windows folder as the storage location for symbols. The advantage of storing symbols by folder is that you can combine symbols created at different times into one storage area.

Note: If you save a .SLW file in a folder used to store symbols, it will appear as a separate library in the Symbol Library.

Turning a Selection into 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 yellow 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. An icon for the symbol will appear in the palette.

5. You may save the symbol as a file now. Click the Save tool at the top of the palette. 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, TurboCAD will open a new symbol library in your drawing.

See Also: Save As Dialog

Changing to Another Symbol Library

You can quickly switch between active symbol libraries open in your drawing. Open a different symbol library in the Symbols Folder list box, just above the display of symbols in the Symbol Library palette. To view the list of available libraries, click the list box display arrow. To switch to another library, click on its name in the list box. The contents of that library will then be displayed in the Symbol palette.

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, click the appropriate icon:

- Names
- Small Icons
- Large Icons

When symbols are displayed either as icons or small icons, the Windows tool tip feature is available to provide additional information about any symbol. Place the cursor on any symbol to view its name. Right-click on a symbol to see its file path name (folder location).

View Blocks

Blocks are similar to groups, in that they consist of a collection of entities unified into a single object. But while each group is a separate object in your drawing, blocks are stored in your drawing's block library and then inserted into your drawing. Each block insert is a reference to the block in your drawing's block library. Because each block insert is a reference to a block, the inserts take up very little memory space in your drawing file. Blocks are therefore most useful if you want to store drawing objects that you will use more than once in a particular drawing.

The View|Blocks command displays the Blocks palette.

Note: Block libraries are only available to a specific drawing. Symbol libraries are available to any drawing at any time. Drawing elements that you want available for repeated use from project to project should be saved as symbols for easy retrieval.

See Also: Create Block, Groups and Blocks, Block Insertion Property Sheet

Subtopics

Inserting Blocks

Drag and Drop Block Creation

Deleting a Block from the Palette

Changing Your View of the Block palette

Inserting Blocks

To insert a block into your drawing, simply drag it out of the Block palette and drop it into your drawing. The inserted block will be selected after you place it, so that you can move, scale, or rotate it.

Drag and Drop Block Creation

You can create a block by dragging any selected objects into the Block palette.

1. Select the objects that you want to turn into a block.
2. Place the cursor over the selection's Reference Point (the yellow circle at the center of the selection rectangle). The cursor turns into a four-directional pointer.
3. Hold down the mouse cursor and drag the selection into the Block palette, then release the mouse button. If you have checked the Prompt for Name option in the Groups and Blocks property sheet, you will be prompted for a block name. If you have checked the Insert Blocks When Creating option, a block insertion will be automatically inserted in place of the selection.

Note: Blocks will be created on the same layer as the objects found in the block. Additional placements of this block will also be on the same layer. If you require the block to be on a specific layer, make sure to change this in Line Properties prior to creating the block.

Deleting a Block from the Palette

To delete a block from the palette, first click on the block name or icon to highlight it, and then click the Delete button (X mark) at the top of the palette.

Changing Your View of the Block palette

The Blocks palette can display its symbols as icons, small icons, or as names. To switch between these three options, click the corresponding button.

- Names
- Small Icons
- Large Icons

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.

Like the Symbol Library/Blocks palette, the Selection Info palette can be docked into the desktop or allowed to float over the drawing area. To switch between docked and floating, double-click on the margin just inside the palette's borders. You can also drag the name tab to float the particular palette while leaving the other palettes docked.

The Selection Info palette contains two windows. The upper window shows the type and structure of the objects selected. If an object such as a group or block contains 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.

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

View Measurement Info

The View Measurement Info command displays and hides the View Measurement Info palette. This palette displays the results when any of the four Measurement commands are used.

Like the Symbol Library palette, the View Measurement palette can be docked into the desktop or allowed to float over the drawing area. To switch between docked and floating, double-click on the margin just inside the palette's borders. You can also drag the tab name to float the palette, leaving the other palettes docked.

Creating and Inserting Objects

The Insert menu gives you access to all of the TurboCAD 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 (lines and circles that you use as reference points or to mark off areas in your drawing).
- Hatch entities (brush patterns defined by a path)

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

- Viewport (a different ''camera'' view of part of the drawing)
- Files (all standard CAD file types).
- Pictures (Windows metafiles or bitmaps).
- OLE objects, which you can edit using other Windows applications.

Note: There are, in addition, two other types of objects that can be inserted into a drawing: blocks and symbols. You insert these directly from the Blocks and Symbols palettes, which you can display from the View menu.

See Also: View Blocks, View Symbol Library

Subtopics

Creating Entities

Insert Menu Commands

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.
- Specifying quantities, such as length and angle, using the Edit Bar.
- Using SEKEs, Single Entry Keyboard Equivalents.

See Also: Finishing and Canceling, 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 TurboCAD Quick Tour from the Opening screen, and look at the TurboCAD Guide to Learning CAD.

Subtopics

Formatting Entities

Defining Points and Specifying Measurements

Drawing with the Edit Bar

Using Single Entry Keyboard Equivalents

Finishing and Canceling

Changing Focus

Using the Local Menu

Formatting Entities

When you draw an entity in TurboCAD, the entity is formatted using the current properties of the tool in use. If you do not adjust these properties, the TurboCAD defaults will be used. 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 (or select the entity, then choose Format|Properties).

Note: You cannot access or change the properties of Construction Entities (construction lines and circles). You can, however, edit the color and line style via the layers dialog.

See Also: Setting Properties, Property Toolbar, Properties Dialog

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.

Subtopics

Defining Points with the Mouse

Defining Points Using the Coordinate Fields

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 TurboCAD's *Ortho* setting.

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, TurboCAD will beep, and it won't define a point at all. To find out how to change 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; TurboCAD then reverts to the Ortho setting in the Modes menu.

If you are using TurboCAD's default 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 Options|Angle). Some CAD users, for example, prefer to restrict angles to increments of 15 degrees.

See Also: Angle Property Sheet, Ortho

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

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 menu; to set a "one shot" snap mode that stays in effect until you define the next point, use the local menu or press a SEKE – Single Entry Keyboard " Equivalent. 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. To use a SEKE, press the key that matches the snap you need.

Note: Both the local menu and SEKEs give you "one shot" mode changes. If you select a snap command using either method, the mode will change back to the setting in the Modes menu after you define the next point.

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), or select either "Lock X Position" or "Lock Y Position" from the Modes Menu.

See Also: Coordinate Fields, Coordinate Systems

Tip: The Coordinate Field lockdown boxes are particularly useful when you lock down just one field. 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 center point 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.

Note: 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.

Subtopics

[Using the Edit Bar Lockdown Boxes](#)

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.

Using Single Entry Keyboard Equivalents

New in TurboCAD v4 are Single Entry Keyboard Equivalents, or SEKEs. Now in addition to using the local menu as you draw, you can press a single key to access the most common snaps and other drawing aids. An example would be to close a multiline by pressing <V> for Vertex. The cursor will use Snap Vertex to attach the multiline to the nearest endpoint.

Taking advantage of SEKEs means you can keep both hands busy, one hand on the mouse and one hand on the keyboard. A list of SEKEs appears in Local Snaps and SEKEs , which covers the Modes menu commands.

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 and concentric circles, are composed of an indeterminate number of parts. You can add as many segments to a multiline, or as many circles to a set of concentric circles, as you like.

Users of earlier versions of TurboCAD may be confused by the way you cancel and finish such entities in TurboCAD v4. However, with a little practice you will find that TurboCAD v4 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: When you are working in the Coordinate Fields, pressing <Enter> simply defines a point as specified in the Coordinate Fields.

Tip: 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, TurboCAD will complete the entity by drawing a line or curve from the entity's last point to its first point.

Changing Focus

When you are drawing and editing, you will need to move back and forth between TurboCAD'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 Status Bar, 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.

Subtopics

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 Status Bar by clicking in one of its fields. Then you return to the drawing area by clicking there.

The tricky part is that when you return to the drawing area, you will probably want to reactivate it first, so that you can get appropriate feedback while you move your mouse, and only then continue the current operation. For this reason, you need to click twice. The first click reactivates the drawing area, restoring all feedback, and the second click actually continues the current drawing operation.

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>.

If you are in the drawing area you can also use <tab> or <shift + tab> to move to the Edit or Coordinate bars.

See Also: Defining Points and Specifying Measurements, Finishing and Canceling

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.

Insert Menu Commands

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. The following table summarizes the contents of the Insert menu:

Tool	Function
Construction	Create construction lines and circles to use as points of reference.
Point	Draw single-point marker entities in various shapes.
Line	Draw lines, rectangles, polygons, and lines tangent to circles and arcs.
Double Line	Draw double line segments, rectangles, and polygons.
Circle/Ellipse	Draw circles and ellipses.
Arc	Draw circular and elliptical arcs.
Curve	Draw Bezier curves, spline curves, and freehand sketches.
Text	Add single- or multiple-line text to the drawing.
Dimension	Draw dimension entities.
Viewport	Place a close-up view in the drawing.
Path Hatching	Draw a hatched or filled area by defining its boundaries.
File	Insert a CAD file into an open drawing.
Picture	Insert images (bitmaps or metafiles) into the drawing.
Object	Insert OLE objects into the drawing.

Construction Tools

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

Tip: If there are existing entities in the drawing, it is generally best to use snap commands (from the local menu or a SEKE) to place constructions.

Note: Constructions 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, but you can change their color and line styles via the layer dialog box.

The following table summarizes the Construction tools:

Tool	Function
Angular Line	Draw a construction line at any angle.
Horizontal Line	Draw a horizontal construction line.
Vertical Line	Draw a vertical construction line.
Center and Point Circle	Draw a construction circle defined by its centerpoint and a point on its perimeter.
Double Point Circle	Draw a construction circle that passes through two defined points 180 degrees apart.
Triple Point Circle	Draw a construction circle that passes through three defined points.

Angular Construction Line

Angular construction lines intersect a point that you define at any angle. You can constrain the construction line to predefined angles using Ortho mode. To place an angular construction line in your drawing:

1. Activate the Angular Construction Line tool.
2. Define a point in the drawing space that the construction line will pass through.
3. Define a second point. The construction line will intersect both points.

You can also define just the first point, specify the slant of the construction line in the Edit Bar, and then press <Enter>.

Horizontal Construction Line

Menu: Insert|Construction|Horizontal Line   **Toolbar:**
Standard, Construction Fly-out

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. TurboCAD draws a horizontal construction line that intersects the point you have defined.

Vertical Construction Line

Menu: Insert|Construction|Vertical Line   **Toolbar:** Standard, Construction Fly-out

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. TurboCAD draws a vertical construction line that intersects the point you have defined.

Center and Point Construction Circle

Menu: Insert|Construction|Center and Point Circle   **Toolbar:** Standard, Construction Fly-out

The Center and Point Construction Circle tool enables you to create a construction circle by defining its center point and any point on its perimeter:

1.   Activate the Center and Point Construction Circle tool.

2.   Define a point for the center of the construction circle.

3.   Do one of the following:

▪   Define a point on the perimeter of the circle. TurboCAD will draw a circle with its center at the first point, and its perimeter intersecting the second point.

▪   Type the radius or circumference of the construction circle in the Edit Bar fields, then press <Enter>.

Double Point Construction Circle

Menu: Insert|Construction|Double Point Circle   **Toolbar:**
Standard, Construction Fly-out

Use the Double Point Construction Circle tool to draw a construction circle by defining two opposite points on the circle's perimeter:

1.   Activate the Double Point Construction Circle tool.

2.   Define a starting position, which will be a point on the perimeter of the circle.

3.   Do one of the following:

▪   Define a second point. The circle will be drawn through the two points, which will be 180 degrees apart.

▪   Type the circle's diameter (or its circumference) and angle of the circle's diameter in the Edit Bar fields, then press <Enter>.

Triple Point Construction Circle

Use the Triple Point Construction Circle tool to draw a construction circle that passes through any three points:

1. Activate the Triple Point tool.
2. Define three points.

Note: The Edit Bar is not used with this tool.

Point

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.

Subtopics

Dot, Star, Square, Cross, and Circle

Dot, Star, Square, Cross, and Circle

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

TurboCAD has five styles of points, four of which 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: Points cannot be scaled or rotated without first exploding them into polygons.

Tip: 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, and then change its Point Type in the Point property sheet. You can also change the size of the point with this method.

Line

Menu: Insert|Line   **Toolbar:** Standard, Line

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. The following table summarizes the line tools:

Tool	Function
Single	Draw a single line segment.
Multiline	Draw a series of connected line segments.
Polygon	Draw a regular polygon with an arbitrary number of sides.
Irregular Polygon	Draw a polygon with arbitrary segment lengths and angles.
Rectangle	Draw an orthogonal rectangle.
Rotated Rectangle	Draw a rectangle oriented at any angle.
Perpendicular	Draw a line perpendicular to an existing line from any point.
Parallel	Draw a line parallel to an existing line, at a specific distance from the line.
Tangent Arc Point	Draw a line tangent to an arc, touching the arc at the line's midpoint.
Tangent to Arc	Draw a line tangent to an arc, with the line's endpoint touching the arc.
Tangent from Arc	Draw a line tangent to an arc, with the line's start point touching the arc.
Tangent to 2 Arcs	Draw a line tangent to two arcs, with the end points of the line tangent to the arcs.

Single Line

Menu: Insert|Line|Single   **Toolbars:** Standard, Line

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:** Standard, Line

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 (a fly-out from the Line toolbar).
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: TurboCAD 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.

Tip: Instead of choosing Finish, you can have TurboCAD 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.

Tip: To undo the most recently drawn segment of the multiline, open the local menu and choose One Step Back, or press   <Backspace>.

Note: Once a multiline is closed, TurboCAD 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 box, or the Properties toolbar.

Polygon

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 center point to one of the vertices, and the degree of rotation is the angle of a line from the center point to one vertex.

Irregular Polygon

Use the Irregular Polygon tool to draw a polygon with irregular sides and angles.

1. Activate the Irregular Polygon tool.
2. Define a point for the first vertex of the irregular polygon.
3. Define a second point: TurboCAD draws a line segment between the two points.
4. Define a third point. TurboCAD draws a closed polygon, connecting the three points.
5. Repeat step 4, adding sides and vertices to the irregular polygon.
6. After you have defined the final point, right-click in the drawing area and choose Finish from the local menu.

Alternatively, you can enter the length and angle of each new line segment from the current vertex to the new vertex in the Edit Bar. Press <Enter> to define each new line segment. Press <Ctrl>+<F> to finish the irregular polygon.

Rectangle

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 using positive-X values from the starting point, and the B size will be drawn to the right of the starting point. To draw toward the negative-X direction, use negative values.

Rotated Rectangle

Use the Rotated Rectangle tool to create a rectangle oriented at any angle. You create rotated rectangles by defining three points:

1. Define two points to specify the endpoints of a line that forms the rectangle's base.

2. Do one of the following:

- Define a third point to specify the distance of the opposite side of the rectangle from the baseline.
- Type the rectangle's dimensions and angle of rotation into the Edit Bar fields, then press <Enter>.

Perpendicular

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 on the line to the endpoint of the line:

1. Activate the Perpendicular Line tool.
2. Click on an existing line segment to which you want to draw the perpendicular line.
3. Select a point on (or near) the existing line.
4. Define the endpoint of the perpendicular line.

If the point you define is beyond the end of the existing line segment, TurboCAD 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

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. Call this the *source line*. As you move the mouse cursor away from the source line, TurboCAD 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.

Tip: To create multiple parallel lines lock the distance in the Edit bar and then use the mouse to position the new line either above or below the existing line.

Tangent Arc Point

Menu: Insert|Line|Tangent Arc Point   **Toolbars:** Standard, Line

The Tangent Arc Point 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 Tangent Arc Point tool.
2. Click on the arc or circle to which the tangent must be drawn. 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>.

Tangent to Arc

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

1. Activate the Tangent 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.

Tangent from Arc

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

1. Activate the Tangent 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>.

Tangent to 2 Arcs

Menu: Insert|Line|Tangent to 2 Arcs   **Toolbars:** Standard, Line

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

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

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. The following table summarizes the double line tools:

Tool	Function
Single	Draw a double line segment by defining two endpoints.
Multiline	Draw a series of connected double lines.
Polygon	Draw a regular double line polygon with an arbitrary number of sides.
Irregular Polygon	Draw a double line polygon with arbitrary line lengths and angles.
Rectangle	Draw a horizontal double line rectangle.
Rotated Rectangle	Draw a double line rectangle tilted at any angle.
Perpendicular	Draw a double line perpendicular to an existing line.
Parallel	Draw a double line parallel to an existing line.

Subtopics

Double Line Properties
Double Line Separation

Double Line Properties

Double Lines have unique properties that you can set in the Double Line property sheet of the Format|
Properties dialog box. 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, or select and right-click for local menu and choose Properties.

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, 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 separation of a double line (the distance between the two single lines that comprise the double line) is set only in its property sheet. If you stretch or scale a double line, its separation always remains the same. If you want the separation 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 polyline. (If the double line entity is itself contained within a group or block, 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 Properties

Double Line

Menu: Insert|Double Line|Single   **Toolbars:** Standard, Double Line

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

1. Activate the Double Line Single 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:** Standard, Double Line

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: TurboCAD 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.

Tip: You can have TurboCAD 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.

Tip: If you want to cancel the previous line segment, open the local menu and choose One Step Back.

Double Line Polygon

Use the Double Line Polygon tool to draw a regular double line polygon by defining points for the polygon's center and for one of its vertices (corners).

1. Activate the Double Line Polygon tool.
2. Enter the number of sides for the double line polygon in the Sides field of the Edit Bar.
3. Define the centerpoint of the polygon.
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.

Double Line Irregular Polygon

Menu: Insert|Double Line|Irregular Polygon   **Toolbars:**
Standard, Double Line

Use the Double Line Irregular Polygon tool to draw a double line polygon with irregular line segment lengths and angles:

1. Activate the Double Line Irregular Polygon tool.
2. Define the first point of the irregular polygon.
3. Define a second point: TurboCAD draws a double line between the two points.
4. Define a third point. TurboCAD draws a closed polygon, connecting the three points.
5. Repeat step 4, adding sides and vertices to the irregular polygon.
6. To define the last point in the series, choose Finish from the local menu.

Alternatively, you can enter the length and angle of each line segment from the current point in the Edit Bar. Press <Enter> to define each new line. Press <Ctrl>+<F> to finish the irregular polygon.

Double Line Rectangle

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

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, and then press <Enter>.

Note: If you use the Edit Bar method, the A size will be drawn toward the positive-X direction from the starting point, and the B size will be drawn to the right of the starting point. You can draw to the negative - X direction by entering negative values.

Double Line Rotated Rectangle

Menu: Insert|Double Line|Rotated Rectangle   **Toolbars:**
Standard, Double Line

Use the Rotated Rectangle tool to create a rectangle oriented at an angle that you choose. You create a rotated rectangle by defining three points:

1. Define two points to specify the endpoints of the line that forms the rectangle's base. The angle of this line determines the angle of the rectangle.

2. Do one of the following:

▪   Define a third point specifying the distance of the opposite side of the rectangle from the baseline.

▪   Specify the length of side B and angle of the double line rectangle in the Edit Bar, and then press <Enter>.

Perpendicular Double Line

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

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.
4. Define the end point of the line.

Tip: If the point you define is beyond the end of the existing line segment, TurboCAD 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.

Parallel Double Line

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

1. Activate the Double 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, TurboCAD draws a double 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 from the Edit Bar.

Tip: To create a parallel double line that is constrained to be the same length as the source line, right-click in the drawing area with the Parallel Double Line tool active, and then switch on the Keep Length option in the local menu.

Circle/Ellipse

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

The Circle/Ellipse menu item opens a submenu containing tools for drawing circles and ellipses. You can also access these tools from the Circle/Ellipse fly-out on the Standard toolbar. The following table summarizes the Circle/Ellipse tools:

Tool	Function
Center and Point	Draw a circle by specifying its centerpoint and radius.
Concentric	Draw a set of circles that share the same centerpoint.
Double Point	Draw a circle by defining two opposite points on its perimeter.
Tangent to Arc	Draw a circle tangent to another circle or arc.
Tangent to Line	Draw a circle tangent to a line.
Triple Point	Draw a circle by defining three points on the circle's perimeter.
Tangent to 3 Arcs	Draw a circle tangent to three existing arcs or circles.
Ellipse	Draw an ellipse by defining its bounding rectangle.
Rotated Ellipse	Draw an ellipse rotated at any angle.
Ellipse Fixed Ratio	Draw an ellipse, specifying the ratio of the major axis length to the minor axis length.

Center and Point

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

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>.

Concentric

Use the Concentric tool to draw concentric circles, that is, circles that share the same centerpoint. You can create as many concentric circles as you need at one time:

1. Activate the Concentric tool.
2. Define a centerpoint for the circles.
3. Do one of the following:
   - Define a point on the perimeter of the circle.
   - Enter the circle's radius in the Radius field on the Edit Bar and press <Enter>.
   TurboCAD will draw a circle with the specified center and radius.
4. Repeat step 3, drawing as many concentric circles as required.
5. Finish the concentric circles by choosing Finish from the local menu, or by pressing <Ctrl>+<F> if you are working in the Edit Bar.

You can also enter the radius or circumference of each concentric circle in succession in the Edit Bar. Press <Enter> to complete individual circles. Press <Ctrl>+<F> to complete the set of concentric circles.

Tip: To draw a circle concentric to an existing circle, choose the Concentric tool, then use Snap to Arc Center to snap to the center of the existing circle.

Double Point

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 Tangent to Arc

Menu: Insert|Circle/Ellipse|Tangent to Arc   **Toolbar:** Standard, Circle/Ellipse

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

1. Activate the Circle Tangent 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 in it (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 Tangent to Line

Menu: Insert|Circle/Ellipse|Tangent to Line   **Toolbar:**
Standard, Circle/Ellipse

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

1. Activate the Circle Tangent 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 in it (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

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, and then the third point through which the perimeter of the circle will pass.

Tangent to 3 Arcs

Menu: Insert|Circle/Ellipse|Tangent to 3 Arcs   **Toolbar:**
Standard, Circle/Ellipse

Use the Tangent to 3 Arcs tool to draw a circle tangent to three existing arcs or circles. You can specify whether the new circle will include or exclude each one of the existing three arcs or circles.

1. Activate the Tangent to 3 Arcs tool.
2. Use the mouse to select an arc or circle to which you want the new circle to be tangent. Click just inside the perimeter of the arc to include it in the new circle, or click just outside the arc to exclude it from the circle. A small arrow will appear on the each arc, indicating whether it is to be included or excluded.
3. Repeat step 2 with the remaining two existing circles or arcs.

Ellipse

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

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:** Standard, Circle/Ellipse

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 major 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 (minor) 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>.

Ellipse Fixed Ratio

Menu: Insert|Circle/Ellipse|Ellipse Fixed Ratio   **Toolbar:**
Standard, Circle/Ellipse

The Ellipse Fixed Ratio tool lets you create an ellipse by specifying its aspect ratio; that is, the ratio of the length of its major axis to the length of its minor axis. For example, if the ratio is 2 then the major axis will be twice as long as the minor axis (the ellipse will be twice as long as it is wide). The axes of the ellipse will initially be horizontal and vertical; however, you can subsequently rotate the ellipse using editing tools.

1.   Activate the Ellipse Fixed Ratio tool.
2.   Define the centerpoint of the ellipse.
3.   Enter the aspect ratio in the a:b field on the Edit Bar, then lock down this field. To return to the drawing area, click in it or press <Ctrl>+<D>.
4.   Define any point of the perimeter of the ellipse.

Arc Tools

Menu: Insert|Arc   **Toolbars:** Standard, Arc

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.

Note: Arcs are drawn in a counterclockwise direction. This can be changed in Options|Angle.

The following table summarizes the arc drawing tools:

Tool	Function
Center and Radius Arc	Draw a regular arc by specifying its centerpoint and a point on its circumference.
Concentric Arc	Draw a set of concentric regular arcs that share the same centerpoint.
Double Point Arc	Draw a regular arc by defining the endpoints of its diameter.
Tangent to Arc	Draw a regular arc tangent to a circle or arc.
Tangent to Line	Draw a regular arc tangent to a line.
Start/Included/End	Draw a regular arc by defining (1) its starting point, (2) a point on its perimeter, and (3) its ending point.
Start/End/Included	Draw a regular arc by defining (1) its starting point, (3) its ending point, and (2) a point on its perimeter.
Tan to 3 Arcs	Draw a regular arc that is tangent to 3 existing arcs or circles.
Elliptical	Draw an elliptical arc by defining its bounding rectangle.
Rotated Elliptical	Draw an elliptical arc rotated at any angle.
Elliptical Fixed Ratio	Draw an elliptical arc by setting its aspect ratio and defining either axis.

Subtopics

Circumference and Length of Arcs
Combining Edit Bar and Mouse When Drawing Arcs

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 the 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 moves 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, it is likely that you will want to use various combinations of the Edit Bar and the mouse to create them.

Tip: 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 Radius Arc

Use the Center and Radius 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 Radius 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. TurboCAD 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 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.

Concentric Arc

Use the Concentric tool to draw concentric arcs; that is, arcs that share the same centerpoint. These concentric circles are a series of center and radius arcs, created in succession. You can create as many concentric arcs as necessary at one time.

1. Activate the Concentric Arc tool.
2. Define a centerpoint for the arcs.
3. Define a point on the circumference of the arc. TurboCAD will draw a circle with the specified center and radius. A dotted line will appear, extending from the centerpoint of the arc 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. Move the cursor counterclockwise to draw the arc, then click the mouse button to define the ending angle.
5. Repeat steps 3 and 4, drawing as many concentric arcs as required.
6. Once you have all the concentric arcs you need, complete the set of concentric arcs by choosing Finish from the local menu, double-clicking, or pressing <Enter>.

You can also enter the radius (or circumference) and the starting and ending angles (or length) of each concentric arc in succession in the Edit Bar. Press <Enter> to complete individual circles. Press <Ctrl>+<F> to complete the set of concentric circles.

Double Point Arc

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. TurboCAD 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 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 Tangent to Arc

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

1. Activate the Tangent 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 move 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.

Tangent to Line

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

Use the Tangent to Line tool to draw an arc tangent to an existing line.

1. Activate the Tangent 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 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.

Arc Start/Included/End

The Start/Included/End 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 Arc Start/Included/End tool.
2. Define the starting point of the arc.
3. Define a point through which the arc will pass.
4. Define the ending point of the arc.

Arc Start/End/Included

The Start/End/Included tool lets you draw an arc by defining three points. This is similar to drawing a triple point circle, except that the first two points you define determine the points at which the arc begins and ends.

1. Activate the Start/End/Included tool.
2. Define the starting point of the arc.
3. Define the ending point of the arc.
4. Define a third point through which the arc will pass.

Arc Tan to 3 Arcs

Menu: Insert|Arc|Tangent to 3 Arcs   **Toolbar:** Standard, Arc

Use the Tangent to 3 Arcs tool to draw an arc tangent to three existing arcs or circles. You can specify whether the new arc will include or exclude any of the three arcs or circles.

1. Activate the Tangent to 3 Arcs tool.
2. Use the mouse to select an arc or circle to which you want the new circle to be tangent. Click just inside the perimeter of the arc to include it in the new circle, or click just outside the arc to exclude it from the circle. An arrow on each arc indicates whether it is included or excluded.
3. Repeat step 2 with two more existing circles or arcs. TurboCAD draws a circle tangent to the three arcs or circles you have selected. A dotted line appears, extending from the centerpoint of the circle to the cursor.
4. Move the cursor 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.

As an alternative to steps 4 and 5, you can specify the starting and ending angles of the arc in the Edit Bar, then press <Enter>.

Elliptical Arc

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. TurboCAD 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

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). TurboCAD 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.

Elliptical Fixed Ratio Arc

Menu: Insert|Arc|Elliptical Fixed Ratio   **Toolbar:** Standard, Arc

The Elliptical Fixed Ratio tool lets you create an elliptical arc by specifying its aspect ratio; that is, the ratio of the length of its major axis to the length of its minor axis. For example, if the ratio is 2 the major axis will be twice as long as the minor axis. The axes of the elliptical arc will initially be orthogonal; however, you can subsequently rotate the elliptical arc using the editing tools.

1. Activate the Elliptical Fixed Ratio Arc tool.
2. Define the centerpoint of the arc.
3. Type the aspect ratio of the arc into the Ratio field of the Edit Bar.
4. Define any point on the perimeter of the ellipse. TurboCAD 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 to draw the arc, then click the mouse button to define the ending angle.

As an alternative to steps 5 and 6, you can specify the start and end angles of the arc in the Edit Bar, then press <Enter>.

Curve Tools

Menu: Insert|Curve   **Toolbar:** Standard, Curve

The following table summarizes the curve tools:

Tool	Function
Spline	Connects two endpoints; gravitates toward control points along its path.
Bezier	Continuous curve that connects a series of points.
Sketch	Freehand multiline drawn with mouse.

Subtopics

Curve Properties

Curve Properties

Curves have special properties, 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 double-clicking on a curve with the Select tool, or by selecting the curve and then choosing Format|Properties.

Tip: 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.

See Also: Breaking Polygons, Closing Multilines, Curve Property Sheet

Spline Curve

To create a Spline 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 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, 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 <Alt>+<F> to complete the entire curve. This procedure is essentially similar to drawing a multiline.

Bezier Curve

To create a Bezier curve, define points along the intended path of the curve. The curve will pass through the defined points.

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 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 or press <Backspace>.

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 <Alt>+<F> to complete the entire curve. This procedure is essentially similar to drawing a multiline.

Sketch Curve

Menu: Insert|Curve|Sketch   **Toolbar:** Standard, Curve

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 [A]

The features for creating, formatting, and manipulating text in TurboCAD v4 are very 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 Property toolbar (using View|Toolbars) and choose settings from the toolbar. As an alternative, you can set text properties by right-clicking on the Text tool, then changing the properties in the Text property sheet. You can use any TrueType font installed in your system.
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.

Note: To edit text that has already been placed on a drawing use the Selection Tool and double-click the text and then edit the attribute string in the General tab.

See Also: For information about installing TrueType fonts, see your Windows documentation.

Subtopics

Text Properties

Text Height and Angle Options

Text Alignment Options

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 right-clicking on the selection and then choosing Properties from the local menu. The Text property sheet has options for setting font, font style, text height, text color, text mode and text placement style.

Tip: 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.

See Also: 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 line 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 line 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.

Tip: To use Ortho while rotating text, hold down the <Shift> key.

Note: The editing and formatting tools provide other options for rotating and resizing text more precisely after it has been drawn.

See Also: 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, and 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 the insertion point will be at the top-left corner of the text.

Dimension Tools

TurboCAD 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 and datum 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. You can also change the dimension text from the general property sheet.

See Also: Dimension Property Sheets

The following table summarizes the dimension tools:

Tool	Function
Horizontal	Draw a linear dimension showing horizontal distance.
Vertical	Draw a linear dimension showing vertical distance.
Parallel	Draw a linear dimension showing absolute distance between two points.
Rotated	Draw a linear dimension showing absolute distance between two points (the length of a line), with extension lines rotated to any angle relative to the line you are dimensioning.
Datum	Draw a series of dimensions connected to angled leader lines.
Baseline	Draw a series of parallel linear dimensions sharing a common baseline.
Continuous	Draw a series of parallel linear dimensions, each measuring a distance from the end of the previous dimension in the series.
Angular	Draw an 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	Draw a dimension indicating the radius of an arc or circle.
Diameter	Draw a dimension indicating the diameter of an arc or circle.
Leader	Draw a simple line leader with label text.

Subtopics

Manual, Segment, and Entity Dimensioning

Associative Dimensions

Setting Dimension Properties

Manual, Segment, and Entity Dimensioning

Horizontal, vertical, parallel, and rotated 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, a group, or a block insert).

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, deselect 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.

Tip: If you generally prefer one of the dimensioning modes over the others, you can save it in a drawing template.

Warning: If you are using the Rotated Dimension tool, the Segment Dimensioning and Entity Dimensioning items in the local menu will be grayed out until after the first two clicks. See Rotated Dimension further information.

See Also: Horizontal Dimension; Parallel Dimension; Rotated Dimension; Vertical Dimension

Associative Dimensions

Dimensions can be *associated* with 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, or in the attribute string of the Dimension Property sheet, 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, deselect 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 right-click on the selection and choose Properties from the local menu to bring up the Properties dialog.

See Also: Dimension Property Sheets

Horizontal Dimension ⟷

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, or in the attribute string of the Dimension Property Sheet.

Vertical Dimension

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 dimension text, type the text you want in the Edit Bar prior to finishing the dimension. Once the dimension is placed you can override the text in the Property sheet's attribute string.

Parallel Dimension

Menu: Insert|Dimension|Parallel   **Toolbars:** Standard, Dimension

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 the <u>Manual, Segment, and Entity Dimensioning</u>.
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.

Rotated Dimension   ▪

Use the Rotated Dimension tool to measure the absolute length of an entity (the distance between the two points on the entity that are farthest apart). Rotated dimensions are similar to parallel dimensions, except that the extension lines of the Rotated Dimension tool can be rotated to any angle relative to the entity you are dimensioning.

1. Activate the Rotated Dimension tool.
2. Define two points to draw a vector specifying the angle and direction of the dimension's extension lines. Alternatively, you can define the first point, then specify the angle of the vector in the Edit Bar.
3. If desired, change the current dimensioning mode, as described in the Manual, Segment, and Entity Dimensioning.
4. 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.)

5. 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 dimension text, type the text you want in the Edit Bar prior to finishing the dimension. Once the dimension is placed you can override the text in the Property sheet's attribute string.

Note: The Segment Dimensioning and Entity Dimensioning menu items will be disabled in the local menu until after you have completed step 2.

Datum Dimension

Use the Datum Dimension tool to display dimension text along with a horizontal or vertical leader line with an arrow pointer. The leader line is bent at right angles, allowing you to offset the dimension text. Datum dimensions are typically used to display a progression of measurements along the horizontal or vertical axis.

1. Activate the Datum Dimension tool.
2. Define the starting point of the datum dimension: This is the point where the arrow will be drawn. A rubberband line appears, indicating the origin of the dimension.
3. Define a second point to position the dimension text. Depending on the position of this point (in relation to an imaginary 65 line), TurboCAD will draw either a vertical or a horizontal dimension.

By default, a horizontal datum dimension displays the X coordinate of the starting point, and a vertical dimension displays the Y coordinate. You can override the automatic dimension text by typing a value into the Text field of the Edit Bar before finishing the dimension.

Subtopics

Changing the Origin of Datum Dimensions

Changing the Origin of Datum Dimensions

By default, the automatic text of a datum dimension displays the X or Y distance from the Absolute Origin (0,0 in the absolute coordinate system). If you like, you can specify an origin for datum dimensions. This is useful if you would like to display distances relative to the entity that you are dimensioning. For example, if you were placing datum dimensions along a wall to show points along its length, you would want to set the origin by snapping to one end of the wall, then draw a series of datum dimensions relative to this point along the length of the wall. To set an origin for datum dimensions:

1. Activate the Datum Dimension tool.
2. Define the origin point using any snap mode or the Coordinate Fields.
3. Right-click in the drawing area and choose Set Origin.
4. Continue to draw the datum dimension as described in Datum Dimension .

Note: Once you have set the origin for datum dimensions, the origin will be saved with your drawing or template, and remain in effect for that drawing or template until you explicitly change it.

Baseline Dimension

Menu: Insert|Dimension|Baseline   **Toolbars:** Standard, Dimension

The Baseline Dimension tool allows you to create a series of parallel linear dimensions that share the same baseline on one axis. The automatic dimension text displays the distance from the shared baseline to the end of the individual dimension. Before you draw the first baseline dimension in a series, you need to select a previously drawn linear dimension, either end of which can serve as the base for the series. (You would choose the left end, for example, if you were dimensioning to the right.) The previously drawn dimension can be any linear dimension (horizontal, vertical, parallel, or rotated):

1. Activate the Baseline Dimension tool.
2. Click on a previously drawn linear dimension, nearest the end that you want to serve as the baseline for the series. A rectangle appears at the end of the dimension nearest the point where you clicked, indicating the position of the baseline.
3. Define a second point using an appropriate snap mode or the Coordinate Fields, relative to the baseline. This creates a new linear dimension displaying the distance between the baseline and this defined point.
4. Repeat step 2 to create a series of baseline dimensions. You can continue to do so as long as the tool is active.

See Also: Continuous Dimension

Continuous Dimension

The Continuous Dimension tool allows you to create a series of parallel dimensions. Each dimension in the series displays the distance from a baseline on the previous dimension in the series to a defined point. If you are dimensioning from left to right, for example, the baseline of each dimension would be the right end of the previous dimension. Before you draw the first continuous dimension in a series, you need to select a previously drawn linear dimension, which will serve as the base dimension for the series. The previously drawn dimension can be any linear dimension (horizontal, vertical, parallel, or rotated):

1. Activate the Continuous Dimension tool.

2. Click on a previously drawn linear dimension, nearest the end that will serve as the baseline for the first continuous dimension in the series. A small rectangle appears, marking the baseline.

3. Define a second point using an appropriate snap mode or the Coordinate Fields, relative to the baseline. This creates a new linear dimension displaying the distance from the baseline to this defined point. The rectangle is now displayed at the end of the current continuous dimension, marking the new location of the baseline.

4. Repeat step 3 to create a series of continuous dimensions. You can continue to do so as long as the tool is active.

Angular Dimension

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 free-form angle that you create by defining a vertex node and two points.

Subtopics

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, TurboCAD 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 rubberband line will appear, extending from the center of the circle to 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, TurboCAD 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

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. TurboCAD 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:** Standard, Dimension

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. TurboCAD draws a dimension through the center of the circle or arc to the 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 is positioned outside the circle.

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

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. TurboCAD 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.

Viewport

New in TurboCAD v4 is the ability to insert a Named View as an element of the drawing. This is useful to create callouts and other forms of detailing.

To use Insert|Viewport, you must have previously created a Named View that matches the area you now want to display as an inserted view. When you select Insert|Viewport, you will be prompted to draw a rectangle on the drawing. Place the rectangle where you want the inserted view to appear. When you complete the rectangle, a list of available named views will appear in a dialog. Select a named view, click Go To, and click Close.

If you make changes to the part of the drawing depicted in the inserted view, the inserted view will update to match.

To select a Viewport either use a window selection, or click near the edge of the Viewport.

Path Hatching

Menu: Insert|Path Hatching   **Toolbar:** Standard

In TurboCAD v4 you can place hatch patterns or solid color fills as independent objects using the Path Hatching command. As you draw the path, a thick purple line shows the path. To use:

1. Choose Insert|Path Hatching.
2. Draw the boundary of the hatch pattern. If the hatch boundary follows the perimeter of an existing entity, use the appropriate snap commands (from the local menu or by typing SEKEs) to conform to the entity.
3. To finish, right-click to summon the local menu, and select Close.

Insert File

Use this to insert a TurboCAD or other CAD file into an open drawing:

1. Choose Insert|File to display the Insert File dialog.
2. Choose the type of CAD file you want to import from the List Files of the 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, then click the OK button.

The complete contents of the file will be inserted into your drawing, alongside any existing objects. The file types that can be inserted are the same as those that can be opened in the Open dialog.

Tip: To see the entire file after it has been inserted, choose View|Zoom|Extents.

See Also: Supported File Types

Insert Picture

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 bitmap pictures or edit their contents.

To insert a WMF or bitmap picture:

1. Choose Insert|Picture to display the Open 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.

Tips: The picture you insert is actually an OLE object, so you can double-click the picture in TurboCAD and then edit it using Paint (or whatever bitmap editor is registered in your Windows system).

To select a previously inserted picture, click along an edge instead of inside the picture.

Bitmap images cannot be plotted on old pen type plotters.

See Also: Insert Object, Insert File

Insert Object

The Insert Object command lets you insert OLE (Object Linking and Embedding) objects from other Windows applications into your TurboCAD drawing. You can edit the object from within TurboCAD 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 option button (Create New is the default).

See Also: Edit Links, Edit Object, Paste Special

Subtopics

Create New

Create from File

Embedded and Linked OLE Objects

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 in the Display As 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 TurboCAD drawing.

You can edit the object using the tools of the source 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 Display As 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 TurboCAD drawing. To edit the object in TurboCAD 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 TurboCAD 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: Edit Links

Warning: If you move a TurboCAD file containing a linked object to another system, you will need to move any linked files along with the TurboCAD 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.

Formatting Objects

The 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 or blocks.
- Explode groups, blocks, 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.

Subtopics

Setting Properties

Property Toolbar

Formatting Text

Text Placement Options

Groups and Blocks

Format Menu Commands

Setting Properties

Every object in TurboCAD 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, style, 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 many other properties associated with specific classes of entities. Dimensions, for example, have a large number of properties that apply specifically to them, located on the Format, Advanced Format, and Advanced Text property sheets. Points, 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.

Note: The Properties dialog is context-sensitive, meaning 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 will not see the Curve property sheet because no curves are included in the objects you have selected.

See Also: Properties Dialog

Subtopics

Setting the Default Properties of a Tool
Setting the Properties of Selected Objects

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 the dialog will display the set of property sheets 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 new entities.

Note: Styles in TurboCAD are used to preselect tool properties, so the user can quickly access a predefined drawing style (blue thick dotted lines on layer 24, for example). TurboCAD does not use styles to adjust the properties of existing entities or objects in a drawing.

See Also: Layers Property Sheet, Properties Dialog, Property Toolbar

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.

Note: In TurboCAD v4 you cannot apply a style to a selection. The Style pull-down box is accessible only when you are setting the default properties of a drawing tool.

The Property toolbar consists of eight list boxes, two fly-out toolbars, and three tools. 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.

Pen 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 Pattern. Use this list box to choose a fill pattern for closed entities.

Pen Width. Use this list box to choose the line width.

Font. Use this list box to choose a font for text in the drawing. All TrueType fonts installed on your computer will be available.

Text Height. Use this list box to set the height of text.

Text Font Attributes. Use the options in this fly-out toolbar to set text to **Bold,** *Italic,* Underline, or S̶t̶r̶i̶k̶e̶t̶h̶r̶o̶u̶g̶h̶. The text will be plain if none of these options are selected.

Text Placement. Five placement styles are available in this fly-out toolbar: Standard, Text Box, Hidden, Scalable, and Flexible. These placement styles are defined in Formatting Text.

See Also: Layers Property Sheet, Properties Dialog

Note: New in TurboCAD v4 is the ability to set the properties of all drawing tools by changing the Property toolbar. This means that if no entity or object is selected in the drawing, changing the settings on the Property toolbar will effect every drawing tool used thereafter. If you want Property settings to work as they did in TurboCAD 3.0, where drawing tool properties are each set independently, make sure the option "Update All Styles" (Menu: Options|Toolbars and Menu) is not selected.

Note: Changing a selected object's properties will have no effect on the default properties for new objects drawn.

Formatting Text

You can format text in TurboCAD using either the Property toolbar or the Text property sheet. The Text property sheet will be covered in [Format Menu Commands](#) .

You can use text tools 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 Standard toolbar, and then choose Text settings on the 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 Property toolbar.

Text Placement Options

TurboCAD v4 offers a variety of text placement options. As shown in the Text Property dialog, they fall into four basic categories.

Font Attributes. These are the characteristics of text common to virtually all Windows programs. The elements are Font Name, Font Style, Height, and Color.

Text Mode. Text may be inserted into a drawing in one of three ways: Standard (always proportional even if resized), Scalable (can be re-sized non-proportionally), and Flexible (pen, brush and slant options available.)

Text Effects. Text may be set to Strikethrough, Underline, Text Box, Hidden, All Caps.

Justification. Any block of text may be aligned to Left, Center, or Right.

See Also: Manipulating Text, Text, Text Property Sheet

Groups and Blocks

TurboCAD drawing objects can be combined into groups or blocks. The drawing objects can be entities, such as lines circles and arcs, or they can be other groups and blocks. You can include groups in blocks, and you can include blocks in groups. TurboCAD treats a block or a group as a single object for purposes of selecting and editing. There are two practical differences between groups and blocks:

▪   When you create a block you store it in your drawing's block library. This allows you to conveniently drag and drop the block into the current drawing, or into another drawing, from the Blocks palette (View|Blocks). Each "copy" of the block that you place in your drawing is called a block insert or block insertion.

▪   Each group contains its own drawing data. This means that making a copy of a group increases the size of your drawing by the size of the group. Block inserts, on the other hand, are references to a single object in a drawing's block library. You can insert multiple block inserts from the Block Palette into your drawing without significantly increasing the size of your drawing.

To break up a block insert or group into its constituent parts, select it, then use the Format|Explode command.

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

Because groups and blocks can themselves contain groups and blocks, you can think of them as complex *hierarchical* structures. This type of structure can be represented as a tree, as shown in the following diagram:

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

See Also: Create Block, Create Group, Explode, View Blocks

Format Menu Commands

The following table summarizes the contents of the Format menu:

Command	Function
Properties	Edit the properties of selected objects or the default properties of the current tool.
Create Hatch	Draw a hatch pattern over selected objects.
Create Block	Define all objects in the selection as a block and save it in the block library.
Create Group	Define all objects in the selection as a group.
Explode	Break up an object into its constituent parts.
Send To Back	Send selected objects to the back of the object stack on their layer.
Bring to Front	Bring selected objects to the front of the object stack on their layer.
Align	Line up all selected objects on the left, right, top, bottom, vertical center, horizontal center, or along a line that you specify.

Properties Dialog

Menu: Format|Properties   **Toolbar:** Standard

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.

See Also: Select, Property Toolbar

Subtopics

General Property Sheet of Properties Dialog
Pen Property Sheet
Brush Property Sheet
Curve Property Sheet
Double Line Property Sheet
Dimension Property Sheets
Format Property Sheet
Advanced Format Property Sheet
Advanced Text Property Sheet
Point Property Sheet
Text Property Sheet
Block Insertion Property Sheet
Group Property Settings
Viewport Property Sheet

General Property Sheet of Properties Dialog

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 new 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. For groups and blocks, the Attribute text field holds the group or block name. This field is also used to hold the text string for Text and Dimension entries.

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 (Options|Layers).

Style. A style is a set of saved property settings. You can select and edit existing styles.

Subtopics

Selecting, Editing, and Creating New Styles

Selecting, Editing, and Creating New Styles

To select a style that has already been defined, click the down arrow on the right side of the Style text box and 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.

Note: 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.

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.

Note: 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. The available line patterns are the styles most commonly used in the various drafting and design professions. The exact look of any particular pen pattern depends on the settings in Dash Scale, Width Scaling, and Width. In addition to specific patterns, By Layer and By Block settings are also available. If the pen pattern is set to either By Layer or By Block, the pattern can change as the setting for the particular layer or block changes.

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. If you select By Block, you will be able to modify the pen color of any object drawn after it becomes part of a block. If By Block is not the color option, the object's pen color will stay even if it is part of a block.

Note: You can create new colors using the Color Palette property sheet (Options|Color Palette).

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. (To set the unit of measure for the Pen Width, use the Advanced Units property sheet.) Zero width uses the width of one screen pixel, and will print at one unit of the printer's available resolution (a 300 dots-per-inch printer will print a zero width line at 1/300".)

See Also: Advanced Units Property Sheet, Layers Property Sheet

Brush Property Sheet

The Brush property sheet lets you set fill properties of objects, as well as the properties of hatch patterns. Fills and hatches appear only on closed entities, such as circles and polygons, and Flexible text.

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 or By Block if you want the color to match the default color of the object's layer (set in the Layers property sheet) or any setting applied to a block containing the object.

Note: You can create new colors using the Color Palette property sheet (Options|Color Palette).

Hatch. Use the Hatch controls to control the scale and angle of the fill pattern. The default scale of the pattern is 1(inch or mm, depending on your setup), 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.

Origin. New to TurboCAD v4 is the ability to modify the hatch origin. By default, hatch patterns use the drawing's origin point (0,0 on the coordinate grid) as a point of reference. If you want to modify the placement of hatches without changing their angle, you can modify the point of origin used to place the hatch. *World* refers to the drawing's origin point, *Entity* refers to the object being hatched. A 0,0 Entity origin refers to the lower left corner of the object. You can offset an origin point for the hatch to either the X (horizontal) or Y (vertical), or both.

Draw Mode. Choose an option button in this group to specify how hatch patterns are drawn over overlapping objects. Choose the Winding option if you want the hatch pattern to cover areas where filled entities overlap. Choose Alternate if you want the hatch pattern to be drawn on every other overlapping layer, creating an alternating pattern.

See Also: Color Palette Property Sheet, Property Toolbar

Curve Property Sheet

The Curve property sheet is visible when a Bezier, Spline, or Sketch (freehand) curve entity is selected, or when a curve tool is active and no entities are selected.

The Curve property sheet contains the following controls:

The number of segments between adjacent control points. Curves are composed of a number of minuscule line segments drawn between control points. Bezier curves intersect the control points; Spline 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 center point 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 start point and end point of double lines are open or closed.

See Also: Double Line Tools

Dimension Property Sheets

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.

Subtopics

Parts of a Dimension

Parts of a Dimension

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. Exterior dimension lines can be supplemented with an optional interior line.

The three dimension property sheets contain options that let you precisely format dimension lines, extension lines, and text:

Format. Lets you format the shape of dimension arrows and set the position of dimension text relative to the dimension line and extension lines.

Advanced Format. Gives you precise control over the drawing of dimension lines and extension lines.

Advanced Text. Gives you precise control over the appearance of dimension text.

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), Tick (a simple angled line), Solid Dot, or Hollow Dot.

Size. Click in this control to choose a size for the length of the arrowhead, (or diameter of the dot) 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 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. (Setting height to paper units will allow the option of using points for text measurement units.)

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.

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, TurboCAD will decide whether to place dimension text inside or outside the extension lines.

Advanced Format Property Sheet

The Advanced Format property sheet includes options for drawing the dimension and extension lines, as well as an option for scaling the size of dimensions relative to your drawing. If you're not sure what any of the terms mean in this section, read the section Parts of a Dimension .

The Dimension Line group includes the following controls:

Force Interior Line. Check this box if you want the dimension to have an interior dimension line even if the text is outside the extension lines. If you leave this box unchecked, TurboCAD will draw the interior dimension line only if the dimension text is inside the extension lines.

Do Not Draw. Click First Line if you don't want TurboCAD to draw the first dimension line; click Second Line if you don't want TurboCAD to draw the second dimension line. Click Outside Extension if you don't want TurboCAD to draw the outside extension line.

Baseline Increment. Type a number in this box to set the distance between each dimension in a series of baseline dimensions.

Color. Click in this box and choose a color if you want the dimension lines to have a different color to distinguish them from the other parts of the dimension.

The Extension Line group includes the following controls:

Extension. The number in this box determines the length of the extension line segments that extend outward beyond the dimension lines. You can enter a value in the box to specify the length of these segments.

Offset. Type a value in this box if you want to specify the distance between the extension lines and the object that is being dimensioned.

Do Not Draw. Click First Line if you don't want TurboCAD to draw the first extension line; click Second Line if you don't want TurboCAD to draw the second extension line.

Color. Click in this box and choose a color if you want the extension lines to have a different color to distinguish them from the other parts of the dimension.

At the bottom of the property sheet you will find a *Dimension Size Scale* text box. This box controls the size of the dimension text and arrows relative to the rest of your drawing. If you use the default setting of 1, the dimension text will be printed at the size specified in the Height box on the Format property sheet.

Tip: Choose a smaller scale if you want to dimension details in the interior of your drawing.

See Also: Parts of a Dimension

Advanced Text Property Sheet

The Advanced Text property sheet contains controls for precisely formatting the appearance of dimension text, and for controlling the display of tolerances. (Tolerances are a number indicating the amount of allowable deviation from the distance shown in the dimension.)

The Appearance group contains the following controls:

Text Gap. Type a number in this box if you want to specify the distance between the dimension text and the dimension line.

Round Off. Type a number in this box to indicate the decimal place to which the dimension text will round off. If you type a value of 0.1, for example, the text will be rounded off to tenths of a unit. A value of 0.001 will generate a round off at thousandths of a unit. Round Off and Trailing Zeros, in this property sheet, and Units and Scale|Precision (in the Options menu) are closely linked.

Linear Measurement Scale. The default value for linear measurement scale is 1, which means, for example, that a dimension will display 1 inch if it is dimensioning a distance of 1 inch in World units. The Linear Measurement Scale control allows you to change the scale of the value displayed by the dimension text relative to World units: If you enter a value of .1, for example, the dimension will display a value of .1 inch when dimensioning a distance of 1 inch in World units.

Tip: You will probably want to leave Linear Measurement Scale unchanged, unless you have an inset in your drawing that uses a different scale than the rest of your drawing.

Color. Click in this box to choose a color for the dimension text.

Append Units. Check this box if you want the dimension text to display its unit of measure.

Trailing Zeros. Check this box if you want trailing zeros to appear in the dimension text. For example, if a dimensioned line is exactly 2.34 units long, Precision (Options|Units and Scale|Precision) is set to 4, and Round Off is set to 4, the dimension text will read 2.3400.

Trailing Zeros, Round Off and Precision are interrelated, and need to be considered as a group when establishing dimension settings. If Round Off is less than Precision, dimension text may not accurately reflect the exact measurement. If Round Off is greater than Precision, then Trailing Zeros will show the additional available precision if Trailing Zeros is set to a value equal to Precision. For most applications, Precision and Round Off should be set to the same level (a Precision of 4 is the same as a Round Off of 0.0001). Trailing Zeros should then be used if necessary to display the level of precision in use. Trailing Zeros is not applicable when fractions (Options|Units and Scale|Format:Fractional) are in use.

Tolerance is a notation showing the allowable difference from the design. The Tolerance group contains the following controls:

Append. Check this box if you want the dimension text to include a tolerance. The other controls in the group are available only if this box is checked.

Relative Height. Type a value in this box to set the height of the tolerance relative to the rest of the dimension text. If you type a value of .5, for example, the tolerance will be half the height of the rest of the dimension text.

+ve. Type a value in this box to set the amount of positive tolerance.

-ve. Type a value in this box to set the amount of negative tolerance.

Point Property Sheet

The Point property sheet 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 TurboCAD 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). These four styles are defined within the font, and are not available for every TrueType font.

Color. Click in this box and scroll to select a color for the text from the color palette.

Mode. Three text placement modes are available in TurboCAD v4: Standard (always proportional even if resized), Scalable (can be re-sized non-proportionally), and Flexible (pen, brush and slant options available.)

Obliquity. The Obliquity option is active if Flexible is the current text placement mode. Think of Obliquity as slant. The numbers are for degrees of slant, i.e., 1 is one degree of slant to the left, -1 is one degree of slant to the right.

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: If you change the justification of existing text, the change will have an effect only if the existing text has multiple lines.

Tip: 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.

Block Insertion Property Sheet

This property sheet will be available if you are formatting the properties of a block insertion. It is not used for formatting the default properties of any tool. This property sheet has settings in three dimensions, specifically for dealing with blocks imported from TurboCAD 3D.

The Block Insertion property sheet contains the following controls:

Position. Shows the location of the selected block's insertion point on the X, Y and Z (for 3D) axes. Changing any of these values will move the block to a new location.

Rotation. Specify values in these fields to rotate the 3D block insertion around its X (vertical), Y (horizontal), and Z (depth) axes.

Scaling. Specify values in these fields to scale the 3D block insertion on its X, Y, and Z axes.

Group Property Settings

Items saved as a group have two adjustable properties, Attribute (the name of the group), and Layer. To rename a group or to change its layer, double-click on a group, or select Properties from the local menu when the group is already selected.

The default property settings of the line tool used to create the group will be the default properties for the group when created.

Viewport Property Sheet

Property settings are available for any viewport inserted into a drawing. To access the property sheet, double-click on the viewport boundary, then click on the Viewport Tab when the Property dialog appears. You may select a new view for the viewport from this property sheet.

Create Hatch

The Create Hatch command applies a hatch pattern to selected entities.

1. To set the hatch pattern you want to use, 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. Or, follow these steps using the Properties toolbar prior to selecting the objects to hatch.

2. Formatting a hatch pattern is essentially the same as formatting a fill pattern, so you can refer to Brush Property Sheet for a description of the property sheet options. Choose the pattern and other options you need (to change Hatch, Origin, or Draw Mode options, you need to access the property sheet.)

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: 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.

Note: It is important to select the objects to be hatched after setting the properties, so that you do not change the properties of those objects. The hatch obtains its properties from the default settings, not the settings of the selected objects.

See Also: Brush Property Sheet

Create Block

Use the Create Block command to combine any currently selected objects into a block. Objects can include drawing entities, pictures, OLE objects, groups, and blocks.

1. Use any selection method to select a set of objects. You may draw a selection rectangle with the Select tool, or <Shift>+<Click> on items to add them to a selection set.
2. Click the Create Block tool or choose Format|Create Block from the menu. If Options|Auto-Naming| Blocks|Prompt For Name is checked, the Create Block dialog appears.
3. Type a name for the new block in the Block Name text box. If you wish to leave a copy of the block at the current location in the drawing, check the Insert at Current Location check box. If you do not check Insert at Current Location, the selected objects will be removed from the drawing and will only be available in the Block palette.
4. Press <Enter> or click OK to store the block.

The next time you open the Block palette using View|Blocks, the new block will be on the palette.

Note: The block will take on the default layer of the Line properties. To select the layer on which the block will appear, right-click the Create Block tool (Standard toolbar) to bring up the Line properties.

See Also: Create Group, Groups and Blocks, Explode, View Blocks

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, groups, or blocks.

2. Click the Create Group tool or choose Format|Create Group from the menu.

Note: The group will take on the default layer of the Line properties. To select the layer on which the group will appear, right-click the Create Group tool (Standard toolbar) to bring up the Line properties.

See Also: Create Block, Groups and Blocks, Explode

Explode

Menu: Format|Explode   **Toolbar:** Standard

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 block that contains a block, a group, a line segment, and some text. After exploding the block, you could then explode its constituents: the group into its parts, the block into its parts, and the text into polylines.

In TurboCAD 3.0 it was only possible to explode one object at a time. TurboCAD v4 allows you to explode all selected objects. The explosion is still one level deep, to allow control over objects with multiple levels of association.

Tip: You can use Explode to break hatch patterns into constituent entities, to break text into polylines, and curves into lines. 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.

See Also: Create Block, Create Group, Groups and Blocks, View Blocks

Bring to Front and Send to Back

In a TurboCAD 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.

TurboCAD 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: 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.

Tip: If you want to systematically place some categories of objects in front of others, use the redraw order property of layers.

See Also: Layers Property Sheet

Subtopics

Bring to Front
Send to Back

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

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 TurboCAD draws a frame around the objects you have selected, you can see the extents of the bounding box.)

The following table summarizes the Align commands:

This Command…	Aligns Objects…
Align Bottom	Along the bottom of the bounding box.
Align Middle	Along a virtual horizontal line passing throughout 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 throughout the horizontal center of the bounding box.
Align Right	Along the right edge of the bounding box.
Align Along Line	Along a construction line at a position and angle that you specify.

Align Bottom

Menu: Format|Align|Bottom   **Toolbars:** Align, Standard

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. TurboCAD will create a rectangular frame around the selection.
2. Choose the Align Bottom command from the menu or toolbar.

Align Middle

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. TurboCAD will create a rectangular frame around the selection.
2. Choose the Align Middle command from the menu or toolbar.

Align Top

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. TurboCAD will create a rectangular frame around the selection.
2. Choose the Align Top command from the menu or toolbar.

Align Left

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. TurboCAD will create a rectangular frame around the selection.
2. Choose the Align Left command from the menu or toolbar.

Align Center

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. TurboCAD will create a rectangular frame around the selection.
2. Choose the Align Center command from the menu or toolbar.

Align Right

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. TurboCAD will create a rectangular frame around the selection.
2. Choose the Align Right command from the menu or toolbar.

Align Along Line

The Align Along Line tool differs from the other Align tools, in that it aligns the selected objects relative to a line that you place at any position an angle in the drawing space.

1. Select all the objects that you want to align, using any selection method. TurboCAD will create a rectangular frame around the selection.
2. Choose the Align Along Line tool from the menu or toolbar.
3. Click the mouse at the point where you want to place the virtual line. TurboCAD will draw a line through the point of the mouse click. (This line behaves like an angular construction line.)
4. Define a second point for the line to pass through: The two defined points determine the angle of the line.

The virtual line disappears as the selected objects are aligned along it.

Customization and Data Management

The Tools Menu provides four features for TurboCAD v4 Professional: Customize, Database, Run Scripts and Measurement. The Tools Menu has three features for TurboCAD v4: Customize, Run Scripts and Measurement. Run Scripts and Measurement have the same functionality in both products.

Customize for TurboCAD v4 Professional allows you to change all elements of the user interface to meet your specifications. You can create new toolbars, change the TurboCAD menu, or change/add keyboard shortcuts. Customize for TurboCAD v4 will only allow you to load different predefined user interfaces.

The Database tools on TurboCAD v4 Professional can be used to create reports using drawing data and other information that you provide.

Use the Run Scripts dialog to run scripts written for TurboCAD. There are several very useful scripts located in the scripts folder.

Use the Measurement dialog to gather information about objects in your drawing. You can measure distance, area, angles, or coordinates in any drawing unit.

Customize, Database and Measurement are new features in TurboCAD v4 and v4 Professional. Experienced TurboCAD users will especially appreciate the additional power and versatility these commands offer.

Note: Database and complete Customize features are only available in TurboCAD v4 Professional.

The Customize dialog in TurboCAD v4 Professional has four tabs: Toolbars, Menus, Keyboard, and Setup. Unless you separately save the user interface changes you make, they become a permanent part of the TurboCAD default user interface. If two or more users share access to one copy of TurboCAD, you can save a set of interface settings for each user by name.

Tip: If you use TurboCAD at more than one screen resolution, you may create a separate custom desktop for each resolution. TurboCAD has different default interfaces for different resolutions. You will need to restart TurboCAD once the screen resolution has been changed in order to load the default for that resolution.

Note: The commands in the Customize dialog do not affect program and drawing settings (the various defaults such as the type of coordinate system in use, grid size, color settings, etc.)

See Also: Options Menu and Dialogs

Subtopics

Customization Options
Measuring Objects
Run TurboCAD Scripts
Database Management in TurboCAD
Tools Menu Commands

Customization Options

Almost every aspect of the TurboCAD v4 Professional user interface can be modified by the user. The Customize dialogs provide a variety of commands and tools for this purpose. Specifically, you can:

- Create new toolbars with varying operational styles
- Create new menus
- Create keyboard shortcuts for TurboCAD commands
- Load and save the alternative configuration files used to store your customizations.

No two experienced TurboCAD users approach the program in the same way. These features make it possible to reorganize the desktop to match individual work styles.

Note: TurboCAD v4 users can only load predefined interface configurations.

Subtopics

Why Change the Desktop?

Creating New Toolbars

Create new menus

Create keyboard shortcuts for TurboCAD commands

Manage configuration files

Why Change the Desktop?

When the topic of customization comes up, some people say "Why bother? Just learn to use the program the way it comes out of the box." Consider the following three scenarios, each offering a valid reason to customize the TurboCAD desktop.

Limited need. A set designer uses TurboCAD to create diagrams for outdoor stage shows. The tools and commands she needs most are a relatively small subset of the program's features. She only uses three line tools and one circle tool, and generally ignores the other entity tools. After a while, the designer realizes she can remove all the existing toolbars on the desktop, replacing them with one toolbar along the top, making more room available for viewing the drawing. The three line tools and one circle tool she uses can be combined into one fly-out toolbar; three important tools now occupy one icon width on her new toolbar.

Preference for the keyboard. In this era of Windows icons and toolbars, a facilities manager who misses DOS still prefers to use keyboard shortcuts as much as possible. He uses [Alt]-[F4] to close the program, and knows that the [Alt]-I, L, S keystroke sequence will select the Single Line command. To him, to remove your hand from the keyboard and not draw a line is a waste of time. The ability to link TurboCAD commands to keystrokes will not only save this user time, but decrease interruptions in the creative process.

Training new users. A network support manager for a Fortune 500 company manages a group of 25 network technicians. Each technician must maintain accurate diagrams for their areas of responsibility. The manager creates custom menus and toolbars for each technician's copy of TurboCAD, increasing staff productivity and shortening learning time.

Creating New Toolbars

You may create new toolbars using almost every command and tool in TurboCAD. You start a new toolbar by selecting Customize from the Tools menu. Select which style of toolbar you will create (Standard, Tabbed, or Fly-out). Drag the New button out of the Customize dialog and onto the desktop. After providing a name for the new toolbar, use the Categories list to find the specific tools or commands you wish to assemble into a toolbar. The toolbar icons for each command will appear in the Buttons field.

Select a category, then click on a button to see its description at the bottom of the dialog. If you want to add the button to your new toolbar, drag it into the toolbar.

Three operation styles are available for new toolbars. *Standard* arranges buttons in a single row along the length of the toolbar. *Tabbed* allows two or more sets of tool icons to share a single toolbar; a tab is attached to each set. *Fly-out* can be used to link a set of related tools; only the first tool in the set is visible until you place the cursor on the tool then press and hold the first mouse button to display the rest of the tools in the set.

One or more Fly-out toolbars may reside in a Standard or Tabbed toolbar. If you create a toolbar with Tabbed tools, all the tool icons must belong to one of the tab groups. If you want to mix toolbar styles (a fly-out inside a tabbed toolbar, for example), each style of toolbar needs to be created separately first. Then you can merge one style into another as desired. See Creating a Mixed Toolbar for detailed instructions

Note: Most of the drawing tools in TurboCAD use fly-out toolbars. The first item in a fly-out group always has a small yellow triangle in the lower right corner of the icon.

The existing pieces of the TurboCAD desktop are saved as a unit in a configuration file. As you create new toolbars, they should be saved into a different configuration file, to keep them separate from the existing desktop configuration. Save new toolbars, menus, and keyboard shortcuts using the Save In button. To create a new configuration, click the list box arrow beneath the Save In button, type a name for the new configuration, then press the Save In button. The toolbars or other customizations will be saved to the new configuration file.

Tools Menu Commands   contains detailed procedures for creating each type of toolbar.

Create new menus

Custom menus may be created using the Tools|Customize|Menus dialog. To create a new menu, the steps are:

1. Open the Customize Menus dialog.
2. Select the top-level item *[Menu Bar]* in the Change What Menu list.
3. Click on the Add Submenu button.
4. Name the new menu and Click OK.

Once you have created a new menu category, you may add commands to it, as follows:

1. Choose the command to be added, by selecting its original menu location from the Category list, then by selecting the actual command.
2. In the Change What Menu list, select the menu which will receive the new command. Usually this is the new menu category defined above, but you could add a command to an existing menu.
3. Select a location in the new menu list from the Position On Menu list box.
4. Optionally, rename the command using the Name On Menu text box.
5. If you want to create a keyboard shortcut to the new menu item, place an & symbol before the character you select as the shortcut. This character will be identified by an underline when you finish the menu.
6. The Position on Menu command is used to set the position of each new command.
7. Click the Add button to accept the change.

New menus, and changes to existing menus, should be saved in a separate configuration file using the Save In command. This allows you to keep the existing TurboCAD desktop separate from custom desktops you establish.

Create keyboard shortcuts for TurboCAD commands

New in TurboCAD v4 Professional is the ability to create keyboard shortcuts for almost every command and tool. Keyboard shortcuts may be created from:

- Any single alphanumeric keystroke
- The combination of CTRL, SHIFT, or ALT and an alphanumeric keystroke
- A Function Key
- The combination of CTRL, SHIFT, or ALT and a Function Key.

Many new keyboard shortcuts have already been configured in TurboCAD. If the shortcut you want to use is already taken, it will appear in the "Currently Assigned To:" section of the Customize Keyboard dialog, just below the "Press New Shortcut Key" box.

Tip: It is possible to reassign existing keyboard shortcut to new commands, and to give commands more than one keyboard shortcut.

The basic steps to creating a new keyboard shortcut are:

1. Select the command being linked to a shortcut, using the Categories and Commands list boxes.
2. Click in the "Press New Shortcut Key" text box, and type the character or keystroke combination you want.
3. Look in the "Currently Assigned To:" section of the dialog to see if the shortcut you have selected is available.
4. If available, accept the new shortcut by clicking the Assign button.
5. Save all changes to a new configuration file with the Save In button.

Manage configuration files

Each of the three customization options allows you to save the changes to a special configuration file. The fourth option in the Configuration dialog, Setup, allows you to manage the configuration files you create.

Before you start customizing, you should give thought to naming the configuration file or files. If you switch between two or more screen resolutions, for example, you could create customized toolbars and menus for each screen resolution (or use the configuration files already supplied for each resolution). In this example, you could name a configuration file for each resolution.

Setup allows you to save all customizations in one file, or to separate them by function (only the toolbar customizations, for example). Generally speaking, it is best to keep Load All Sections and Save All Sections options checked, instead of segmenting the various customizations.

Note: Both TurboCAD v4 and TurboCAD v4 Professional use the above dialog.

Measuring Objects

The Measurement options make it easy to gather information about objects in the drawing. Tools are available to find:

- Coordinates of the point
- Distance
- Angle
- Area

To use any of the measurement commands, start by displaying the Measurement palette. Select View| Selection Info or click the Measurement Info button on the View toolbar. Then select the measurement command you need from the Standard toolbar or Tools|Measurement.

Use snap tools as necessary to trace the boundaries of the objects you are measuring, or select the By Entity or Single Entity options from the local menu to select boundaries based on entities. A purple tracking line will follow as you identify coordinates or entities. Select Finish from the Local Menu; the results will be displayed in the Measurement palette, in both World and Paper coordinates.

To copy the contents of the Measurement palette to the Windows Clipboard, press the Copy button near the top of the palette. You may then paste the information into any word processor, text editor or other program that will accept text.

When using any measure command, the local menu (available with a right-click of the mouse) lists alternative units of measurement, as well as the usual options of Cancel, Close, One Step Back, and Local Snap.

Run TurboCAD Scripts

TurboCAD v4 is a fully programmable CAD system. Scripts are programs a TurboCAD user or an entrepreneur can write to automate specific tasks or to create custom features for specific applications. Its built-in BASIC interpreter allows programmers to create scripts that can access most of TurboCAD's functionality via TCAPI40.DLL.

You can create scripts using any text editor, word processor, or the new on-line debugger, then run the script directly from TurboCAD using the Tools|Run Script command. You can attach scripts to your drawing using the Scripts property sheet in the Options menu. Once scripts are attached to your drawing they appear in a submenu off of the Run Scripts menu item on the Tools menu, and as buttons on the Script toolbar.

Advanced programmers can create their own DLL's in any language capable of calling TurboCAD's exported functions.

For detailed information on scripting, TurboCAD's Enable BASIC language or TCAPI40 functions, see the "Programming TurboCAD with Enable Basic" manual included in the box with TurboCAD v4 Professional, or on the CD with TurboCAD v4.

See Also: Run Scripts, Scripts Property Sheet

Database Management in TurboCAD

As you draw, you see entities and objects on the screen. But within the program, these items are elements of a database. For example, to record a line as part of a drawing, TurboCAD must save information about line width, color, coordinate locations of each endpoint, line type and associated settings, and other information. If this information were stored as a list of items in a table, the categories would be *field names;* the items of information for each line would be data entries in a field.

The database management features in TurboCAD v4 Professional allow the user to add information to objects and to create reports using this information. A drawing of a bolt, for example, can have information about that bolt attached to it. If you are familiar with any of the popular database management programs in the Windows environment, you will quickly recognize the structure and possibilities in the TurboCAD database.

Consider the following two scenarios if the idea of using database management techniques with a CAD program is not clear to you.

Facilities Management. A facilities manager uses TurboCAD to plan office layouts and to track furnishings. She keeps a current drawing for each section of the building, showing every item assigned to each office. Every item in the drawing has additional data added, such as Name of Item, Manufacturer, Price, and Assigned To. As necessary, the facilities manager can generate reports from the information in the drawing, such as total number of chairs assigned, or a list of items by manufacturer.

General Construction. An independent home builder draws plans for every new home he builds in TurboCAD. Over time he had build up a variety of symbols for the common items he uses as he builds homes: sheet rock, plumbing fixtures, etc. Each symbol carries with it field names such as Item Name, Cost, and Supplier. As he draws using symbols, he enters specific data for each item. If he draws new items not already represented by a symbol, he adds fields names and data as he draws. When finished with the drawing, he uses the TurboCAD Database|Report command to create a Bill of Materials, item counts, and other reports.

Additional information can be associated with any object or entity at any time, but it is most productive to plan in advance how to take advantage of the database management capabilities in TurboCAD. Such planning would include determining what pieces of information should be tracked and what to call the various pieces of information. Additional pieces of information can be added to individual entities or objects, as well as to groups, blocks and symbols. You can add descriptive information, such as Part Number or Item Name, or information which can be summarized, such as Cost.

Subtopics

Database Field Setup

Edit Object Data

Adding values for drawing entities and objects

Creating Reports

Database Field Setup

The first step in using TurboCAD's database is to define fields. Fields are the basic building blocks of any data base. Consider the most common database in existence, the telephone book. The fields are Name, Address, Phone Number. The telephone company builds a database for each exchange by filling in these three fields for each customer.

In TurboCAD you define the fields for each item you wish to include in the database. There is one database for the drawing; all fields will belong to this master database field list. Enter the field names using the Define Fields dialog (Tools|Database|Define Fields).

Subtopics

Automatic Field Names

Field Properties

Automatic Field Names

When you first define fields, TurboCAD displays six field names. These *automatic fields* match the basic characteristics of the items in the drawing, and cannot be edited within the database.

Object ID. Every object created in TurboCAD is assigned a unique Object ID number by the program. This is similar to how a database management program will assign a unique sequence number to every record created.

Layer. This field will display the layer name of each item in the database.

Type. This field will display the entity or object type, such as line, multiline, arc, etc.

Attribute. Any information which appears in an object's Attribute property sheet will appear here.

Pen Color. The name of an object's selected pen color will appear here.

Brush Color. The name of an object's selected brush color will appear here.

Field Properties

The six automatic fields, as well as any new field you create, have the following properties:

Field Name. For the automatic fields, these names are assigned. For new fields, you may assign any descriptive name you choose. TurboCAD requires unique field names, and will not let you use a field name twice in any one drawing's database.

Data Type. TurboCAD will place the name of the object or entity type (line, arc, group, etc.) in this field.

Visible. If a field is checked as visible, it will appear as an item in other Database dialogs, and can be sorted or added to a report. If Visible is not checked for any item, the field will not appear in other Database dialogs.

Tip: If you know you will not be using the values in any automatic fields when you prepare reports using the database, deselect Visible for those fields. The fields will not appear when you edit or prepare reports.

Editable. If this box is checked, then data can be entered or edited at any time in the Edit Object Data dialog. If this box is not checked, then any data in the field cannot be changed.

Note: The six automatic fields are not editable; they represent data elements that must be changed as properties. Object ID cannot be changed.

Description. This field is for comments or other brief description. Generally you use this field to expand the description in Field Name. It can also be used for other comments about items added to the database.

Edit Object Data

Once field names and data types have been established, the next step is to add information about each item into the database. Select one or more objects in the drawing, then select Menu: Tools|Database|Edit Object Data.

The various pieces of information you enter in the database (for example, a part number) are called *Values.* The Edit Object Data dialog is used to enter and edit values.

When first opened, the Edit Object Data dialog shows a summary view of field names and values for all objects and entities selected in the drawing. If more than one object or entity is selected, the summary view will show all the field names, but will display "Not Equal" under values unless all the selected items have the exact same value for a particular field. If you selected four matching office tables, for example, the summary view would show values for Part Number, Price, and Item. The Show All and Collapse buttons switch the dialog between the summary and detail views.

From either view, it is possible to enter new values or edit existing values. If a value of "Not Equal" is edited in the summary view, the value for all selected objects will be changed. Values marked not editable in the Define Fields dialog will be gray in the Edit Object Data dialogs.

Field Info. Click on a field name or a value, and Field Info will appear, listing Field Type, Editable, and Description for the selected field.

Adding values for drawing entities and objects

To add new information to the database about entities and objects, select the object then open the Edit Object Data dialog (Menu: Tools|Database|Edit Object Tools). If only one object is selected, it does not matter whether you use the summary view or the show all view. Click on a field, then type the required information. You may use <Tab> to move from field to field. Click OK when finished.

Creating Reports

The purpose of a drawing database is to generate reports based on available information. The Report Create dialog is used to generate reports. A report may be as simple as a request for all values on all objects, or as detailed as a list of items with a cost of between $0.49 and $2.97 drawn with Red Pen Color on Layer 13.

The various Report dialogs are used to define a report, view the selected data, and to export the data to the printer or another program. Two report styles are available, Summary and Detail. The differences are subtle:

Summary Report. All values in a Summary Report may be edited before printing, even if they come from a non-editable field.

Detail Report. Only values from editable fields may be modified in a Detail Report. Field Info is shown for any selected value.

The data in either style of report can be sorted in Ascending (A-Z or lower to higher) or Descending (Z-A or higher to lower) order, using any field as the sorting criteria. A SUM function is available, to calculate a total or an item count in any column.

A completed report can be saved to disk, printed, or copied to the Windows Clipboard (to be pasted into another program). Reports saved to disk are stored in a spreadsheet file, in the TCW40\Reports folder.

Subtopics

Advanced Report Options

Advanced Report Options

A second dialog, Advanced Report, is available from the main Reports dialog. The dialog varies, depending on whether a Detail Report or a Summary Report is being created.

Advanced Detail Report. Field Names (Report Header Name) may be edited.

Advanced Summary Report. Report Header Name may be edited, and one of five math functions may be added to a field:
- SUM, which calculates the total of all values in the field;
- AVG, which calculates an average for all values in the field;
- MIN, which displays the smallest amount among all values in the field;
- MAX, which displays the largest amount among all values in the field;
- COUNT, which totals the number of values in the field.

Tools Menu Commands

The following table summarizes the commands accessible from the Tools menu. Only the major items are shown.

Command	Function
Customize	Modify or create new toolbars, menus and keyboard shortcuts.
Database	Assign data to drawing objects and create reports from the data.
Run Scripts	Run a TurboCAD script.
Measurement	Measure distance, angle, area and location of coordinate points.

Customize tools

Menu: Tools|Customize

The Customize dialog contains four sections:
- Toolbars (TurboCAD v4 Professional Only)
- Menus (TurboCAD v4 Professional Only)
- Keyboard (TurboCAD v4 Professional Only)
- Setup

The first three sections allow you to modify the various aspects of the user interface. The fourth section, Setup, is for managing the configuration files you create when you customize the program.

Subtopics

Customize Toolbars

Customize Menus

Customize Keyboard

Customize Setup

Customize Toolbars

The Toolbars dialog contains the following controls:

Categories. This list box displays an expandable chart of all TurboCAD commands and tools, listed by menu location. As you build a new toolbar, you will select a category of tools, and the available tool buttons will appear to the right. At the bottom of the list is "All Commands." Select this item and every TurboCAD command will appear as text entries instead of as tool icons. This feature is useful if you need to create a toolbar from commands that don't have tool icons, such as the Database tools or the text format tools.

Buttons. Click on any button displayed in this box; its description will appear in the Description box. To add a button to a toolbar, drag the button onto the toolbar.

Style. New toolbars may be in one of three styles:
- ▪   Standard: Buttons are arranged in a straight line.
- ▪   Tabbed: Two or more sets of buttons share a single toolbar; each set is identified by a unique tab.
- ▪   Fly-out: Related buttons are hidden behind the first button in the set; the entire group appears when you click and hold on the first button. Fly-out toolbars have a yellow triangle in the lower right corner of the first button.

Toolbar. Drag the New button onto the desktop to start a new toolbar. To delete or rename an existing toolbar, select the toolbar in the gray area, then click the Delete or Rename button.

Load From. Retrieve a previously saved configuration file containing one or more customizations. Click the list arrow to see the names of previously saved configuration files.

Save In. Save the current customization as a unique configuration. Click the list arrow to see the names of previously saved configuration files. Click in the text box to type a name for a new configuration.

The *Load From* and *Save In* options make it possible to have several desktop configurations. You may want to create separate configurations for different users of the same program or for different tasks.

Description. Explains the purpose of a selected button.

Close. Click this button to finish.

Tip: Make all your customization changes (Toolbars, Menus, Keyboard), then use the Save In command to name them. This will store all the changes in one configuration file. This tip assumes that Save All Sections boxes are checked in the Customize|Setup dialog.

Subtopics

Creating a Standard toolbar
Creating a Tabbed toolbar
Creating a Fly-out toolbar
Creating a mixed toolbar

Creating a Standard toolbar

A standard toolbar is one that does not use tabbed or fly-out tools. Follow these steps to create a standard toolbar.

1. From the menu, select Tools|Customize. Make sure Toolbar is the active tab.
2. Click the Standard radio button.
3. Drag the New button out of the Customize dialog and onto the desktop.
4. The New Toolbar dialog will appear. Type a descriptive name for the new toolbar and click OK. The new toolbar will appear, floating on the desktop.
5. In the Categories list, find the first tool you wish to add to the toolbar. Expand or contract each portion of the list as necessary to find the tool you need. To identify any tool, click on it in the Buttons section of the dialog. To add the tool to the new toolbar, drag the tool from the Buttons area to the new toolbar.
6. Repeat Step 5 for each additional tool.
7. Click the Close button when finished.

Creating a Tabbed toolbar

A tabbed toolbar is a group of tools that share a common tab. The use of tabs makes it possible to have a larger number of tools in a smaller toolbar.

Follow these steps to create a toolbar with two tabs.

1. From the menu, select Tools|Customize. Make sure Toolbar is the active tab.
2. Click the Tabbed Radio button.
3. Drag the New button out of the Customize dialog and onto the desktop.
4. The New Toolbar dialog will appear. Type a descriptive name for the first tab of your new tabbed toolbar, and click OK. A new toolbar will appear, floating on the desktop.
5. In the Categories list, find the first tool you wish to add to the toolbar. Expand or contract each portion of the list as necessary to find the tool you need. To identify any tool, click on it in the Buttons section of the dialog. To add the tool to the new toolbar, drag the tool from the Buttons area to the new toolbar.
6. Repeat Step 5 for each additional tool.
7. To start a second tab group, drag the New button out of the Customize dialog and onto the desktop.
8. The New Toolbar dialog will appear. Type a descriptive name for the second tab of your new tabbed toolbar, and click OK. A new toolbar will appear, floating on the desktop.
9. Repeat Steps 5 for each additional tool desired for the second tab.
10. When the second tab contains all the desired tools, drag the second tab onto the first one. The second tab will merge into the first toolbar, and the toolbar frame for the second tab will disappear.
11. Click the Close button when finished.

Creating a Fly-out toolbar

A fly-out toolbar is a group of tools that share the same space on a toolbar; the entire group of tools "fly out" of the toolbar when you press and hold on the first tool. The use of fly-out tools makes it possible to place a large number of tools in a small toolbar, and to combine related tools in a small space. A small yellow triangle in the lower right corner identifies the first tool in a fly-out toolbar.

Fly-out toolbars must be added to newly created or existing standard or tabbed toolbars; it is not possible to drag the New button onto the desktop when the Fly-out radio button is checked.

Any tool in an existing toolbar can become the first tool in a fly-out tool group, or a group of fly-out tools may be added to an existing toolbar without changing any existing tools in the toolbar.

Follow these steps to add a group of fly-out tools to an existing toolbar.

1. Make sure the toolbar you will expand is floating on the desktop.
2. From the menu, select Tools|Customize. Make sure Toolbar is the active tab.
3. Drag the dialog as necessary so you can see both the dialog and the toolbar.
4. Click the Fly-out Radio button.
5. In the Categories list, find the tool you wish to become the first tool of the fly-out group. Expand or contract each portion of the list as necessary to find the tool you need. To identify any tool, click on it in the Buttons section of the dialog. To select the tool, drag the tool from the Buttons area to the floating toolbar; place it on the toolbar but not on top of an existing tool.
6. Drag the second tool for the fly-out group to the new toolbar. Place it directly on top of the first tool.
7. The New Toolbar dialog will appear. Type a descriptive name for the fly-out tool group, and click OK. The two tools will now appear as the start of a fly-out tool group. A small yellow triangle identifies the first tool in the group.
8. Drag additional tools onto the fly-out group as desired.
9. Click the Close button when finished.

To create a fly-out tool group using a tool in an existing toolbar as the first tool in the fly-out group, follow the preceding steps, changing Step 5 by dragging the first tool on top of an existing tool. The existing tool will become the first tool in the fly-out group; the first new tool you add will become the second tool in the fly-out group.

Creating a mixed toolbar

It is possible to create toolbars that mix Standard, Tabbed and Fly-out display styles. Start with either a Standard or Tabbed toolbar, then click the radio button for Tabbed or Fly-out. The new tools you drag onto the toolbar will display in the selected style.

Customize Menus

The Menus dialog contains the following controls:

Categories: This list box displays an expandable chart of all TurboCAD menus. When you select a menu, its contents appear to the right in the Commands list.

Commands: As a menu is selected in the Categories list, the individual commands are listed here.

Change What Menu: Select a menu to modify, or select [Menu Bar] to create a new menu.

Position On Menu: When you have selected a menu to modify, and have selected a command to add to it, click on the list box arrow and select a position within the menu for the new item. The choices are:

- Auto: TurboCAD will insert the items in the order selected.
- At top: The new menu item will be the first item in the menu.
- At bottom: The new menu item will be placed at the bottom of the menu.
- Separator: Select this option to separate menu items into groups.

Name On Menu. You may use this command to rename any new menu item. Click on the text box and type the new name. To select a letter in the menu name as the accelerator key (also known as a keyboard shortcut), place the & sign directly in front of the character. Finish by clicking the *Add* button.

Description. Explains the purpose of a selected menu item.

Rename. This button appears when you have selected a menu item. The location of this button is shared by the *Add* button. Click *Rename* when you have typed a new name for a menu item in the *Name On Menu* field.

Add. Click this button to add a menu item to a selected menu. Be sure the correct *Position On Menu* choice is showing.

Remove. Use this button to remove an item from a menu (new or existing).

Reset All. Cancel any changes, revert to original menu status.

Add Submenu. When an existing menu is selected in *Change What Menu:*, a new submenu will be added. When [Menu Bar] is selected in *Change What Menu:*, a new primary menu will be added to the menu bar.

Load From. Retrieve a previously saved configuration file containing one or more customizations. Click the list arrow to see the names of previously saved configuration files. Click the *Load From* button to accept the selected configuration file.

Save In. Save the current customization as a unique configuration. Click the list arrow to see the names of previously saved configuration files. Click in the text box to type a name for a new configuration. Click the *Save In* button to save the new configuration.

The *Load From* and *Save In* options make it possible to have several desktop configurations. You may want to create separate configurations for different users of the same program or for different tasks.

Close. Click this button to finish.

Subtopics

Adding Accelerator Keys to Menu Items

Adding Accelerator Keys to Menu Items

Place the ampersand (&) in front of the letter you wish to designate as the accelerator key (also known as a keyboard shortcut) for any new menu item. The letter will be underlined in the menu. To access the shortcut, press ALT plus the letter that opens the menu, then press the accelerator key for the item.

Customize Keyboard

The Keyboard dialog contains the following controls:

Categories. This list box displays an expandable chart of all TurboCAD menus. When you select a menu, its contents appear to the right in the Commands list.

Commands. As a menu is selected in the Categories list, the individual commands are listed here.

Press New Shortcut Key. When you have identified a command, click in this text box and type the keystroke or key combination you wish.

Current Keys. If the selected command already has a keyboard shortcut, it will appear in this text box.

Description. Explains the purpose of a selected command.

Assign. After typing the new shortcut in the *Press New Shortcut Key* box, click this button to link the shortcut to the command.

Remove. Click this button to remove a shortcut from the selected command.

Reset All. Cancel any changes, revert to original status.

Load From. Retrieve a previously saved configuration file containing one or more customizations. Click the list arrow to see the names of previously saved configuration files.

Save In. Save the current customization as a unique configuration. Click the list arrow to see the names of previously saved configuration files. Click in the text box to type a name for a new configuration.

The *Load From* and *Save In* options make it possible to have several desktop configurations. You may want to create separate configurations for different users of the same program or for different tasks.

Close. Click this button to finish.

Customize Setup

All three customization options may share a common set of configuration files. This dialog controls whether or not the file names are common.

The key elements are the *Load All Sections* and *Save All Sections* check boxes. If these boxes are checked, then all three customization options will use the same configuration files. If one of these boxes is not checked, then you will need to individually select which customization options are active when configuration files are loaded or saved.

Load Configuration File Section. This list box displays the current types of customization configuration that can be loaded and modified. If Load All Sections is not checked, you must individually select customization options from this list.

Save Section in the Configuration File. This list box displays the current types of customization configuration that can be saved when modified. If *Save All Sections* is not checked, you must individually select customization options from this list.

Load From. Retrieve a previously saved configuration file containing one or more customizations. Click the list arrow to see the names of previously saved configuration files.

Save In. Save the current customization as a unique configuration. Click the list arrow to see the names of previously saved configuration files. Click in the text box to type a name for a new configuration.

The *Load From* and *Save In* options make it possible to have several desktop configurations. You may want to create separate configurations for different users of the same program or for different tasks.

Reset All. Cancel any changes, revert to original status.

Close. Click this button to finish.

Database tools

Menu: Tools|Database

The Database tools, in TurboCAD v4 Professional, are used to add information to objects and entities, and to create database reports from that information. Four command subjects are listed beneath Database in the Tools menu: Define Fields, Edit Object Data, Report, and Regenerate. One database can be created for each drawing. As many custom reports as necessary can then be generated from the database.

Subtopics

[Define Fields](#)

[Automatic Fields](#)

[Edit Object Data Command](#)

[Report](#)

[Regenerate](#)

Define Fields

The first step in creating a database is to define the types of information to be recorded. *Field Name* describes the type of information, *Value* is the specific piece of information recorded under a field name. Other field types are recorded for each piece of information. A related set of values entered into the data base for the set of fields is known as a *Record.*

Six *Automatic Fields* are predefined for each drawing's database. These six fields contain object and entity information that cannot be edited or changed from within the database; these are properties that must be changed in a property sheet.

To start a new database, select Define Fields from the menu. Click on the first blank under Field Name, and type the new field name. Then continue to fill in the blanks for the other fields.

Field Name. This is the name of the piece of information to be recorded. To add a field name to the database, click on the first blank item in the Field Name column, and type the new name.

Data Type. Click on the blank under Data Type. A list box arrow will appear. One of four data types may be selected:
- Text, for alphanumeric information which does not have to be calculated.
- Integer, for whole numbers.
- Double, for decimal numbers.
- Currency, for dollar amounts.

Visible. An on-off option. If checked, the field will be visible and available for use in the Edit Object Data and Report dialogs. If not checked, the field will not be visible in the other dialogs and will not appear in reports.

Editable. An on-off option. If checked, the field can be edited in the other database dialogs. If not checked, the field is locked and cannot be edited in the other dialogs. Auto fields are always non-editable.

Description. An optional field, for notes or other comments about the field.

To delete a field name or other information, click on the text box, then click the Delete button.

Automatic Fields

Six Automatic Fields (called Auto Field in the dialogs) are automatically generated for each drawing's database. These automatic fields represent object or entity information, and cannot be edited from within the database; it must be changed in a property sheet. The six fields are:

Object ID. Every object created in TurboCAD is assigned a unique Object ID number. It cannot be changed.

Layer. This field will display the layer name of each item in the database.

Type. This field will display the entity or object type, such as line, multiline, arc, etc.

Attribute. Any information which appears in an object's Attribute property sheet will appear here.

Pen Color. The name of an object's selected pen color will appear here.

Brush Color. The name of an object's selected brush color will appear here.

Edit Object Data Command

Once fields have been defined, the next step is to enter data. If you draw a wing nut, for example, and want to record data about it in the database, you would select the wing nut (most likely existing as a group, symbol or block in the drawing), then access the Edit Object Data dialog to enter information about the item.

There are two versions of the Edit Object Data dialog:

Summary View. This dialog shows the fields and values for one selected record. This is the default dialog if only one item is selected in the drawing. Click any white text box to enter information. If more than one item is selected in the drawing, editing any field will set the values for all selected items.

Show All View. This dialog displays all fields for all items in the drawing. Values may be entered for any editable field.

Click the Show All/Collapse button to switch between the Summary and Show All views.

The *Field Info* dialog lists Field Type, Editable status, and the Description of the selected value.

Report

Menu: Tools|Database|Report

After fields have been created, and data entered, the database can be used to generate reports. Four dialogs control the various phases of the report process.

Subtopics

[Report dialog](#)

[Report Edit dialogs](#)

[Summary Report / Detail Report dialog](#)

[Summary and Detail Report tools](#)

Report dialog

The Report dialog appears when the Report option is selected from the menu. From this dialog, you can start the process of creating a new report, edit an existing report, or generate a report. The buttons that select these options are:

Report. Click this button to generate a previously designed report. A report name must be selected from the list of Available Reports for this button to be available.

Edit. Click this button to modify an existing report.

Create. Click this button to create a new report. A "Create New Report" dialog will appear; type in a name for the report. The Report Edit dialog will then appear.

Delete. Click this button to delete a report selected in the Available Reports list.

Report Edit dialogs

The Report Edit dialog has two tabs, Report and Advanced Report. The following categories are found in the Report tab:

Report Title. The report name assigned by the Create Report dialog appears here, and may be edited if desired.

Report Type. There are two options, Detail and Summary:
- **Detail.** Only values from editable fields may be modified in a Detail Report. Field Info is shown for any selected value.
- **Summary.** All values in a Summary Report may be edited before printing, even if they come from a non-editable field.

Comments. A line describing the purpose of the report may be entered in this text box.

Available Fields. All fields previously marked as visible will appear in this list. Select a field name, then use one of the buttons to the right of the list to add a field to the Report Fields list.

Report Fields. Any fields selected using the Add or (Add) All buttons will appear in this list. To remove an item from this list, use the Remove or (Remove) All buttons.

Query Criteria. This field shows any query created to sort data. To create a query, click the Edit button below the Query Criteria field to activate the Create Query dialog.

Note: The Create Query dialog is the exact same dialog used by the Select By Query tool. For a complete description of this tool, refer to **Select By Query**.

Apply. Click this button to accept the settings in the dialog.

Two possible Advanced Report dialogs are available, depending on the status of the Report Type in the Report field. If Detail is checked, the Advanced Report allows Field Names (called Report Header Name) to be edited. If Summary is checked, Report Header Names may be edited, and one of five math functions may be added to a field:
- SUM, which calculates the total of all values in the field.
- AVG, which calculates an average for all values in the field.
- MIN, which displays the smallest amount among all values in the field.
- MAX, which displays the largest amount among all values in the field.
- COUNT, which totals the number of values in the field.

Summary Report / Detail Report dialog

When the Report button is clicked in the opening Report dialog, one of two reports will be generated, Summary or Detail. (Which report is run depends on the Report Type setting in the Report Edit field.) All values in a Summary Report may be edited before printing, even if they come from a non-editable field. Only values from editable fields may be modified in a Detail Report. Field Info is shown for any selected value.

Summary and Detail Report tools

Seven tool icons appear in both the Summary Report and Detail Report dialogs. From left to right, the seven tools are:

Export. Click this button to save a report to disk. The file may be saved as a Microsoft Excel Spreadsheet (XLS), Microsoft Access Database (MDB), a Comma Separated Value (CSV), or a Tabbed Text File (TXT). The default folder is to the \TCW40\Reports. The file is a text file readable by other database programs, word processors, etc.

Page Setup. Click this button to set the format for a printed version of the report, using a standard Windows Print Setup dialog.

Print. Click this button to print the report using current Print Setup options.

Copy. Places all report data on the Windows Clipboard, so it can be pasted into another Windows application.

Sort Ascending. Click a Field Name, then click this button to sort from A to Z or from smallest to largest, using the selected field as the sort criteria.

Sort Descending. Click a Field Name, then click this button to sort from Z to A or from largest to smallest, using the selected field as the sort criteria.

Grand Total. Click a blank field at the bottom of any column, then click this button. If the field contains Currency, Integer or Double as the data type, the Grand Total button will calculate a total for the column. If the field contains Text information, the Grand Total button will display the total number of items in the column.

If new data has been added to the database since the last time the report was used, click the *Update* button to regenerate the report, adding the new data. This is the same as using the Regenerate tool from the menu (Tools|Database|Regenerate).

Regenerate

If new drawing elements have been created, and new data entered into the database, select the Regenerate command from the menu. This will force an update of the database to account for all new entities, objects, and data elements.

Run Scripts

Menu: Tools|Run Scripts   **Toolbar:** Script

The Run Scripts tool lets you run scripts placed in a drawing, or browse for scripts located anywhere on your computer (using Options|Script). Scripts are created in TurboCAD's built-in BASIC language using Notepad or any text editor or word processor. You can place scripts in your drawing using the Scripts property sheet (Options|Script). You can also debug your script using the new Basic debugger.

Scripts you have placed in your drawing will appear in a submenu off the Run Script menu, followed by a More Scripts menu item. The scripts will also appear as buttons on the Script toolbar. You can run one of these scripts by choosing it from the submenu or clicking its tool button.

If you want to run a script that is not placed in your drawing, just choose Run Script (if no scripts are placed in your drawing), or More Scripts from the submenu (if there are scripts placed in your drawing). You can also click the Run Scripts tool shown in the heading of this topic. The Run Script dialog will appear.

Use this standard Windows dialog to browse to the script that you want to run, then press <Enter> to run the script. For additional information on creating scripts, see the "Programming TurboCAD with Enable BASIC" manual.

Note: TurboCAD's Scripts folder contains some sample scripts that you can run and explore.

See Also: Scripts Property Sheet

Measurement

The measurement tools are used to find the area, distance, angle, or location of objects in the drawing. The results will appear in the Measurement palette, which automatically appears when any Measurement tool is activated.

Subtopics

[Measurement Tools and the Local Menu](#)
[Measure Coordinates of the Point](#)
[Measure Distance](#)
[Measure Angle](#)
[Measure Area](#)

Measurement Tools and the Local Menu

When using measurement tools, the local menu contains available units of measurement, as well as selection aids that vary with the measurement tool. Right-click to summon the local menu, and click on a unit of measurement or a selection aid.

Measure Coordinates of the Point

Menu: Tools|Measurement|Coordinates of the Point   **Toolbar:**
Measurement

Click on any location in the drawing to see the coordinate locations. Snap commands may be used.

Local menu. All available units of measurement are listed, as well as local snaps. No other specific aids are available.

Measure Distance

Use this command to measure both the distance from one point to another, and to measure a perimeter. As you identify locations on-screen, a purple tracking line will show the measurement path.

Local menu. All available units of measurement are listed, as well as local snaps. Selection aids for Measure Distance are Finish, Close, One Step Back, By Entity, and Single Entity.

Measure Angle

Use this command to measure the angle between two lines. As you identify locations on-screen to identify the angle, a purple tracking line will show the measurement path. Identifying two locations or entities will generate an angle measurement; if you select a third location, you start a new measurement command.

Local menu. Selection aids for Measure Angle are Finish, One Step back, By Entity and all local menu snaps.

Measure Area

Use this command to measure the area in the drawing. The area does not have to be currently bounded by lines or other objects. As you identify locations on-screen to select the area, a purple tracking line will show the measurement path. Click on Close in the local menu to finish selecting and calculate the area.

Local menu. All available units of measurement are listed, as well as all local snaps. Selection aids for Measure Area are Close, One Step Back, and By Entity.

Geometric Editing

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. The following table summarizes the modify menu and its functions.

Command	Function
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 a 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, and 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

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. TurboCAD will not allow you to remove the last remaining part of the drawing touching the cutting edge.

Note: In previous versions of TurboCAD, a curve could not be a cutting edge for Object Trim. Curves are now acceptable as a cutting edge.

Warning: Object trim will not work with certain complex entities, specifically dimensions and text. To use the tool on these types of objects (as well as on groups and block inserts), you must first select the object that you want to trim, then use the 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

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), with a SEKE (to access a snap) 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: 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.

See Also: Node Edit

Split

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, arcs and curves.

Subtopics

Splitting Line and Double Line Entities

Splitting Arcs and Circles

Splitting Curves

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 temporary 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.

Notice that TurboCAD leaves a temporary red cross to indicate where the split occurred on the object.

Splitting Arcs and Circles

To divide an arc into two arc segments:

1. Choose the Split tool.
2. Click on the arc you want to split. A dotted line appears, connecting the cursor to the center point 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).

Splitting Curves

It is helpful to display the a curve's frames before splitting it. This is because the split actually takes place on the frame. To display a frame, double-click on the curve to summon the Property dialog, select the Curve property sheet, and click to activate Show Frame.

To split a curve:

1. Choose the Split tool.
2. Click on the curve you want to split. The frame for that portion of the curve will be highlighted.
3. Move the cursor to the exact location you desire, and click to split the curve.

Note: To split a closed curve, you will need to split in two locations, defining each end of the curve section to be split away from the rest of the curve.

Meet 2 Lines

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 Tools

Menu: Modify|Chamfer   **Toolbar:** Modify, Standard

Use the Chamfer tools to join two line segments or double line segments with a flattened corner.

Subtopics

[Values for Calculating Chamfers](#)

[Distance and Distance Chamfer](#)

[Distance and Angle Chamfer](#)

[Length and Angle Chamfer](#)

Values for Calculating Chamfers

When you activate a chamfer tool, the values you can use to calculate the chamfer appear in the edit bar.

The following table describes these values and their meaning. Note that "point of intersection" means the point where the two lines intersect, or the point where they would intersect if they were extended.

Value	Meaning
Distance A	The distance from the point of intersection to the chamfer on the first line clicked.
Distance B	The distance from the point of intersection to the chamfer on the second line clicked.
Length	The length of the chamfer line.
Angle	The angle of the chamfer line from the first line clicked.

The TurboCAD chamfer tools use three combinations of these values:
- Distance and Distance
- Distance and Angle
- Length and Angle

Note: You can use the Multiline option to chamfer all the corners of a multiline or polygon simultaneously.

Note: If the values for length are too large in relation to the existing lines, the chamfer will not be possible, and TurboCAD will issue the warning "Chamfer is not possible in this case."

Distance and Distance Chamfer

Menu: Modify|Chamfer|Distance/Distance   **Toolbars:** Modify, Standard

The Distance and Distance 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 Distance and Distance 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 on the first, then the second line.
- Click on a multiline or polygon, right-click to open the local menu, then choose Multiline. This will chamfer all the corners of the multiline or polygon that you selected.

Distance and Angle Chamfer

Menu: Modify|Chamfer|Distance/Angle   **Toolbars:** Modify, Standard

The Distance and Angle Chamfer tool draws a chamfer from a point on the first line clicked calculated by its distance from the point of intersection, at a specific angle from the first line clicked:

1. Activate the Distance and Angle Chamfer tool.
2. Type values in the Distance and Angle fields on the Edit Bar. Return to the drawing area by clicking there or pressing <Ctrl>+<D>.
3. Do one of the following:

- Click on the first, then the second line.
- Click on a multiline or polygon, right-click to open the local menu, then choose Multiline. This will chamfer all the corners of the multiline or polygon that you selected.

Length and Angle Chamfer

This method joins two lines with a chamfer of the specified length, at a specified angle from the first line clicked:

1. Activate the Length and Angle Chamfer tool.
2. Type values into the Angle and Length fields of the Edit Bar. Return to the drawing area by clicking there or pressing <Ctrl>+<D>.
3. Do one of the following:
- Click on the first, then the second line.
- Click on a multiline or polygon, right-click to open the local menu, then choose Multiline. This will chamfer all the corners of the multiline or polygon that you selected.

Line Length

Menu: Modify|Line Length   **Toolbars:** Modify, Standard

Use the Line Length tool 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), and then press <Enter>.
▪   Move the mouse to lengthen or shorten the line, then click when it is the length you want.

Shrink/Extend Line

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. TurboCAD will adjust the length of the line or double line until it precisely meets the second entity.

Arc Complement

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

Use the Fillet tool to connect two lines, either single or double lines, with a smoothly fitted arc. The result is a rounded corner.

TurboCAD 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. TurboCAD then trims or extends the lines to form the smooth corner.

Note: You can use the Multiline option to fillet all the corners of a multiline or polygon simultaneously.

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 on the first, then the second line.
- Click on a multiline or polygon, right-click to open the local menu, then choose Multiline. This will fillet all the corners of the multiline or polygon that you selected.

Note: If the radius specified is too large for the arc to fit between the two lines, TurboCAD will be unable to perform the fillet and will display an error message.

T-Meet 2 Double Lines

Use the T-Meet 2 Double Lines tool to form a T intersection of two double lines. The first line clicked will extend or shrink to form the stem of the T.

1. Activate the T-Meet 2 Double Lines tool.
2. Click on the line that will form the stem of the T.
3. Click on the line that will form the top of the T.

Note: You can choose whether or not to make a clean intersection of the two double lines. To do this, activate the T-Meet 2 Double Lines tool, right-click in the drawing area, then select or deselect the Cleanup option in the local menu.

Tip: Use Node Edit to close the ends of double lines. Select the Node Edit tool and click on the double line to close. Move the cursor over the node in the gap, right-click to summon the local menu, and select Close Double End.

Intersect 2 Double Lines

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.

 Note: You can choose whether or not to make a clean intersection of both double lines. To do this, activate the Intersect 2 Double Lines tool, right-click in the drawing area, then select or deselect the Cleanup option in the local menu. If you choose not to make a clean intersection, there will be a gap in the first line clicked, but not in the second line clicked.

Snaps and Coordinates

Modes enable you to control how TurboCAD 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 TurboCAD 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.

Subtopics

Snap Modes

Modes Menu Commands

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 entities, 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, TurboCAD will define the point precisely on the object. The point that TurboCAD 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.

Subtopics

Snap Aperture

Local Snaps and SEKEs

Coordinate Systems

Cartesian Coordinates

Polar Coordinates

Drawing with Coordinate Systems

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.

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

To get additional visual feedback when using snap modes, turn on Magnetic Point mode. 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.

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

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

See Also: Program Setup Dialog

Local Snaps and SEKEs

You can change snap modes in four ways. You can use the menu, the Snap toolbar, the local menu, or a Single Entry Keyboard Equivalent (SEKE). 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 and SEKEs are different. Use them for "one shot" mode changes. This means that when you change snap modes in the local menu or by pressing a SEKE, the mode remains in effect until you define the next point. TurboCAD then switches back to the last mode you set in the main menu or toolbar, or defaults to No Snap mode.

One-shot snaps are useful because they let you set the snap mode you use most frequently in the main menu or toolbar, then switch modes "on the fly" 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 can press V (for Vertex) or change the snap mode temporarily to Vertex in the local menu and define a point on the polygon corner. TurboCAD then automatically switches back to Snap to Grid, the mode they most commonly use.

Most SEKEs are snap commands. But a few perform other functions. Here is a list of SEKEs, in alphabetical order. For a complete list see Help|Keyboard.

- **A** Set Absolute Coordinates. All coordinate references will be in Absolute Distances.
- **C** Snap Arc Center. Connect to the center of an arc or circle.
- **D** Relocate reference point of a selected object.
- **G** Snap Grid. Connect to the nearest grid point or intersection.
- **H** Snap Opposite. Continue line same distance from object as from first point to object.
- **I** Snap Intersection. Connect to the intersection of two entities.
- **J** Snap Single Projection. Connect to an entity and project a perpendicular line.
- **M** Snap Middle Point. Connect to the midpoint of an entity.
- **N** Snap Nearest on Graphic. Connect to the location on a object the cursor touches.
- **P** Set Polar Coordinates. All coordinate references will be Polar (distance and angle).
- **Q** Snap Quadrant Point. Connect to the nearest quadrant point on a circle or arc (0, 90, 180, 270).
- **R** Set Relative Coordinates. All coordinate references will be in Relative Distances.
- **S** No Snap. Enter the location of the point.
- **V** Snap Vertex. Connect to the end of the line or other object.
- **X** Lock X. Lock the current X position as displayed in the Edit Bar.
- **Y** Lock Y. Lock the current Y position as displayed in the Edit Bar.
- **<F7>** Select the last object drawn (no other objects must be selected).
- **<F6>** Select the first object drawn (no other objects must be selected).

Coordinate Systems

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

The current coordinates of the cursor are displayed in the Coordinate Display, on the same line as the Status Bar at the bottom of the desktop. You can access these fields by clicking in them, or by pressing <Shift>+<Tab>.

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 the 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).Take a moment now to move the cursor around the screen, and notice how the Coordinate Display changes.

Subtopics

Absolute Coordinates
Relative Coordinates

Absolute Coordinates

When using absolute coordinates in TurboCAD, 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 the Page Setup dialog).

See Also: Page Setup Dialog

The menu command Modes|Coord System|Relocate Origin allows you to reset the origin point used by the Relative and Polar coordinate reference systems, not the Absolute reference system.

Relative Coordinates

When you are working in relative coordinates in TurboCAD, 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.

TurboCAD also has a feature that lets you fix the Relative Origin in one place, then move it manually. If you check the Fixed Relative Origin box in the Program Setup dialog, the Relative Origin will stay in one place until you explicitly move it, using the Relocate Origin command.

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.

Note: The Edit Bar uses polar coordinates to define the length and angle of lines.

See Also: Relative Coordinates, Relocate Origin

Drawing with Coordinate Systems

TurboCAD 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.

Tip: Advanced users can "force" TurboCAD 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.

Modes Menu Commands

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

Command	Function
Snaps	Set the current snap mode.
Coord	Set the current coordinate system.
Lock X Position	Locks the X (horizontal) value as you draw
Lock Y Position	Locks the Y (vertical) value as you draw

Snap Tools
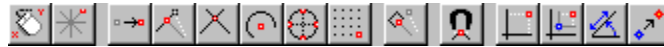
Menu: Modes|Snaps   **Toolbar:** Snap

The following table summarizes TurboCAD's snap modes:

Command	Function
No Snap	Turn off all snap modes; point is defined at the location of the mouse click.
Vertex	Snap to the nearest end of a line segment, endpoint of an arc, or corner of a polygon or multiline.
Nearest on Graphic	Snap to the nearest point on any entity.
Midpoint	Snap to the midpoint of a line segment.
Arc Center	Snap to the center of an arc or circle.
Quadrant Point	Snap to the nearest quadrant point (at 0, 90, 180, or 270 degrees) on the circumference of a circle or arc.
Intersection	Snap to the nearest intersection of two entities.
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.)
Perpendicular Snap	Use Perpendicular Snap Opposite to connect a line to an entity so that the line is perpendicular to the entity.
Show Magnetic Point	Increase visual feedback; the rubber band line you are currently drawing will gravitate toward the snap point.

Subtopics

Combining Snap Modes

Combining Snap Modes

Some snap modes can be combined, and some are mutually exclusive. The No Snap tool turns off all other snap modes. Nearest on Graphic 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 of the other 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.

Note: The snap modes that can be used at the same time are all in the same section of the Modes|Snaps submenu.

No 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.

No Snap is the least accurate of all snap modes. It relies on cursor position, which is dependent on screen resolution and the zoom factor.

The SEKE version of No Snap will not turn off other snap modes. You may find it useful to have all other snap modes on, and use No Snap as a SEKE when you need to place a point freehand.

Note: 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.

See Also: Ortho

Snap to Vertex

Menu: Modes|Snaps|Vertex   **SEKE:** V   **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 TurboCAD 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 or curve.

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, or the double line's reference point, depending on which point is closer to the cursor.

Snap to Nearest on Graphic

When Snap to Nearest on Graphic mode is active, a point will be defined at the nearest point to the cursor on an object within the snap aperture.

Nearest on Graphic is similar to No Snap in that it relies on cursor position, which is dependent on screen resolution and the zoom factor. The difference is that Nearest on Graphic will find the nearest vertex, center point, etc.

Snap to Midpoint

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

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 **SEKE:** Q   **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

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

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.

If Advanced Grid options frequency is set to a figure of more than one, invisible grid lines will also respond to this snap mode.

See Also: Grid Property Sheet

Perpendicular Snap

Use Perpendicular Snap Opposite to connect a line to an entity so that the line is perpendicular to the entity. Start by defining the first point of the line (using any snap as needed). Then move the cursor onto the target line, and press <J>. The line will snap to the entity so that the line is perpendicular.

Ortho

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

Note: Pressing the <Shift> key while using any drawing tool provides a temporary ortho mode.

Show Magnetic Point ![icon]

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 useful to have this mode on all the time.

See Also: Snap Aperture

Coordinate System

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. The following table summarizes the coordinate system commands:

Command	Function
Absolute	Change to absolute Cartesian coordinates, in which you define points by their distance from a fixed (absolute) point of origin.
Relative	Change to the relative coordinate system, in which you define points by their X and Y distance from the Relative Origin point.
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

Absolute Coordinates Command

Menu: Modes|Coord System|Absolute Coords   **SEKE:** A   **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, (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

Relative Coordinates Command

Menu: Modes|Coord System|Relative Coords   **SEKE:** R   **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: 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. The Relative origin is displayed as a small red square when Relative Coordinates is the active mode.

Tip: You can fix the Relative Origin so that it does not move by checking the Fixed Relative Origin box in the Program Setup dialog (General property sheet). You can then move the Relative Origin manually using the Relocate Origin command.

See Also: Coordinate Systems, Relocate Origin, Program Setup Dialog

Polar Coordinates Command

Menu: Modes|Coord System|Polar Coords   **SEKE:** P   **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. Since Polar Coordinate mode is a variation of Relative Coordinates, the origin point is displayed as a small red square when Polar Coordinates is the active mode.

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. Press <Shift>+<Tab> to access the Coordinate Display.
3. Type 3.5 into the length text box on the status bar.
4. Type 45 into the angle text box on the status bar, then press <Enter>.

See Also: Coordinate Systems

Relocate Origin

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 if Relative or Polar Coordinates is the active mode. 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

Lock X Position / Lock Y Position

Menu: Modes|Lock X Position, Modes|Lock Y Position   **SEKE:** X, **SEKE:** Y

These two commands are menu alternatives for clicking the lockbox corresponding to the X and Y coordinates in the Status Bar. They function as toggles: When one is selected from the menu, the command is active. To deselect, click the menu item again or click the appropriate lockbox in the Status Bar.

When either Lock X or Lock Y is selected, the values in the Status Bar for the selected field doesn'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.

See Also: Using the Edit Bar Lockdown Boxes

Options Menu and Dialogs

The Options menu gives you access to dialog boxes that control TurboCAD program and drawing settings. These features let you:
- Set program options that are saved with the application.
- Set drawing options that are saved with a drawing or a template.

Subtopics

[Program and Drawing Setup](#)

[Units and Scale](#)

[Setting Up the Grid](#)

[Setting Up Layers](#)

[Options Menu Commands](#)

Program and Drawing Setup

The Options menu contains two large sections, Program Setup and Drawing Setup. Changes that you make in the Program Setup are saved whenever you exit TurboCAD. The next time you start TurboCAD, 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 to create other drawings, you can save the Drawing Setup options in a template using the File|Save As dialog. You can make the template that you use most frequently the default template (the template that TurboCAD 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.

Tip: 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.

See Also: Drawing Setup Dialog, Program Setup Dialog, Save As Dialog

Units and Scale

TurboCAD has two distinct "spaces": world space and paper space. The units in world space apply to the real-world object that your drawing illustrates. 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 modeling 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.

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

The Units and Scale property sheet and the Advanced Units property sheet give 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 TurboCAD's standard settings, which correspond to settings common to specific CAD disciplines, or you can specify custom settings.

See Also: Advanced Units Property Sheet, 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, isometric or polar coordinate locations. You can make the grid visible or invisible using the Grid Display tool on the Grid toolbar, or the View|Grid menu command. You make the grid active or inactive by switching the Snap to Grid option on and off (using the Modes|Snaps|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 or not the grid is visible has nothing to do with whether it is active.

TurboCAD 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. If you want to customize the grid setup, you can specify how many minor grid points there are between major grid points; and you can specify how frequently the grid points are visible, so that only larger grid divisions are visible, but you can still snap to the finer divisions.

You can also rotate the grid at an angle, offset the grid so that it aligns with an object in the drawing, or change the magnification of the grid.

See Also: Advanced Grid Property Sheet, Grid Property Sheet, Snap to Grid, View Grid Options

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 objects. Layers are like acetate sheets used in traditional drafting.

Layers in TurboCAD 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 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

Options Menu Commands

The following table summarizes the commands accessible from the Options 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.

Command	Function
Program Setup	Display the Program Setup dialog, in which you can set options that are saved when you exit TurboCAD.
Drawing Setup	Display the Drawing Setup dialog, in which you can set options that are saved with your drawing.

Program Setup Dialog

Menu: Options|Program Setup   **Toolbars:** Setup, Standard

The Program Setup dialog contains property sheets for controlling TurboCAD program settings, summarized in the following table:

Property Sheet	Functions
General	Specify template, how to create a new drawing, read-only file access, user information, snap aperture, zoom factor, save, and advanced settings.
Desktop	Display and hide user-interface controls, control desktop colors and paper display.
Toolbars and Menu	Set options for use and appearance of toolbars and menus. Access to Customization options for toolbars, menus, and keyboard shortcuts (TurboCAD v4 Professional only).
Auto-Naming	Set options for naming groups, blocks and symbols.
File Locations	Specify directories for autosave, backup, template, symbol library, script, raster, and drawing files.
Symbol Libraries	Specify libraries to be available in the symbol library palette.
Color Palette	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 Options 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 TurboCAD, and will remain in effect the next time you start the program.

General Property Sheet of Program Setup Dialog

Menu: Options|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:

- Specify the default method of creating a new document.
- Set the default template used when you start a new drawing, and specify whether TurboCAD 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.
- Specify options for saving drawings, including backup copies.
- Choose Geometric Selection or Fixed Relative Origin.

The General property sheet contains the following controls:

New Document From. Select the preferred default behavior of the File|New menu option.

Default Template for New Files. If you do not specify a template when you start a new drawing, TurboCAD 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 TurboCAD 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.

See Also: Save As Dialog, Open, New

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 TurboCAD 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.

Zoom Factor. The zoom factor setting controls how much you zoom in or out of the drawing when you use the Zoom In (Numeric Keypad <+>) or Zoom Out (Numeric Keypad <->) 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.

Snap Aperture. The snap aperture is a circular area around the mouse cursor. TurboCAD 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.

Tip: You can view the Snap Aperture as you work by choosing View|Cursor|Aperture.

See Also: Snap Aperture, Snap Modes, Snap Tools, View Crosshairs, Zoom In, Zoom Out, Zooming

Subtopics

Save Options
Advanced Settings

Save Options

The General property sheet includes several ways to control how TurboCAD saves drawing files.

Prompt for summary information. If this box is checked, TurboCAD 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, TurboCAD 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 TurboCAD 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, TurboCAD 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. If there is a system crash while TurboCAD is in use, TurboCAD will open the autosave version of your drawing as soon as you launch TurboCAD. Autosave files are stored in the \AutoSave folder.

Save desktop on exit. If this box is checked, TurboCAD will save the desktop settings when you exit the program. This means that TurboCAD 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| Coord System|Relocate Origin command.

Geometric Select Edit. Check this box if you want to select objects by their geometric extents, rather than their cosmetic extents. This permits geometrically accurate scaling of objects like double lines and lines that have width, because the geometric and cosmetic extents differ.

See Also: Geometric and Cosmetic Select Modes, Advantages of Geometric Select Edit

Desktop Property Sheet

Menu: Options|Desktop   **Toolbars:** Setup, Standard

The Desktop property sheet contains controls that let you:
- Display and hide docking palettes and tabbed toolbars.
- Display and hide the paper margins and background.
- Set the color of the paper, background, and floating palettes.
- Set the status of Workspace controls.

See Also: View Toolbars, View Rulers, View Edit Bar, View Symbol Library, View Measurement Info

Subtopics

Palettes

Workspace Controls

Desktop Colors and Paper Display

Palettes

Palettes. This expandable list displays the names of dockable palettes and tabbed toolbars. Click the plus symbol before each name to expand the list; click the minus symbol to contract the list again. When the list is expanded, you can click the selection box to choose which palette will be active. You may see one or more docking palettes depending on whether or not the palettes are sharing the same window. Palettes can be given their own window (and allowed to float) by dragging the palette's name tab into the drawing area.

Workspace Controls

The Workspace section of the Desktop Property Sheet contains four check boxes that control the display of:

- Edit Bar
- Status Bar
- Scroll Bars
- Rulers

Checking a box displays the corresponding control; clearing the box hides the control.

Tip: It is a good idea to switch off controls that you do not regularly use, to increase the size of your drawing area.

Desktop Colors and Paper Display

The Desktop Color control group contains buttons for customizing the color of the paper, background, and floating palettes. Click the arrow in the drop down box and Select Paper, Background, or Floater. Click the Modify button to change the color using the Color Wheel.

Tip: You can also click the different elements in the preview to bring up the color wheel for that element

See Also: Using the Color Wheel

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:

Show Margins. Check this box to display page (printer) margins; clear the box to hide them.

Show 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.

Toolbars and Menu Property Sheet

Menu: Options|Toolbars and Menus   **Toolbars:** Setup, Standard

The Toolbars and Menu property sheet contains controls that let you select toolbars for display, and modify the appearance of toolbar buttons. Links are also available, in TurboCAD v4 Professional, from this sheet to the customization commands for creating keyboard shortcuts and modifying menus.

Subtopics

Toolbar controls
Menu and Keyboard Controls

Toolbar controls

Toolbars. The Toolbars scrolling list displays the names of all the toolbars available in TurboCAD. Currently displayed toolbars 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 gray-scale 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.

Properties|Update All Styles. Check this box to allow the Properties Toolbar to change all styles for all tools. This function is also available in the View|Toolbars dialog.

The next five buttons, available in TurboCAD v4 Professional, offer commands for creating, modifying and deleting toolbars.

New. Click the new button to create a new toolbar.

Rename. Click this option to rename the selected toolbar.

Delete. Click this box to delete the selected toolbar.

Reset All. Click this box to reset all the toolbars appearing on-screen to the original style.

Customize. Click this box to customize the contents of new or existing toolbars.

Menu and Keyboard Controls

The Menus and Keyboard control group, at the bottom of the Toolbars and Menu property sheet, offers to modify existing menus and keyboard shortcuts, and to return to established defaults for menus and keyboard shortcuts.

Menus Reset. Click this button to reset all menus to program defaults.

*Menus Customize.*Click this button to customize the contents of new or existing menus.

Keyboard Reset. Click this button to reset all keyboard shortcuts to program defaults.

Keyboard Customize. Click this button to create new keyboard shortcuts, or to edit existing keyboard shortcuts.

Auto-Naming Property Sheet

Menu: Options|Auto-Naming   **Toolbars:** Setup, Standard

The Auto-Naming property sheet lets you control whether and how TurboCAD generates names for groups, blocks and symbols, and set whether TurboCAD prompts for group and block names.

Tip: The name of a group, block, or symbol is displayed on the General property sheet of the Properties dialog. To view this dialog, you can simply double-click on the object in the drawing.

See Also: Create Group, Create Block , Properties Dialog

The following controls let you determine how TurboCAD handles the naming of groups:

Generate group names. If this option is checked, TurboCAD 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 TurboCAD will use this string to automatically generate group names. All the characters you type into the box are used literally, except the '@' character, which is a placeholder for a group number that TurboCAD 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, Block and Symbol Numbering .

Prompt for name. Check this option if you want TurboCAD to prompt you for a name every time you create a new group.

Note: If you turn off both the Prompt   for Name and Generate Group Names options, TurboCAD will not name groups.

Tip: 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 click OK to close the dialog.

See Also: Create Group, Explode, Properties Dialog

Subtopics

Block Options
Automatic Group, Block and Symbol Numbering
Symbol Options

Block Options

The following controls let you determine how TurboCAD handles the naming and initial insertion of blocks:

Generate block names. If this option is checked, TurboCAD automatically generates a name whenever you create a block.

Block name prefix. If the Generate Block Names option is on, you can type a string (a series of characters) into the Block Name Prefix text box, and TurboCAD will use this string to automatically generate block names. All the characters that you type into the box are used literally, except the '@' character, which is a placeholder for a block number that TurboCAD 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, Block and Symbol Numbering .

Prompt for name. Check this option if you want TurboCAD to prompt you for a name every time you create a new block.

Insert blocks when creating. If this option is checked, TurboCAD will place an insertion of the block in your drawing at the time that you create it. If this option is cleared, then TurboCAD will create the block, which will appear in the Blocks palette, but will not leave an insertion of the block in your drawing.

Tip: If you clear the Generate Block Names option, TurboCAD will automatically prompt you for block names, since every block must have a unique name.

See Also: Automatic Group, Block and Symbol Numbering, Create Block

Automatic Group, Block and Symbol Numbering

When TurboCAD generates a group, block or symbol name, it automatically gives the group or block a number, which increases by 1 whenever you add a new group or block to the drawing. This ensures that all groups, blocks and symbols have unique names. By default, the names that TurboCAD generates are "Group 1", "Group 2", "Block 1", "Block 2", "Symbol 1," "Symbol 2," and so forth.

You can change the default that TurboCAD uses for automatically naming groups, blocks and symbols in the Group Name Prefix, Block Name Prefix and Symbol Name Prefix text boxes. The names that you type in these text boxes 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 TurboCAD 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: 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.

See Also: Create Group, Create Block

Symbol Options

The following controls let you determine how TurboCAD handles the naming of symbols:

Generate Symbol Name. If this option is checked, TurboCAD will automatically assign a name to each new symbol you create. (Symbols are created by dragging a selection into the symbol palette.) The information in Symbol Name Prefix will be used to assign this name. If the option is not checked, TurboCAD will prompt you to supply a name for each new symbol.

Symbol Name Prefix. If Generate Symbol Name is checked, TurboCAD will use the information in this text box to name new symbols. All the characters that you type into the box are used literally, except the '@' character, which is a placeholder for a sequence number that TurboCAD generates automatically. You can type the '@' character in the name at the place where you want the sequence number to appear.

File Locations Property Sheet

The File Location property sheet allows you to specify the directories where TurboCAD places these different types of program files:

Templates (.TCT). Files in which you save settings, as well as block libraries, symbol libraries, and scripts, that you can start with as the basis of a new drawing.

Scripts (.BAS). Files containing Enable BASIC scripts. This setting specifies the default directory when you choose Tools|Run Script.

Backups (.BAK). These are copies of drawing files that TurboCAD saves, if you check the Make Backup Copies option in the General property sheet.

AutoSave Files (.ASV). These are program files that TurboCAD saves automatically, if this option is turned on in the General property sheet. These will allow you to restore work that would otherwise be lost due to a system crash.

Symbol Library Files (.SLW). These are sets of related symbols, saved to one common file.

Raster Files. These are bit-image files, possibly created by a scanner. The Trace raster-to-vector conversion utility can generate vector images from bit-image (raster) files.

Drawings (.TCW). These are the drawings you create as you use TurboCAD.

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. TurboCAD 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 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.

Symbol Libraries Property Sheet

Menu: Options|Symbol Libraries   **Toolbars:** Setup, Standard

The Symbol Libraries Property Sheet displays the names of all folders which contain symbols, as well as all .SLW files. Use this palette to select the active symbol library, to modify the name of a library folder or file, and to delete or modify symbol library files and/or folders.

New. Click this button to add a folder to the list of folders available for symbol placement.You may specify any folder on any available drive as a symbol library folder.

Modify. Click this button to modify the definition of a selected folder.

Remove. Remove the selected folder from the list of available symbol libraries. This action does not remove a folder from the hard drive, it only removes mention of it from the list of available symbol libraries.

Remove All. Remove all listed folders from the list of available symbol libraries.This action does not remove these folders from the hard drive, it only removes mention of them from the list of available symbol libraries.

Color Palette Property Sheet

Menu: Options|Color Palette

The Color Palette dialog displays all colors currently available in TurboCAD. These colors and their names are displayed in a scrolling list, and also in a palette to the right of the scrolling list. The Color Palette dialog lets you add new colors, modify existing colors, and delete colors from the list.

Subtopics

Adding a New Color

Modifying an Existing Color

Deleting a Color

Using the Color Wheel

Picking Colors with the Mouse

Setting RGB Values

Setting Hue, Saturation, and Value

Adding a New Color

By blending color values (explained in Using The Color Wheel), it is possible to create new colors. 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 Pick a Color dialog.
3. Use the Color Wheel 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 Wheel

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 Wheel.
3. Use the Color Wheel 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 Wheel

Deleting a Color

To delete a color that you no longer need to use in your drawing work:

1. Select the color in the scrolling list or in the palette control.
2. Click the Delete button.

Note: Objects that have already been colored in a color that you delete will still have the deleted color.

Using the Color Wheel

The Color Wheel dialog 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 scroll bar 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 scroll bar.

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 the wheel on 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: Options|Drawing Setup   **Toolbars:** Setup, Standard

The Drawing Setup dialog includes eight property sheets containing settings that are saved in a drawing or template file. These property sheets are summarized in the following table:

Property Sheet	Functions
Display	Set options for optimizing redraw speed.
Grid	Choose the grid type and other grid options.
Units and Scale	Set the unit of measure, the drawing scale, and related options.
Angle	Choose direction, units, and base angle for angular measurements.
Scripts	View and maintain scripts in current drawing or template.
Layers	Create new layers, modify existing layers, and delete layers.
Advanced Grid	Control the major and minor grid origin and frequency.
Advanced Units	Set paper, text, and line width units.

Display Property Sheet

Menu: Options|Display

The Display property sheet contain options for adjusting the quality and redraw speed of the screen display.

Quick text. Check this box to display all text as small boxes.

Quick line styles. Check this box to simplify the display of line styles.

Quick vector brush. Check this box to simplify the display of hatch patterns.

Suppress display of objects smaller than. Type a value to set the size at which TurboCAD will draw a simplified representation of objects. This size is measured in device units (the "device" being your computer's display), so that one inch will be equal to approximately one inch of space on your screen. At smaller sizes, TurboCAD will display entities using a simplified representation, decreasing the amount of time that TurboCAD needs to calculate and display the entity. As you zoom in, you get the detail required for precise editing; as you zoom out, you get faster redraw.

Device Pen Width. This setting controls the printed width of lines set to "zero" value in a drawing.

Tip: If you are working in a very large drawing or if you are using a relatively slow system, you may be able to increase your screen redraw speed by clicking one or more of the options in the Performance control group of this property sheet. The amount of speed that you gain depends on the amount of text, styled lines, and fill patterns that your drawing contains.

Grid Property Sheet

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 control grid type (Ortho, Isometric, or Polar), display style (Points, Crosses, and Lines), the distance between grid points (or intersections), and the grid color.

Subtopics

Grid Type
Grid Color
Grid Style
Grid Spacing
Other Grid Controls

Grid Type

Click the appropriate option buttons in the Type area of the property sheet to set the grid to any one of the following types:

Ortho. Choose this option for an orthogonal grid. Points of the orthogonal grid are aligned parallel to the X and Y axes of the Cartesian plane.

Isometric. Choose this option for an isometric grid. Points in the isometric grid are aligned along lines measured 30º, 90º, and 150º from the horizontal axis. This type of grid is used in isometric drawing, which uses perspective drawing techniques to represent three-dimensional objects.

Polar. Points in the polar grid radiate from the origin, and are aligned by their radial distance from the origin and their angular distance from the horizontal axis.

Tip: Advanced users can set the base angle of the grid to something other than horizontal, using the Base setting in the Advanced Grid property sheet.

See Also: Coordinate Systems

Grid Color

Click the buttons in the Color group to choose a new color for the major and minor grid. Click the Major or Minor button, then choose a color using the Color Wheel.

See Also: Using the Color Wheel

Grid Style

Choose from the following Grid Style options:

Points. Click this option to display the grid as a matrix of points.

Crosses. Click this option to display the grid as a matrix of crosses.

Lines. Click this option to display the grid as intersecting lines, similar to graph paper.

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.

If you have chosen an orthogonal or polar grid in the Grid Type control:

1. Type the amount of horizontal space between each grid point in the X box.
2. Type the amount of vertical space between each grid point in the Y box.

If you have chosen a polar grid:

1. Type the angular distance (the distance between the points along a circle around the center of the grid) in the Angular box.
2. Type the radial distance (the distance measured along a line radiating from the center of the grid) in the Radial box.

Note: 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.

Other Grid Controls

The Grid property sheet also contains these controls:

Hide grid when finer than... 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|Display from the menu.

Advanced Grid Property Sheet

The Advanced Grid property sheet is not listed in the menu; you must select it from the Drawing Setup dialog. This property sheet gives you precise control of:

- Grid *frequency,* which determines how visible grid lines alternate with invisible grid lines.
- The *minor grid,* or the finer grid points that appear between points on the major grid.
- The grid *origin,* which lets you control how the grid aligns itself to the coordinate system and to drawing objects.

Subtopics

Grid Frequency

Minor Grid Settings

Grid Origin

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|Display to deselect grid display, then the whole grid will be invisible.

If you are customizing an orthogonal or isometric grid:

1. Type a value in the X box to set the frequency of horizontal grid points.
2. Type a value in the Y box to set the frequency of vertical grid points.

If you are customizing a polar grid:

1. Type a value in the Angular box to set the angular frequency of grid points (the frequency of points arranged in a circle around the center of the grid).
2. Type a value in the Radius box to set the radial frequency of grid points (the frequency of points arranged along a line radiating from the center of the grid).

Minor Grid Settings

There are two control groups for minor grid settings: Minor Grid X, for setting horizontal minor grid points, and Minor Grid Y, for setting vertical minor grid points. You change the settings in these two control groups independently, using identical procedures. Each group contains three option buttons:

None. Select this option to turn the minor grid off. This means that all of the grid points or lines will be displayed as major grid points.

Division. Select this option to turn the minor grid on. You can set the number of divisions between each major grid point using the spin button control near the option button.

Offset. Select this option to create minor grid points at a fixed distance (offset) from each major grid point. You can use positive or negative numbers.

Grid Origin

The grid origin is located by default at the origin of the absolute or polar coordinate system, which is at coordinate (0,0). You can reset the origin point of the grid, causing the grid to be offset from the coordinate system locations. You can also change the base angle of the grid, aligning it to an object that is not oriented orthogonally in the drawing space (for example, a house that is oriented at an angle in its lot).

To reset the grid origin, type absolute coordinate values into the X and Y text boxes. To change the base angle of the grid, enter the angle in the Base text box.

Note: Grid Origin is also accessible from a tool in the Grid toolbar.

Tip: This feature allows you to use a "local" grid. If you are working in an area of your drawing you can relocate the grid origin so that it aligns to the set of objects you are working on, rather than to the drawing space as a whole.

Units and Scale Property Sheet

The Units and Size property sheet contains options for setting the drawing scale and for controlling the display of linear units.

Tip: TurboCAD'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.

Subtopics

Measuring System
Scale
Units in Work Space
Use Paper Space Coordinates

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 modeled (World units). TurboCAD 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:

System	Example
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:

Format	Example
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 TurboCAD 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.

Tip: Checking and unchecking this box is exactly the same as clicking the Paper/World button in the upper-left corner of the Ruler.

Advanced Units Property Sheet

The Advanced Units property is not listed in the menu, nor is there a toolbar button for it. You must select it when the Drawing Setup dialog is active. This property sheet contains controls for setting units in Paper space.

The property sheet contains the following controls:

Paper Units. Click in this drop-down list and scroll to choose a unit for setting and displaying Paper measurements on the Ruler, in the coordinate system, on the grid, and in the Edit Bar.

Text Units. Click in this drop-down list and scroll to choose a unit for setting and displaying the size of text.

Line Width Units. Click in this drop-down list and scroll to choose a unit for setting and displaying the width of lines in the Pen property sheet of the Properties dialog.

Angle Property Sheet

Menu: Options|Angle   **Toolbars:** Setup, Standard

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.

Subtopics

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 TurboCAD 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 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. TurboCAD's default sets the step angle to 90 degrees.

Scripts Property Sheet

Menu: Options|Scripts   **Toolbars:** Setup, Standard

The Scripts property sheet associates descriptive names with BASIC script files, and allows users to browse descriptions embedded within script files. For specific information about the Enable Basic language, or about how to access TurboCAD's internal functions, use Help, or use the manual entitled "Programming TurboCAD with Enable Basic" shipped in the box with TurboCAD v4 Professional and on the CD with TurboCAD v4.

Subtopics

Adding a New Script

Deleting a Script

Debugging a Script

Adding a New Script

The Add button lets you associate a descriptive name with a script file and run the script using the descriptive name from the Script toolbar or menu.

1. Type a name for the script in the Name text box.
2. Click the Add button. TurboCAD will display the standard Open dialog. Use the dialog to choose the script's .BAS file, then click OK to return to the Drawing Setup dialog.

When you select a script, the name of the script, as well as its file name and path, appear in the scripts list box. If descriptive information is available in the script's file, this information will appear in the Description area of the property sheet. Once the script is added to your drawing, it will appear in a submenu to the right of the Tools|Run Scripts menu. You can run the script immediately by choosing it from the submenu.

Deleting a Script

To delete a script's descriptive name from the list, select its name in the scripts list box, then click the Delete button.

Note: Deleting a script removes the script's association with the drawing, but leaves the script's .BAS file intact, so you can add it to your drawing or template again at any time.

Debugging a Script

To debug a script, select its name in the scripts list box, then click the Debug button.

Layers Property Sheet

Menu: Options|Layers   **Toolbars:** Setup, Standard

The Layers property sheet allows you to create new layers, set the current drawing layer, set the properties of layers, and delete layers. Each layer property you can modify has its own column in the property sheet.

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 modeling, 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.

Use the Layers property sheet to establish new layers for your drawing, and to set their characteristics. Use the Layers dialog box in the Properties toolbar to change the active layer as you draw.

Note: In the previous version of TurboCAD, it was possible to set the active layer in the Layers Property Sheet. In this version you do so only from the Properties toolbar.

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.

See Also: Format Property Sheet

Note: 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.

Subtopics

Creating a New Layer
Setting the Current Drawing Layer
Setting Layer Properties
Deleting a Layer

Creating a New Layer

To create a new layer, click the New button, then type a name for the layer in the Name text box. If you want the layer to be automatically assigned a sequential layer number, Click OK without changing the name. The @ symbol represents the sequential layer number that TurboCAD will assign. When you click OK, the new layer name will appear in the Layers dialog list box. You may then adjust the various layer settings.

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 letter, select the name of the layer from the Layers list box in the Properties toolbar. (To add the Properties toolbar to the desktop, select View|Toolbars from the menu,   and select Properties from the list of available toolbars.)

Setting Layer Properties

You can set the properties of a layer using the controls in the Properties area of the Layers dialog. Notice that the dialog box scrolls horizontally -- not all the columns of layer properties are visible at one time.

Visible. The "eyes" icon at the top of the first column represents setting the visibility status of a layer. Check this box to make the layer visible and uncheck it to make the layer invisible. The All Visible button, below the Layers dialog screen, can set all layers to visible or not visible at one time.

Read Only. The yellow padlock icon at the top of the second column represents setting the edit status of a layer. If the box for a particular layer it is not checked,   you can draw, insert, and edit objects on the layer. If the box is checked, then you can only view the layer, but you can not change it.

Note: 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.

See Also: Bring to Front, Format Property Sheet, Send to Back

Layer. Use this item to rename existing layers, by clicking on the layer name and typing a new name. Click on the OK button directly below the list to accept the change.

Color. Use this item to select a color for the layer. Black is the default color for new layers. To change the color, click on the color bar for the layer you wish to change. Select the new color from the list box that appears. 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.

Style. You may need to scroll horizontally to see this item. Use this item to select a line style for the layer. Click on the line for the layer you wish to change. Select a new line style from the list box that appears. 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 item to assign an order number to the layers. Order numbers determine the order in which layers are drawn. 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. To change the order number for a layer, click on the number. A set of horizontal scroll arrows will appear. You may type in a number, or use the scroll arrows, to set the value.

After you have made changes to the properties of the selected layer, click the OK button to apply the changes.

Deleting a Layer

To delete a layer, you must first clear any objects on the layer. You can select the objects on a layer using the Edit|Select By|Layer dialog, then clear the selected objects. Once the layer is cleared of objects, you can delete it in the Layers dialog by selecting its name from the list and clicking the Delete button.

See Also: Select by Layer Dialog

Working with Multiple Windows

TurboCAD uses a *multiple document interface*. This means that the TurboCAD application window can contain multiple document windows, each of which contains a TurboCAD drawing. Whenever you open a TurboCAD drawing or start a new TurboCAD drawing, the drawing will appear in its own window, within the TurboCAD application window. You may also use multiple windows to display varying views of the same drawing.

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, 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.

Subtopics

[Moving and Copying Objects between Drawings](#)
[Window Menu Commands](#)

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.

Note: 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.

Tip: 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 completely visible, without overlap.

Subtopics

Copying/Cutting and Pasting Objects between Drawings

Drag and Drop

Using Multiple Windows With One Drawing

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 also remove the selected objects from the source document.)
3. Activate the target document by clicking in its window (or on its title bar), 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 block and symbol libraries from their library palette into another drawing.

Subtopics

[Dragging and Dropping between Drawings](#)
[Dragging and Dropping Blocks Between Drawings](#)

Dragging and Dropping 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 Blocks Between Drawings

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

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

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

Tip: You can use this same drag-and-drop technique to place symbols, blocks or drawing elements in other Windows applications, such as Microsoft Word.

Using Multiple Windows With One Drawing

When working on a large and complicated drawing, it is often useful to have simultaneous close-up views of more than one part of the drawing. In TurboCAD v4, not only can you have multiple windows open on the same drawing, you can also start any drawing or editing command in one window and finish in another. For example, you may need to draw a line connecting two widely separated objects. Open a second window on the drawing using Windows|New Window. Then tile the windows (Windows|Tile). Adjust the zoom value in each window to get a close-up of each item. Then start the line in the first window, and finish it by clicking at the correct location in the second window.

Note: When you use multiple windows with one drawing, you are still creating one drawing, not multiple copies.

Window Menu Commands

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

Command	Function
New Window	Opens a new window on the existing drawing.
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 the 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.

New Window

Menu: Window|New Window   **Toolbar:** Standard, Window

Use the New Window command to open a new window on the currently active drawing. If the current window is maximized when you use New Window, the new view will also be maximized. If you tile the new window, you will be able to set two different zoom views of the same drawing, and draw from one window to another.

Tile

Use the Tile command to arrange all the open drawing windows into a mosaic. The windows are resized so they all are 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.

Tip: You may want to turn off rulers and scroll bars (in the View menu) to gain more screen space while working with multiple windows.

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

When you minimize a drawing window, Windows shrinks it to an icon. Use Arrange Icons to line up these minimized windows along the bottom of the application window.

Close All

Menu: Window|Close All   **Toolbar:** Standard, Window

Use Close All to close every open drawing window.

Note: TurboCAD 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 to return to TurboCAD without closing the drawing windows.

Changing the Active Window

TurboCAD 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, TurboCAD makes it the active window. This is the same as clicking in a window, or clicking a window's title bar.

Tip: You may prefer to click the title bar if you are in the middle of a drawing or editing command, so you do not interrupt the command while changing windows.