

Command and Macro[®] Language Reference Manual

USER MANUAL

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CHAPTER 1 - INTRODUCTION

About This Manual

TurboCAD is basically a "command driven" CAD system. This means that everything you do in TurboCAD is accomplished by entering commands. Commands, in turn, have Options which determine precisely how a command is to accomplish its mission. Options generally have a definition and a list or range of acceptable Values.

When you run TurboCAD on your Macintosh, which provides for icons ("tools"), pull-down menus, and dialogs, this interface merely is a way to get TurboCAD to execute commands for you. If you want to really become an expert in TurboCAD, knowledge of the TurboCAD commands and what they can do is essential. Furthermore, a friendly user interface, no matter how extensive, can never give you access to the full power of commands.

This manual is concerned primarily with defining all of the TurboCAD commands, their function, the options available in each of them, the acceptable formats in which commands and options must be entered, and the additional capabilities TurboCAD gives you to execute commands efficiently and promptly. Details regarding Icons, Menus, and Dialogs are found in the User Manual.

Typographical Conventions

The characters described below have been used throughout this manual to represent a specific item, function or phrase.

Single Quote ' '

Characters or words enclosed in single quotes represent a key on your keyboard. Usually you are to press the key on your keyboard that is labeled like the quoted string. Example: A <Return> will terminate the command.

Double Quote " "

Words used to describe a special function or word that have a meaning unique to the TurboCAD environment will be shown in double quotes. An explanation will usually accompany these words. Many of them are also listed in the Glossary.

Back Quote ` `

Character strings that must be quoted when entered on the TurboCAD command line are shown in back quotes. This is exactly the way they must be quoted when entered on the command line to be read as a literal string by the command processor. See the Command Entry section for more information on quoting literal strings.

Back Slash \ \

Performs the same function as the back quote, except that back slashes allow variable substitution. See the Command Entry section for more information on enclosing strings between back slashes.

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Parentheses ()

Parentheses in the documentation contain a side comment relating to the contents of the paragraph where they appear. Its usually just some comment appropriate to the situation.

Capitalized Words and Letters

Capitalized words in the middle of a sentence are usually the name of a TurboCAD command or option. This is just to make sure that the word is recognized as such. In the Commands chapter, capitalization of some of the letters in a command name or option describes the minimum characters that must be typed to access that command or option to ensure no conflict with another command or option.

CHAPTER 2 - BASICS

Setting Up a Drawing

Creating a New Drawing (New... selection on File Menu)

The first thing that happens when you fire up TurboCAD is that the program expects you to set up a New or "blank" drawing. The "environment" for this blank drawing is the result of your choice for parameters which define the size, units, scale, etc. of the drawing. When TurboCAD is fired up via an existing drawing, the resulting environment is that of the drawing as it was saved.

TurboCAD forces you to execute the New command automatically in order to set up a new drawing. This command is the same command you would execute any time you want to start a new drawing. The New command has the following options with their corresponding preset or "startup" environmental values. These are the values which would be preset the first time you fire up TurboCAD.

Width	34
Height	22
Scale	1
Mscale	1
Work Units	Inches
Plot Units	Inches
Resolution	Fractional

TurboCAD gives you the opportunity to change any or all of these values before the blank drawing is created. The meaning of each of these options is as follows:

Width and **Height** are the width (horizontal) and height (vertical) of the drawing you want as expressed in Plot Units. This is equivalent to the sheet size you would use to plot the drawing on a pen plotter.

Scale is the ratio you desire between Plot Units and Work Units, disregarding the fact that these units may be different. This ratio is commonly referred to as the English Scale. For example, if you set the Work Units to feet, the Plot Units to inches, and the Scale to 1/8 (.125), then you are working at 1/8"=1'.

Mscale is also the ratio you desire between Plot Units and Work Units but taking into account the units themselves. This ratio is commonly referred to as the Absolute (or Metric) scale, and is equivalent to the ratio between the size of an object in real life and its size on the drawing. In the example above, the Mscale would be 96, because working at 1/8"=1' is really the same as saying that you are going to draw things 96 times smaller than real life. You can calculate the Mscale by taking the expression for the English scale, converting both sides to

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and dividing the right side by the left side. The example above would yield the expression $1/8"=12"$ giving the result of 96.

The **NEW** command only allows you to set either the Scale or Mscale, and it automatically calculates the other. If you are not familiar or comfortable with these concepts of scale, take the time to sort them out until you are comfortable.

Work Units are just that -- the units in which you want to work. In the example above, you would have set the Work Units to feet. This means that when you enter a distance without specifying the units, TurboCAD will assume that you mean feet. That distance will then appear on your drawing at the proper scale. If you enter a distance and specify units other than the Work Units, TurboCAD will still read them correctly. Furthermore, TurboCAD allows you to enter any valid calculator expression for a distance or number. In the example above, the values 3.5, 3.5', 3+.5, 42", and unit(1066.8 mm ft) would all yield the same result. See the Calculator command for specific details as to all of the acceptable formats for entering numbers, distances, and calculator expressions.

Plot Units are simply the units you want your drawing width and height measured in. Most of the time you will set this to either IN or MM.

Resolution determines whether you want to work in a Decimal or Fractional environment. If you choose Fractional, TurboCAD will give you distances in fractional format when you query for a distance or location. Likewise, choosing Decimal will cause TurboCAD to give you distances or locations in decimal format. The choice of one or the other does not affect the fact that in TurboCAD you can enter distances in either format. See then NEW command for more information on Resolution.

See the SETUP command for details on how to change any or all of these environmental parameters after a drawing has been set up as above.

Preferences (Preferences... selection on Setup Menu)

After a drawing has been set up, the Preferences command gives you additional control over your drawing environment. This command has numerous (28) options, and it would be confusing to go into all of them at this time. However, you should be at least familiar with the following in order to avoid confusion. They are listed with the TurboCAD factory or "startup" values.

Zoom	.5
Lock	15
Polar	0
Searcharea	.13
Cursor	Crosshairs
Rounding	1/1024
Format	Fractional
Store	None

Zoom is the factor by which objects on your drawing will be enlarged every time you "zoom in" by using the "Z" key. A value of .5 means that you will see things twice as large each time

you "zoom in". Likewise, it means that objects will become twice as small when you "zoom out" by using the "U" key. Most people find the value of .5 to be just right. See the **Locating Points** section below for more information on using keys.

Lock is the angle increment to which the crosshairs will snap if you are locating points by using the <A> (angle lock) key. You may want to set this to a smaller or larger value depending on the particular situation. To get a feel for this, start a line (i.e. you are "rubberbanding" from a point), then press the <A> key on your keyboard and move the crosshairs in a circular path around the starting point. You should see the rubberbanding snap to the Lock angle.

Polar is the angle which you want to be considered the "zero" angle when entering angles in Polar coordinates (i.e. distance, angle). At startup, TurboCAD uses a standard engineering (counterclockwise) convention for angles. That is, 0 degrees means horizontal - right), 90 degrees means vertical - up, 180 degrees means horizontal - left, and 270 degrees means vertical - down. The angle entered for this option determines what the "zero" angle will be. For example, an entry of 90 will mean that from then on a polar coordinate entered with an angle value of 0 will mean "up". See the **Entering Coordinates** section for more details on polar coordinates.

Searcharea is a subjective area beneath the crosshairs which will be searched when you ask TurboCAD to "search" for a vertex (endpoint), object, etc. The "startup" value of .13 was arrived at after considerable experience, so you will typically leave it as it is unless you are having trouble during searches. Keep in mind that when TurboCAD does a search, it gives you the "first" object found within the search area rather than the "best" object found.

Cursor is simply the type of cursor you want to use - i.e. Crosshairs, X, etc. Most people just leave it at Crosshairs. Remember that you can change the type of cursor "on the fly" by entering the "" key while the crosshairs are active. See the **Locating Points** section for more information.

Rounding is a tricky one that can cause some confusion. First of all, it should not be confused with Grid Snap (Cursor Rounding), which is the amount you want the cursor (i.e. the crosshairs) to "jump" on the screen and which is controlled by the Rounding command. Rounding in Preferences means the increment to which you want TurboCAD to "round" numerical values (i.e. distances, coordinate readouts, etc.). For example, if working in a fractional environment, you may not want to see distances displayed to the nearest 1/1024 th of a unit. Instead, you may want to see them displayed to the nearest 1/64 th or 1/32 th of a unit. Remember that this Rounding value is interpreted as relating to Work Units unless you specify otherwise. Using the sample (1/8"=1') New drawing which we set up above, entering a value of 1/32 for Rounding means that numerical values will be rounded to the nearest 1/32 th of a Foot since Work Units were set to Feet. This will yield an impractical result since 1/32 th of a foot is 3/8", and having values "rounded" to the nearest multiple of 3/8" is very impractical. If you really want numerical values which are calculated or displayed by TurboCAD to be "rounded" to the nearest 1/32 th of an Inch, then enter 1/32" (with an " mark) to get the desired results. One problem is that the Preferences rounding affects the current Cursor Rounding value. To avoid conflict, make sure the Cursor Rounding is set to a value greater than or equal to, and a multiple of, the Preferences rounding. For example, Preferences rounding of 1/32" and Cursor Rounding of 1/8" works well.

Format works in conjunction with Rounding and can cause unexpected results if set improperly. Valid values for Format are Fractional, Integer, One, Two, Three, Four, Five, Six, or Float. Fractional is just fractional and control is really in the hands of the Rounding value. The other seven are decimal and range from zero decimal places (Integer) to as many as

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TurboCAD can give you (Float). Typically if you are working in a Fractional environment, this option should just be set to Fractional, unless you have a need to see numerical values (distances, etc.) in decimal format. If this is the case, remember that those distances will be expressed as the decimal equivalent of the fractional distance values. If you are working in a decimal environment, the Format option will override the Rounding option if it is set to a lesser number of decimal places than the Rounding option. The best value for this option in a decimal environment is Float, unless you have a specific need to see numerical values with a limited number of decimal places or in fractional format.

Store really doesn't control your current drawing environment but can have a material effect on how things are set the next time you fire up TurboCAD. It controls how the settings of TurboCAD "dialogs" are set the next time you fire up TurboCAD. Valid values are None, Quit, or Immediately. None means that they will be set to factory defaults. Quit means they will be set the same as when you next Quit the program (or Application), Immediately means they will be remembered as soon as the Preferences command is executed and will set just as they are at the time. Most users prefer Quit. The factory default is None.

Rounding (Grid Snap selection on Setup Menu)

Rounding is the increment to which the TurboCAD crosshairs will "snap" to on the screen, and we commonly refer to it as Grid Snap or Cursor Rounding. Imagine that you have an imaginary grid of points on your drawing and that the crosshairs are restricted to "snapping" only to those points. The distance between those points (both in the X and the Y axes) is controlled by the Rounding command and its options. Although it is possible to set the increment different for the X and Y axes, it is seldom practical to do so. Factory default values are 1/8" for fractional drawings and .1 mm for metric drawings. These values are interpreted in Work Units, so specifying the units (i.e. the " mark) is important.

Remember that the Rounding option of the Preferences command can affect the current rounding values, so make sure that it is set properly first (i.e. set the Cursor Rounding equal to or larger than, and a multiple of, the Preferences rounding.) Keep in mind also that Cursor Rounding can be "toggled" on and off via the "R" key. See Locating Points for more information on use of Keys. If you are an architect and never want to draw anything with the crosshairs to a tolerance less than, say, 1/2", then a Cursor Rounding value of 1/2" is appropriate. Likewise, an engineer laying out a circuit board may want to use Cursor Rounding to limit his design to minimum distances. Cursor rounding does not affect the accuracy of distances or coordinates which are keyed in. Those go in exactly as entered regardless of Cursor Rounding. Cursor Rounding is independent of Reference Grid settings as defined below.

Grids (Grids selection on Setup Menu)

Grids are "reference grids" which appear on your screen to help you determine distances while creating a drawing. The distance between grid lines is controlled by the GRID command, and their color is controlled by the Colors command. They appear on your screen but are never printed or plotted.

For both axes, TurboCAD allows you to define "Small" grid lines and "Large" grid lines. Think of a sheet of "gridded" vellum. Typically there are light grid lines at some small increment (i.e. 1/8", 1/4", 1/10") and slightly heavier grid lines at some larger increment (i.e. every 1"). TurboCAD interprets values entered for grid distances in Work Units.

For example, if your Work Units are feet and you are working at 1/8"=1', you may want to set up a light grid with increments every 1' (equivalent to 1/8" on the paper) and a heavier grid with increments every 10' (equivalent to 1.25" on the paper). Since the values are interpreted in Work Units, the a value of 1 for the small grid spacing and a value of 10 for the large grid spacing would do it. You could also enter 12" and 120" to achieve the same result. If your Work Units are the same as your Plot Units and you are working at a scale of 1"=1" (i.e. 1:1), then you would enter 1/8" and 1.25" to obtain a grid which is physically the same as above.

Factory defaults for Grid increments are .25" (metric .1 mm) for the Small increment and 1" (metric 1 mm) for the Large increment. These are typically too small when you are working at scale, so you may have to "zoom in" a lot until you can see the small grids.

For convenience, TurboCAD allows you to make Grid increment values a multiple of (or fraction of) Cursor Rounding values. See the Grid command for more information on how to do this. Grid color is controlled by the Colors command as described below.

Colors (Colors ... selection on Setup Menu)

Everything you draw in TurboCAD has a "Pen" (a number between 0 and 255) associated with it. You will learn more about Pens in the Drawing Structure and Common Options sections and by studying the Pen command. In turn, every Pen number has a specific color associated with it. The assignment of Colors to Pens is controlled by the Colors command or on the colors palette. The current Pen you are drawing in is displayed on the Status palette.

Getting Status and Help (only accessible via commands)

So far you have learned how to set up a drawing and its environment. You have used (or have accessed via a menu) , at least the New, Setup, Preferences, Rounding, Grid, and Colors commands. You should also know how to find out what the options of these commands are set to at a particular time. This is what the Status command is for. The Status command (or its acceptable abbreviations, STA or SS) followed by the name of any command will give you information about the current values of the options of that command. For example, typing "SS GRID" followed by a <Return> will give you information about the current Grid option settings. On computers which have an Icon and Menu interface in addition to a Command Line you can generally find out the same information by accessing the command, looking at the "Dialog" which appears on the screen, and then clicking on the cancel button if you do not want to make any changes.

The Help command gives you information about the options and acceptable values for those options for any command. What you will see is an abbreviated version of the full option definitions and values contained in this manual. It works the same way as the STATUS command. Furthermore, entering Help followed by a 'return' (no specific command) will give you a list of all of the TurboCAD commands and their command codes. This is intended for those who are familiar with the contents of this manual.

Drawing Structure

Primitives, Groups, and Symbols

TurboCAD is not a paint package—its drawing data is "object"-oriented. Therefore, there are no pixel manipulation commands in TurboCAD, only object creation and manipulation commands. Just remember that TurboCAD thinks in terms of "objects. ”

Objects are either "primitives" or "groups. ” Primitives are the basic building blocks of a TurboCAD drawing. They are the simplest geometric objects in a drawing file. All commands which create objects create primitives of a specific type. For example, the Line command creates line primitives, the Circle command creates circle primitives, and so forth. The types of unique primitives are: Line, Doubleline, Leaderline, Fillet/Chamfer, Text, Rectangle, Circle/Arc, Ellipse, Polygon, Point, Dimension and Wireframe. The commands used to create these are accessed either by typing them in on the Command Line or accessing them via the Icon Menu. See the Operational Overview section for more information on the Icon Menu. As you can see, TurboCAD has more unique primitives than most CAD systems. The advantages of this will become more obvious as you work with TurboCAD.

Groups are primitives and/or groups "glued" together to form a single object. Groups can be hierarchical. That is, a group can contain other groups (and primitives) that contain more groups and so on. Editing (i.e. changing) can be done on either the primitives within a group (called members) or on the entire group itself. Each time that a larger group is made of smaller groups you have the choice of adding another "tier" to the group's hierarchy or having these groups blend together without tiers. Groups can be "ungrouped" (i.e. exploded) either by tiers or all the way down to primitives. Members of a group do not have to be physically close on the drawing (i.e. a group can contain two objects which are anywhere on the drawing). Grouping and ungrouping are controlled by the Group and Ungroup commands, which can also be accessed via the Icon Menu.

Currently TurboCAD cannot edit at intermediate tiers. This means that you could move, for example, the primitives (called members) of a group or the whole group (called an object), but not an intermediate tier of the group. To reach lower tiers for editing, you would have to ungroup the higher tiers.

Symbols are merely groups (or primitives) which have been saved to disk as such for later use on other drawings. Typical symbols are north arrows, graphic scales, electronic components, architectural details, etc. Symbols are saved and retrieved via the Put and Get commands which are also accessible via the Symbol Menu.

In TurboCAD, one must also be familiar with the concept of multi-segment lines. Lines (and other linear primitives like doublelines and fillets) can contain large numbers of segments in a single primitive.

Layers

Every primitive you create in TurboCAD has a Class, Layer, Pen, Style, and Priority assigned to it. Unless you specify otherwise, the assignments given to a given primitive are those displayed in the Status area at the time the primitive was created. You will learn more about setting these values in the Common Options chapter in this section.

Layers are pretty much of a standard in the CAD industry. They correspond to "overlays" in conventional drafting. Each TurboCAD drawing can contain up to 256 layers, numbered 0 through 255. You can display or suppress the display of one or any number of layers at any time. At the minimum, you must display the single layer on which you are working (i.e. the layer number which appears on the Status area). Data on layers which are not displayed is not editable (i.e. cannot be affected—moved, deleted, or modified—in any way). The current layer assignment is controlled either via the Layer command or the Work command. Display of layers is controlled either via the Display command or the Layer command. These can also be accessed via the Layers. selection in the Setup Menu.

Classes

Classes are similar to layers and give you a greater degree of flexibility when used in conjunction with layers. They should be considered a complement to layers. Each TurboCAD drawing can contain up to 256 classes, numbered 0 through 255. One difference is that groups have a single Class assignment as a whole, while the primitives which comprise the group retain their Layer assignments. Another difference is that classes can be made uneditable even if they are displayed. Another difference is that you can save any combination of classes (excluding all others) into a separate drawing, or merge the contents of a saved drawing into a specific class of the current drawing. Another difference is that you can search any combination of Classes (excluding all others) when using commands such as the Count or List command, which count and list groups which have been assigned "Tags. " Most users begin by using just layers or just classes. As they become more experienced, they find that utilization of both is most advantageous.

Pens

Pen assignments control the Color and Width of lines. In TurboCAD, colors are assigned to pens as was explained in the Colors paragraph of the Setting Up a Drawing chapter in this section. Line widths are also assigned to Pens via the Width command. This command can be also accessed via the Pens/Line Weight... selection in the Setup Menu. For more information on line Widths, see the Width command.

Priorities

TurboCAD allows you to assign a Priority between 0 and 255 to any object. The TurboCAD "default" priority is 125, unless you change it via the Priority command. If two objects overlap, the one with the highest priority will be the one found if you place the crosshairs where they overlap and try to locate a point on the object. For more information on how to assign and change Priorities, see the **Locating Points** and **Common Options** chapters in this section and the Priority command.

Worksets

TurboCAD allows you to assign a name of your own choice to a any group of values for Class, Layer, Pen, Style, and Priority. We call these groups Worksets. For example, a designer may wish to assign a specific name (i.e. "Walls" or "Components") to Class 5, Layer 5, Pen 3, and Solid line style. Such names are assigned via the "Workset" option of the Work command. Once defined, the Workset name can be chosen instead of individually choosing a Class, Layer, and so forth. Worksets can also be defined via the Worksets selection in the Setup Menu and selected for use on the Status Palette. For more information, see the **Common Options** chapter in this section and the Work command.

Command & Graphic Modes

When working in TurboCAD, you are always in one of two modes, Command or Graphic.

Command Mode

The command mode means that you are not currently "in" a command, the program is waiting for a command to be typed. You are in the command mode any time that your crosshairs are not being displayed. The standard Macintosh pointer will be displayed. While in the command mode, the Prompt line will read "Select from a palette or menu." You cannot pan or zoom or perform any of the graphic mode functions while in the command mode before a command has been entered. Clicking the mouse button once under this condition will automatically put you in the Line command.

Graphic Mode

The graphic mode means that you are currently "in" a command, the program is waiting for you to locate points or pick objects. You can recognize the graphic mode by the fact that the crosshairs are displayed. While in the graphic mode, the Prompt line will display messages telling you what you are to do next—locate a point, pick an object, etc. You cannot type commands while you are in the graphic mode. You must first exit the command you are currently in by pressing the <Return> key. Some commands, like Text, require more than one <Return> to exit.

Command Line

The TurboCAD command line is contained in the Prompt palette. This is the line where commands appear as you type them in. The command you are executing will remain displayed on the line as long as you are in that command. The line will be blanked when you exit the command to enter a new command. Therefore, the line will be initially blank when in the command mode and will display the command you have entered when in the graphic mode. (Command mode and graphic mode are described above.)

The Command Line is actually the bottom line of a scrolling log that is automatically kept of all the commands that have been entered. Visualize that previously entered commands have disappeared from view by being moved up and off of the Command Line. You can scroll through this command log by activating the Commands window. You can execute any line in the log by highlighting it with the cursor and pressing the <Enter> or <Return> key. You can edit the line by moving the text cursor and adding or deleting text, and then execute it in the same manner.

Entering Commands

This section describes how to enter commands onto the TurboCAD command line. It does not describe how commands are executed from the user interface icon menus, pull-down menus or dialogs. If you want to see which command and options you are executing when you use an icon or menu, simply look on the command line or command window.

The TurboCAD command and Macro language has an English language format. For example, to draw a line you enter the command "Line" and to move an object you type "Move. " Each command also has options that can be entered after the command. When you have correctly entered a command name and the options you desire, you press the <Return> or <Enter> key to execute the command.

Entering Command Names

You will notice by looking at the Command List in the Commands chapter that TurboCAD commands are either one or two words long. An example of a single word command is "Move," while an example of a two word command is "Move Vertex." TurboCAD commands can be entered by typing them in their entirety or by abbreviating them to the portion shown capitalized. You can abbreviate each word of two word commands as shown. A space or tab can be used to separate the first and second word. A comma cannot be used because the item following the comma will be read as the first option of the command instead of the second word of the command name itself. The two words can be strung together into a single word by separating them with an underscore (_). However, when entered this way, the minimum abbreviation requires that the first word be spelled out.

All TurboCAD commands also have a unique code that can be entered instead of the command name or abbreviation. One word commands have a two character code and two word commands have a three character code. Options do not have codes, they are just abbreviated.

Case is not important when entering TurboCAD commands or options. Entering "Line", "line", "LINE" or "LiNe" will all be read the same. Likewise, if you are entering the name of a disk file as part of a command, case is unimportant because the Macintosh operating system does not differentiate between upper a lower case letters.

Entering Options

Options are entered on the command line after the command. You must separate options from one another and from the command with either a comma, space, or tab character. If you use the "keyword=value" format, options can be entered in any order. Otherwise, their position is important.

Most options are comprised of three parts; the keyword, the assignment and the value. The keyword is the name of the option. It can be typed in full or abbreviated to the minimum characters shown capitalized in the documentation. Some options are keyword-only options. This means that the option has no value to be set, the option is only active when it is specifically called for on the command line. "Line Convert" is a good example. If the Convert option is not entered, the command defaults to drawing lines, not converting objects into a line.

The assignment part of an option is always an equals sign (=). It must always be entered between the keyword and the value. Entering spaces or tabs between the keyword and the

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assignment or between the assignment and the value is legal. For example, "Radius=2" and "Radius = 2" will be read the same.

The last part of an option is the value that the option will be set to. Everything from the assignment to the next option will be read as the value. Values are typically either a number or specific word. If it is a number, the documentation for the option will tell you the valid number range and format. For example, all Pen option values are limited to a whole number between 0 and 255. You can also enter any valid calculator expression, whose result will be used for the value. If the value is a word, the valid words will be listed and explained in the documentation. Sometimes the value is the name that you wish to assign to a symbol or drawing. Because these file names often contain spaces, they should be placed in quotes. See **Quoted Strings** below.

Positional Order

In addition to entering options in the keyword-equals-value format they can be entered in a fixed positional order. Fixed positional order means that you can enter just the values of a command's options as long as they are entered in the order shown in the documentation. The command processor always checks the command line for positional order values before looking for keywords. Once a keyword is entered onto the command line, the positional order is broken and all following options on the command line must be in the keyword-equals-value format. For example, entering "Circle, Rad,2" will set the Construction option of the Circle command to Radius and the Size option to two.

Delimiters

Delimiters are the characters that are used to separate the command and options entered onto the command line. TurboCAD allows you to use spaces (space bar), tabs, and commas. Spaces and tabs are considered "soft" delimiters. When using spaces and tabs, any amount of white space on the command line is read as one delimiter. So, one space is as good as twenty. It does not matter if that white space is three tabs and four spaces or a single space.

Note: Because variable substitution and calculations are done before the command line is scanned for positional order and keywords, using soft delimiters (tabs and spaces) can sometimes cause unexpected results. Entering "Rotate Three 45 -20 10" will not assign the three values shown the first, second and third positional order of the Rotate Three command. The reason is that 45 -20 is a valid calculator expression and is therefore processed first. The end result is read as "Rotate Three 25 10". Not quite what you might have wanted. I recommend you eliminate the possibility of this happening by always entering strings of numbers separated by commas.

Quoted Strings

Sometimes it is necessary to type spaces, tabs or commas on the command line which are not to be read as delimiters. This is accomplished by placing the character string in quotes. Everything enclosed in the quotes will be read as a literal string. No calculations will ever be performed on quoted strings. Variable substitution will be performed based on the type of quote characters used. The two types of quote characters are:

<u>Quote Character</u>	<u>Meaning</u>
``	Literal string, no variable substitution
\\	Literal string with variable substitution

The back slash character (\) is used when you want variable substitution to be performed inside of the quotes. The back quote character (`) is an absolute literal string and no substitution will be performed.

The most common use of a quoted string is entering the save name of a drawing or symbol file. In the following examples, assume that the current text height is one unit. "%TX.height" is the TurboCAD Current Status Word for the current text height.

Save Drawing=\Plan %TX.height\ would save your drawing as "Plan 1". The Current status word %TX.height is substituted by its value, which is 1.

Save Drawing=`Plan %TX.height` would save your drawing as "Plan %TX.height". No variable substitution is performed.

Save Drawing=Plan %TX.height would cause an error. The space following the word Plan is read as a delimiter, making %TX.height be read as the next option of the command. Since "1" is not a valid option of the Save command and is not in a valid positional order, an error is generated.

Entering Point Coordinates

Most TurboCAD commands require points in order to accomplish their function. For example, the Line command requires at least two points to define a single line segment. Likewise, the Move command requires at least two points, one to define what is to be moved, and another to define where it is being moved to. In addition to using the crosshairs to locate these points, they can be entered at any time by typing in their coordinates through the keyboard. In general, points are entered in one of four formats, namely Relative, Absolute, Polar, and Deflection. TurboCAD allows you to either enter points before they are needed or as they are needed. If entered before, they are kept in the TurboCAD "point stack". Anytime you are in a TurboCAD command that is expecting a point or points to be entered (i.e. the crosshairs are active), TurboCAD first looks to the point stack and uses any points which are there. If none are there, it waits for you to provide them, either by using the crosshairs or by typing them in. In this manual, commands that can accept coordinates on the command line will have an option named "coordinates."

A point is a set of coordinates (i.e. x.y.z) which, as a group, define the point. The definition of a point or group of points entered from the keyboard is started by typing in a left square bracket key ([) and is ended by typing in a right square bracket (]) followed by a <Return>. The right square bracket (]) before the <Return> is optional. If you are not in a command when the "[" is entered, all points keyed in will go to the "point stack." If you are in a command which is expecting points, the points you type in will immediately be used by that command. Several coordinate sets (points) may be entered on the same line if they are each enclosed in brackets or if they are separated by the vertical bar (|) character within one pair of brackets.

The first value entered after the "[" is the coordinate type code, which is an abbreviation as described below. This is followed by values for "x," "y," and "z" for Relative or Absolute types or by values for "distance" and "angle" for Polar or Deflection types. The last value is the "key," which corresponds to one of the "View/Locate" keys described in the **Locating Points** section.

If the coordinate type code is omitted, it is assumed to be Relative. All distance and angle values may be positive, negative, or zero. If a distance or angle value is omitted, it is assumed to be zero. However, some delimiters must be used to assure that the values entered are assigned to their proper location in the definition of the point. If the key is omitted, it is assumed to be a <spacebar> (i.e. no View/Locate function).

Spaces, commas or tabs can be used as delimiters of the values which define a coordinate. We highly recommend you use commas. Because calculations are performed before the coordinate is processed, entering [4 -3] is read as a relative move of one unit in the X axis, not 4 in the X and -3 in the Y.

Coordinate type codes can be entered in upper or lower case letters - they are not case sensitive. Coordinate numerical values are always processed in the current Work unit as defined by the NEW or SETUP commands unless a unit other than the current Work is specified. For example, a value of 3 is interpreted as 3 Inches if the current Work unit is Inches and is interpreted as 3 Feet if the current Work unit is Feet. However, a value of 3" (with the " mark) is interpreted as 3 Inches regardless of the Work unit, even if you are working in metric units. It is important to remember that TurboCAD allows you to enter any valid mathematical expression for numerical values. For example (1+SQRT(4)) or (1+SQRT(4))" are equivalent to 3 and 3" above.

The COORDINATE command allows you to make the point stack active or inactive, to put points in the point stack by using the crosshairs, and to clear the point stack. You can see a list of the points currently on the stack by entering "Status Coordinate" or "SS CD" in abbreviated form (these are options of the STATUS command).

TurboCAD uses two coordinate systems, Viewplane and World. The Viewplane system is the generally accepted Cartesian coordinate system, in which your screen is the x-y plane and positive "z" values are towards you in front of the screen. The World coordinate system is related to 3D objects and may or not be in the same orientation as the Viewplane coordinate system, depending on whether you have rotated it in order to look at 3D objects from a view other than Plan View (from the top). Although TurboCAD has commands which convert 2D objects into 3D "Wireframes," the only command which creates 3D objects is the Wireframe command. For this reason, just about all of the time you are working in 2D in the Viewplane coordinate system and the values for "z" are irrelevant.

Point Formats

The format for Relative and Absolute type coordinates is:

[typecode , x , y , z , key]

The acceptable type code abbreviations for Relative and Absolute are:

<u>Type</u>	<u>Type Code</u>
Relative (Viewplane)	none , R , RV , VR
Absolute (Viewplane)	A , AV , VA , V
Relative World	RW , WR
Absolute World	W , AW , WA

The format for Polar and Deflection type coordinates is:

[typecode , distance , angle , key]

The acceptable type code abbreviations for Polar and Deflection are:

<u>Type</u>	<u>Type Code</u>
Polar	P
Deflection	D

The format for Input type coordinates is (this special type is further defined below):

[I]

There are no type codes applicable to Input type coordinates.

Definition of Coordinate Types

Relative (Viewplane)

A point will be entered the specified distance from the last point located along the X, Y, and Z axes of the current view plane. "key" is the key on your keyboard that you would like the point located with. See the **Locating Points** section for an explanation of the functions performed by each key.

Absolute (Viewplane)

A point will be entered the specified distance from the drawing origin along the X, Y, and Z axes of the current view plane. The drawing origin is set with the Origin command. "key" is the key on your keyboard that you would like the point located with. See the **Locating Points** section for an explanation of the functions performed by each key.

Relative World

A point will be entered the specified distance from the last point located along the X, Y, and Z axes of the World coordinate system. "key" is the key on your keyboard that you would like the point located with. See the **Locating Points** section for an explanation of the functions performed by each key.

Absolute World

A point will be entered the specified distance from the drawing origin along the X, Y, and Z axes of the World coordinate system. The drawing origin is set with the Origin command. "key" is the key on your keyboard that you would like the point located with. See the **Locating Points** section for an explanation of the functions performed by each key.

Polar

A point will be entered the specified distance and angle from the last point located. The angle is measured from the "zero polar angle" as set by the Polar option of the Preferences command. The Viewplane coordinate system is always used for polar coordinates as it is strictly a 2-dimensional coordinate type. "key" is the key on your keyboard that you would like the point located with. See the **Locating Points** section for an explanation of the functions performed by

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each key. The Angle units (degrees or radians) are set by the Angle options of the Preferences command.

Deflection

A point will be entered the specified distance and angle from the last point located. The angle is measured from the current angle as set by the last two points located on your drawing. The Viewplane coordinate system is always used for deflection coordinates as it is strictly a 2-dimensional coordinate type. "key" is the key on your keyboard that you would like the point located with. See the **Locating Points** section for an explanation of the functions performed by each key. The Angle units (degrees or radians) are set by the Angle options of the Preferences command.

Input

This coordinate type will display the crosshairs and wait for a single point to be entered interactively. It is used when you are keying in a string of points but want to define one or more of them "on the fly" with the crosshairs. It is a very useful feature for use in TurboCAD Macros. Only the letter "I" need be entered, and only input type keys will be accepted as valid input (i.e.: V, M, O, not Z, U, F).

Examples

[a,4,5]

An absolute viewplane coordinate will be placed on the point stack that would locate a point a positive four units along the X axis (to the right) and a positive five units (up) from the current drawing origin. Because the key value was omitted, the <spacebar> was used.

[va,,(3+2) | 2 | ;]

Three viewplane coordinates will be placed on the point stack. They are: an absolute coordinate of zero in the X axis and five units in the Y axis; a relative coordinate of two units to the right of the previous point; and a carriage return. The <spacebar> key will be used for the first two points on the stack, if the Circle command was entered, a circle with a radius of two units would be drawn centered five units to the right of the drawing origin. The carriage return (;) on the stack would then exit the Circle command.

Rect Constr=Center [[1,1]

The Rectangle command will be executed with its Construction option set to Center. Two viewplane coordinates will be placed on the point stack. They are: relative 0,0,0 and relative 1,1,0. The two points will be read from the stack immediately upon entering the command thereby drawing a rectangle that is two units square centered on the last point located. The <spacebar> key will be used to locate the points. After all of the points on the stack have been used, the crosshairs will be displayed to allow the entry of more points in the Rectangle command.

Line, [wa,3,3,SQRT(9)] [v 23 65 z] [i] [;] [p,%tx,he,45]

The Line command will be executed. Five coordinates will be placed on the point stack for processing. The first coordinate will locate a point at an absolute three units in the X, Y, and Z axes of the World coordinate system. The second coordinate will zoom in on a point that is twenty-three units to the right and sixty-five units up from the drawing origin. The third coordinate will display the crosshairs and wait for the operator to locate a point. The fourth coordinate is a <Return> and will exit the Line command. Because there is no longer an active

graphics command to read from the point stack, the last polar coordinate will remain in the stack until the next graphics command is executed. The polar coordinate has a distance equal to the current text height and an angle of forty-five degrees from the current polar angle.

Locating Points

View, Locate, and Construction Keys

One of the most powerful and flexible features of TurboCAD is the ability to pan and zoom, locate new points, snap to existing objects, turn grids on and off, and more, all while in any graphics command (i.e. the crosshairs are active).

Points are located by pressing a specific key on your keyboard or the button on your mouse. If your mouse has multiple buttons, the left one is it. Each key performs a specific viewing or snap-to function at the location of the crosshairs when the key is pressed. Because of the functions they perform, these keys are often called View/Locate Keys or Graphic Mode Keys. In summary, the name of each key and function it performs is:

<u>Key</u>	<u>Function</u>	<u>Key</u>	<u>Function</u>	<u>Key</u>	<u>Function</u>
space	any point	M	midpoint	enter	return
click	any point	N	normal/perp.	;	return
A	angle lock	O	origin	%	undo
B	boundary	P	pan	0 - 9	priority
C	closest	R	rounding	esc	special
D	display	S	set point	^	esc
E	end	T	tangent/para.	[coordinate
F	full view	U	unzoom	.	zoom window
G	grid	V	vertex	+	last obj. picked
H	opposite	W	where	-	last obj. created
I	intersection	X	horiz. lock		change X-hairs
J	junction	V	vert. lock	=	actual size
K	close	Z	zoom in	\no.	remem. window
L	line lock	Return	return	/no.	view window

Keys that perform viewing functions (i.e. pan, zoom ...) generally do not record a permanent point on the drawing, whereas keys that perform locate functions do. We describe these as Non-input type and Input type keys, respectively. Locating a point with an input type key will make that point the next point of whatever you are drawing or editing. It will also be remembered in the Current Status Words %CD.X and %CD.Y (and %CD.Z for 3D functions).

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Case is not important when entering View/Locate keys. A capital letter will perform the same function as a lower case letter. The exception is obviously keys that contain different shifted characters, like the number keys.

In the descriptions below, "search" means the area around the current crosshair location is searched for existing objects. The size of the area is set by the Searcharea option of the Preferences command.

spacebar (any point)

Locates a point or picks an object on the drawing. If you are locating a point, the point will be located at the current crosshair location. If you are picking an object, the Searcharea will be searched for an existing object on your drawing. If no object is found, an error is issued and no point is located. This is an input type key.

mouse click (any point)

This key performs the same function as the <spacebar> key.

A (angle lock)

Locates a point as close as possible to the crosshair location that is at some multiple of the Lock angle from the last point located. Using this key for the first point in a sequence makes no sense. The Lock angle is set by the Lock option of the Preferences command. This is an input type key.

B (boundary)

This key is an input type key, has many uses, and is very powerful. Its effect depends on the command you are currently executing.

When used for the first point of, or start of, a line, TurboCAD searches the area at the present crosshair location for an existing object and automatically forces a perpendicular (or normal) line starting at the object found. If it finds a line, the first segment of the new line will be perpendicular to the line found. If it finds a circle or arc, the first segment of the new line will be normal to the circle or arc.

When used as the first point in the Dimension command, which requires two or three points to get started (i.e. first point, direction, second point), and if a line segment is found beneath the crosshairs, it will automatically assume that the underlying segment is what you want to dimension and take its start point, direction, and end point as the points to be used for dimensioning.

For commands that have a Select option, such as the Delete command, and when this option is set to require that you define a "box" around the items to be affected (i.e. select is set to any value other than Pick), this key allows you to also select objects by picking them. For example, entering "Delete Select=Inside" requires that you locate the diagonally opposite corners of a box to delete the object inside. Some objects may prove easier to pick than to box. Pressing the key allows you pick these objects while in the select-by-box mode.

In the Area, Crosshatch and Boundary commands, this key searches for an existing object and uses it as part of the perimeter being defined.

The key sometimes works in conjunction with <Esc> defined below.

C (closest)

Searches for an object and locates a point on the object as close as possible to the location of the crosshairs. If no object is found, an error is issued and no point is located. This is an input type key.

D (display)

Redisplays the current window. This is useful for correcting pixel-blanking or a cluttered screen from extensive editing. Crosshair location is irrelevant when using this key. This is a non-input type key. On Tektronix keyboards the "Page" key will perform the same function.

E (end)

Ends the current object being drawn or function being performed and purges the Undo buffer. This is the same as exiting the current command and re-executing it with all options set to the same values. Crosshair location is irrelevant when using this key. This is a non-input type key.

F (full view)

Displays your entire drawing such that it fills the screen. Crosshair location is irrelevant when using this key. This is a non-input type key.

G (grid)

Toggles the display of the reference grids on and off. Crosshair location is irrelevant when using this key. This is a non-input type key.

H (opposite)

Searches for an object and locates a point at the opposite (farthest) endpoint of the segment or arc found. This key is the complement of the <V> key defined below. If no object is found, an error is issued and no point is located. This is an input type key.

I (intersection)

Searches for an object and locates a point at the intersection of the object found and the line defined by the last two points you entered (input type keys) prior to the "I." The resulting point may or may not be on the object found. For example, the intersection of a given line segment and a "target" line segment is not necessarily on the "target" but is somewhere on the theoretical extension of the "target". If no object is found, an error is issued and no point is located. This is an input type key.

J (junction)

Searches for two objects and locates a point where they cross. Both objects must be in the Searcharea to be found. If two objects are not found, an error is issued and no point is located. This is an input type key.

K (close)

In the Line, Leaderline, Doubleline, Fillet, Wall and Spline commands, this key closes the object currently being drawn by locating a point at its origin (starting point). Crosshair location is irrelevant when using this key. This is an input type key.

In the Boundary, Crosshatch and Area commands, this key will close the perimeter currently being defined by returning to its starting point. The enclosed area will not be immediately filled, crosshatched or calculated so that additional areas can be defined. It will be filled, crosshatched or calculated when a "Return" is entered or the command is otherwise terminated.

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In all other commands, the function of this key is undefined. For right now it locates a point at the plot origin (lower left corner) of your drawing.

L (line lock)

Locates a point at the intersection of the last angle and the vertical or horizontal crosshair, whichever is intersected first. The last angle is formed by the last two points located on your drawing (input type keys), not counting the <L> key you are pressing. In other words, draw an imaginary line through the last two points located on your drawing until it intersects the closest crosshair and this is where the point will be located. This is similar to the <I> key, except that the "target" line is one of the crosshairs. This is an input type key.

M (midpoint)

Searches for an object and locates a point at the midpoint of (half-way along) the segment or arc found. If no object is found, an error is issued and no point is located. This is an input type key.

N (normal/perpendicular)

Searches for an object and locates a point on the object (or an imaginary extension of the object) that is perpendicular from the last point. If no object is found, an error is issued and no point is located. This is an input type key.

O (origin)

Searches for an object and locates a point at the object's origin. If the object found is a group, the point is located at the group origin. Otherwise, the point is located at the primitive origin. Circles, Arcs, Ellipses and Rectangles have their origins at their center point. All other primitives have their origin at the first point located to create the primitive. If no object is found, an error is issued and no point is located. This is an input type key.

P (pan)

Moves the display window so that it is centered on the crosshairs. It takes the location at which you hit the <P> key and makes it the center of the new window. The display window does not change in size, only in location. This is a non-input type key.

R (rounding)

Toggles the rounding (crosshair jumping) on and off. Crosshair location is irrelevant when using this key. This is a non-input type key.

S (set point)

Pressing this key means you want the coordinates of the next point located remembered but not entered. After pressing the <S> key, the point defined by the next input type key will essentially become a "construction" point. This is most useful for defining a point which is a known relative (x & y) or polar (distance & angle) from an existing point on the drawing. An example is when you want to start (or end) a line, say, two units to the left of and three units down from the endpoint of an existing line. The <S> key is non-input type. However, the key pressed after the <S> key will be input type. Crosshair location is irrelevant when hitting the <S> key, but very relevant when you hit the next input type key.

T (tangent/parallel)

Searches for an object and locates a point that is tangent or parallel from the last point. After defining the starting point of a line (i.e. "rubberbanding" from it), locating the crosshairs anywhere on top of an existing line and hitting the <T> key will result in a line segment

parallel to the "target" line. If the "target" is a circle or arc, it will result in a segment tangent to the circle or arc. If no object is found, an error is issued and no point is located. This is an input type key.

U (unzoom)

Makes the display window larger (zooms out) centered on the crosshair location. The percentage of change in the display window size is set by the Zoom option of the Preferences command. This key is the complement of the <Z> key. This is a non-input type key.

V (vertex)

Searches for an object and locates a point at the nearest endpoint (vertex) of the segment or arc found. The crosshair may be placed anywhere along the visible length of the "target" object. If no object is found, an error is issued and no point is located. This is an input type key.

W (where)

Toggles the dynamic coordinate readout of the crosshair location between Absolute, Relative, Polar, Plot units and Off, in that order. The coordinate readout is displayed on the right side of the Prompt Line when active. Crosshair location is irrelevant when using this key. This is a non-input type key.

X (horizontal lock)

Locates a point as close a possible to the crosshair location that is horizontal (along the X axis) from last point. The vertical (Y axis) distance from the last point will be zero. This is an input type key.

Y (vertical lock)

Locates a point as close a possible to the crosshair location that is vertical (along the Y axis) from last point. The horizontal (X axis) distance from the last point will be zero. This is an input type key.

Z (zoom in)

Makes the display window smaller (zooms in) centered on the crosshair location. The percentage of change in the display window size is set by the Zoom option of the Preferences command. This is the complement of the <U> key. This is a non-input type key.

return (return)

Terminates the current graphics command, exits the graphic modes and returns you to the command mode to enter the next command. The Undo buffer is cleared for use by the next command. Some commands require more than one return to exit, like Text. Crosshair location is irrelevant when using this key. This is a non-input type key.

The 'line feed' key on some keyboards will perform the same function.

enter (return)

This key performs the same function as the <Return> key.

;(return)

This key performs the same function as the <Return> key.

% (undo)

Removes the last point located. The last object drawn or edit performed based on this point is undone. The <%> key can be pressed repeatedly to undo all points that have been located in the current command. You can undo back to the beginning of the command or the last <E>

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key. The exact undo function performed in each command is described in that command's documentation. Crosshair location is irrelevant when using this key. This is a non-input type key.

On Macintosh computers, the <Delete> and <clear> keys can also be used. On most other system keyboards, the , <delete>, <undo>, <cancel> or <backspace> key will perform the undo function.

0 thru 9 (priority)

Searches for an object and changes its "pick" priority. TurboCAD allows you to assign a priority between 0 and 255 to any object. The TurboCAD "default" priority is 125, unless you change it via the Priority command. If two objects overlap, the one with the highest priority will be the one found if you place the crosshairs where they overlap and try to locate a point on the object. For purposes of simplicity, we have "mapped" the 256 priorities to the numbers 0 through 9. The <0> (zero) key assigns priority zero to the object, <1> assigns priority 25, and so on in increments of 25. For example, assume that there are two horizontal lines on your drawing of different lengths which overlap for some distance and you want to "snap" to one of the "hidden" vertices. If these lines have the same priority (i.e. 125), which "hidden" vertex will you snap to when using the "V" key? Simply place the crosshairs on a non overlapping portion of one of the lines and hit a number greater than 5. This will assign a priority to that line greater than 125. Now you will snap to the vertex of the line with the higher priority when using the "V" key on the overlapping portion. You may change the priorities of several objects, but a number must be entered before picking each object. See the Priority command for more information on priorities. This is a non-input type key.

esc (escape)

This is our special function key. Its function differs based on the command you are using it in. A brief description of its functions is shown here. See the command documentation for more detailed information on what function the escape key performs. Crosshair location is irrelevant when using this key. This is a non-input type key.

Command	Function
Arc	Draws complement of last arc.
Area	Toggles between adding and subtracting areas.
Distance	Toggles between adding and subtracting distances.
Doubleline	Makes next segment invisible (gap).
Command	Function (Continued)
Change	Applies last change to all members of a group.
Fillet	Changes the fillet/chamfer size.
Gap	Gaps other side of a Doubleline.
Get	Rotates a retrieved symbol 90 degrees.
Insert	Mirrors last inserted symbol.
Line	Makes next segment invisible (gap). Toggles between the normal and the two tangents if a line was started with the "B" key on a circle or arc.

Rotate	Moves last rotated object.
Scale	Moves last scaled object.
Spline	Starts a new spline.
Transform	Moves last transformed object.
Wall	Cleanup intersection at last point.
Wireframe	Changes current Z depth.

^ (shift-6)

This key performs the same function as the <Esc> key.

[(coordinate entry)

Allows points to be entered by typing in coordinates through the keyboard. See the **Entering Coordinates** section below. Crosshair location is irrelevant when using this key.

. (zoom window)

The < . > (period) key will define the first of two diagonally opposite corners of a rectangular area you want to be the new display window. Any input type locate key can be used to locate the second point. See the **Entering Coordinates** section. This a non-input type key.

+ (last object picked)

Locates a point at the origin of the last object picked. When used in a create command (like Line) you will be drawing to the origin of the last picked object. When used in an edit command (like Move) the last picked object will be picked at its origin. This key is handy when several commands are used to perform edits on the same object. It eliminates having to accurately place the crosshairs and will find the object faster than a regular search. Crosshair location is irrelevant when using this key. This is an input type key.

- (last object in created on drawing)

Locates a point at the origin of the last object created in the drawing. When used in a create command (like Line) you will be drawing to the origin of the last object in the drawing. When used in an edit command (like Move) the last object in the drawing will be picked at its origin. This key is handy when several commands are used to perform edits on the same object. It eliminates having to accurately place the crosshairs and will find the object faster than a regular search. Crosshair location is irrelevant when using this key. This is an input type key.

(change crosshairs)

Toggles between six different types of crosshairs. They are the default full screen crosshairs, a plus, an X, large isometric, small isometric and none, in that order. See the Preferences command for more information on the Cursor types. Crosshair location is irrelevant when using this key. This is a non-input type key.

= (actual size)

Changes the size of the display window such that the drawing is being displayed as close as possible to the size it will plot. The display window will also be moved so that it is centered on the crosshairs. This is a non-input type key.

\ number (remember window)

Remembers the current display window with a number from 1 through 9. Enter the backslash key (\) followed by a number key (1 thru 9). Remembered windows are recalled by using the

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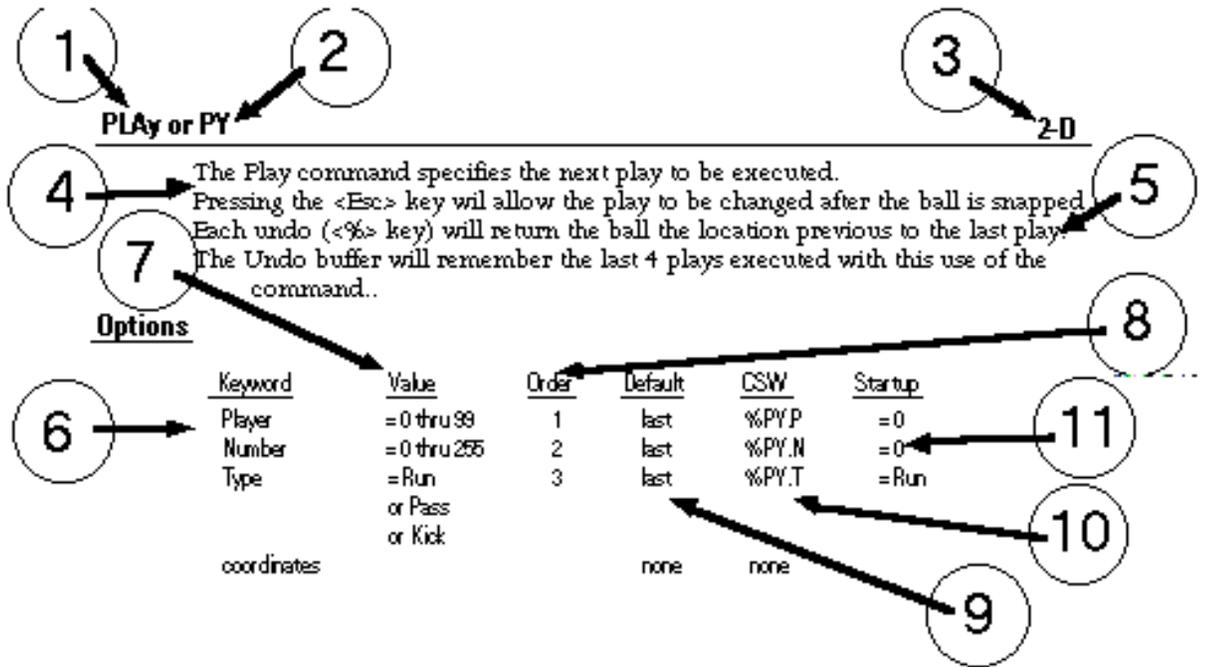
</> key. See the Window command for more information. Crosshair location is irrelevant when using this key. This is a non-input type key.

/ number (view window)

Displays a remembered widow. Enter the backslash key (/) followed by a number key (0 thru 9). Window number zero (0) will always recall the previous window. Windows are remembered by using the <\> key. See the Window command for more information. Crosshair location is irrelevant when using this key. This is a non-input type key.

CHAPTER 3 - HOW COMMANDS ARE DESCRIBED

This section describes the format that is used to describe the TurboCAD commands and their options in the Commands chapter. Below is a "sample" command's documentation. The labeled items on the documentation correspond to the descriptions below.



Defintion of Options

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Player=

This option selects the player that is to carry or receive the ball. A valid Player value is the number of any player currently on the field.

Warning: The numbers of defensive players are also valid, and if entered, will result in an interception, so be careful.

Number=

This option selects the play that is to executed. Enter the number of any play from the play book.

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Type=

Specifies the type of play to be executed. Valid play Types are Run, Pass and Kick.

Run means the ball will be run with by the specified player.

Pass means the ball will pass in the air to the specified player, who is to receive the ball.

Kick means the ball will be punted. This option is usually used o the last down.

Examples

Play Player=45 Number=22 Type=Run

py.,5,p

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Player number 45 will run with the ball while executing play number 22.

Pass play number five will be executed. The receiver will default to the last ball carrier.

Item 1 - Command Name.

This is the name of the command that is being described. The whole command name may be entered, or any abbreviation with the minimum abbreviation being the portion shown in capital letters. The two or three letter command code can also be entered to execute the command. The command name or code must be separated from any options by a comma or a space.

Item 2 - Command Code.

All TurboCAD commands have a unique two or three character code that may be entered instead of the command name or abbreviation. The command code must be separated from any options by a comma or a space.

Item 3 - Command Type.

Tells you if the command is capable of creating or editing 2-dimensional or 3-dimensional objects, or both. Commands are designed to be used only in a Macro are labeled as such.

Item 4 - Description.

Explains what the command does and how to use it.

Item 5 - Undo.

Explains how the Undo feature works in this command. Any other keys that have a special function will also be described in this area.

Item 6 - Option Name.

Listed in this column are the names of the options of this command. We call the name of an option its keyword. Keywords can be abbreviated like command names down to the portion shown in capitals. Some options consist of both a keyword and a value separated with an equals sign. When entering keyword-equals-value options, the options can be entered in any order. Some options have only a keyword. Keyword-only options are only active when they are entered.

Item 7 - Option Value.

This is the value that an option can be set to. The value is sometimes a number or a word. If it is a number, the range of possible numbers is generally shown. Example: `Class = 0 thru 255`. If the value is a word, the acceptable words are listed. If the value is shown in italics, the word in italics is to be replaced with a value. The word in italics is merely describing the type of value you can type in. Example: `Style = style`. This is done to save space in the manual and avoid repetition, because some options and their values are used by many commands. These common options are described in the **Common Options** section of the manual.

Item 8 - Positional Order.

Option values have a fixed positional order in which they must be entered if you do not wish to enter the option keyword. In the example: `"Play,5,6,R"` the values 5, 6 and R will be assigned to the Player, Number and Type options, respectively.

Item 9 - Default Value.

This column lists the default value that will be used if the option is not entered. The word "last" means the last used value for that option will be used again. If it hasn't been used, it will default to the Startup Value described below. The word "none" means the option will not be used unless entered (i.e. keyword-only options).

Item 10 - Current Status Word (CSW).

This column shows the name of the Current Status Word that remembers the last used value of an option. A CSW is like a variable whose value is automatically assigned by the program. See the Current Status Words section for more information.

Item 11 - Startup Value.

This column shows the value that an option is set to when the TurboCAD program is first run.

Item 12 - Notes, Tips and Warnings.

Special notes, warnings or tips to the operator are shown like this throughout the documentation.

Item 13 - Option Description.

Each option is described in detail in the "Definition of Options" part. The description explains what the option does and how to use it. The valid values for the option will also be listed and described here.

HOW COMMANDS ARE DESCRIBED

Item 14 - Examples.

Examples of the command as it would be entered onto the command line are shown at the end of the documentation for each command. A description of the example accompanies it.

CHAPTER 4 - COMMON OPTIONS

There are many options that are common to several TurboCAD commands. To save space and avoid repetition in the manual, these common options are explained here. Each command that uses one of these options will refer you to this section if you require more information. The common options defined below are:

SELECT = LAYER =
OBJECT = PEN =
GLOBAL = STYLE =
ANGLE = PRIORITY =
SYSTEM = WORKSET =
CLASS = MIMIC

Select =

This option determines how objects are to be selected in commands which have a "Select=" option. . If this option is not used or no value is entered for it, the object will be selected per the current Select option on the Status palette. Valid Select values are:

Pick	Objects are picked with the crosshairs one at a time
Inside	Objects completely inside a rectangular area
Outside	Objects completely outside a rectangular area
Touching	Objects touching a rectangular area
Notinside	Objects touching or outside a rectangular area
NOTOutside	Objects touching or inside a rectangular area
NOTTouching	Objects completely inside or outside a rectangular area
All	All objects on your drawing

Pick means that objects are "picked" one at a time by locating a point on or near the object. The TurboCAD "default" value is always set to Pick. All means that all objects on the drawing will be selected. The remaining values require that two points be located to define the diagonally opposite corners of a box.

When selecting objects by defining the diagonally opposite corners of a box (the Inside, Outside, Touching, Not inside, NotOutside or Not Touching values), objects can also be picked by pressing the key (or <Shift>-clicking) while the crosshairs are on any part of the object even if the object is not completely contained in the box. Objects picked with the key under these circumstances will be included in the selection regardless of where they are located. This combination of both picking and boxing objects to select them can be handy when working in congested areas of a drawing.

COMMON OPTIONS

Object =

This option determines the type of objects that can be affected by this command. Valid Object values are one of the following names or two letter codes:

Any	or AN	Any and all objects
ARcs	or AR	Only ungrouped Arc or Circle primitives
Boundaries	or BD	Only ungrouped Boundary primitives
Circles	or CC	Only ungrouped Circle or Arc primitives
Copies	or CP	Only copied groups
Dimensions	or DM	Only ungrouped Dimension primitives
Doublelines	or DB	Only ungrouped Doubleline primitives
Ellipses	or EL	Only ungrouped Ellipse primitives
Fillets	or FL	Only ungrouped Fillet primitives
Groups	or GP	Only unique & copied groups
Leaderlines	or LL	Only ungrouped Leaderline primitives
Lines	or LI	Only ungrouped Line primitives
POints	or PT	Only ungrouped Point primitives
POLygons	or PG	Only ungrouped Polygon primitives
Primitives	or PR	Only graphic primitives (not groups)
Rectangles	or RE	Only ungrouped Rectangle primitives
Splines	or SP	Only ungrouped Spline primitives
Text	or TX	Only ungrouped Text primitives
Wireframes	or WF	Only ungrouped Wireframe primitives

Global =

This option determines how global changes to copied groups will be handled. A “global change” means if you are altering the structure of a group that has multiple copies on your drawing, one, some or all of the other copies should be altered in the same manner. This option simply gives you the ability to pre-answer the global question if you know you will be altering copied groups on your drawing. The default value for this option is the current setting of the Global option of the Preferences command (%PF.G). %PF.G is set to "Ask" on startup. Entering a Global value here will override the Global setting in the Preferences command. Valid Global values are:

Ask All Some One Cancel

Ask means that if a global change is possible, you will be prompted to answer how it should be handled. Valid answers to the global question would be any of the other values; namely, All, Some, One, Cancel.

All means that all copies of the altered group will be changed automatically.

Some means that some copies of the altered group are to be changed. All of the copies found on your drawing will be highlighted and you will be prompted to pick the ones you want changed. To execute the change, press the <E> key or <Return> key after you have picked all the desired copies. The altered copies will no longer be considered identical copies to the non-altered copies of the group.

One means that only the picked group is to be altered. The picked group will no longer be considered a copy, it is now a unique group.

Cancel means you wish to cancel your selection of a group. No change will be made to the picked group or any of its copies. This value is provided mainly as an answer to the global question when it is asked. Entering this value directly onto the command line will automatically cancel the selection of any objects you pick making the command useless.

Angle =

This option sets an angle. A valid Angle value is any positive or negative whole or fractional number within the range of -360 thru 360 if the angle units are degrees and -2^1 thru 2^1 if they are radians. A positive value means counterclockwise rotation and a negative value means clockwise rotation. Zero angle is along the X axis (horizontal) pointing to the right. Angle values can also be entered in a degrees- minutes-seconds format. See the Calculator command for more information.

Some commands allow the word `Points` to be entered instead of an angle value allowing you to locate a point, or points, to define the angle.

System =

Sets the type of coordinate system that will be used. Valid System values are Viewplane and World.

Viewplane means the X, Y and Z axes are relative to your physical screen and is always the system used for 2D commands. The X axis runs horizontally across your screen with positive values being to the right. The Y axis runs vertically on your screen with positive values being up. The Z axis is perpendicular to your screen with positive values being towards you.

World means the X, Y and Z axes are relative to the 3-dimensional data in your drawing. As the 3-dimensional world is rotated the World coordinate system is rotated with it. In a plan view the World and Viewplane coordinate system are aligned in all axes.

Class =

A valid value for this option is always a number between 0 and 255.

If this is an option of a command which creates an object (i.e. Line, Circle, Dimension, etc.), the value entered will set the class in which the object is created and will temporarily override the class value which appears on the Status palette. If this option is not used or no value is entered for it, the object will be created in the class value which appears on the Status palette.

If this is an option of a command which changes (or edits) an object (i.e. Change, Change Group, etc.), the value entered will set the class to which the object will be changed regardless

COMMON OPTIONS

of the class value which appears on the Status palette. If this option is not used or no value is entered for it, the the class value for the object will not be changed.

The class value which appears on the Status palette is set on the palette itself, by the "Class=" option of the Work command or by the "Work=" option of the Class command.

Note: This option should not be confused with the "Classes=" option which appears in the Count, Display, List and Save commands. This option has different valid values (i.e. ranges, etc.) and is explained further in each of these commands.

Layer =

A valid value for this option is always a number between 0 and 255.

If this is an option of a command which creates an object (i.e. Line, Circle, Dimension, etc.), the value entered will set the layer in which the object is created and will temporarily override the layer value which appears on the Status palette. If this option is not used or no value is entered for it, the object will be created in the layer value which appears on the Status palette.

If this is an option of a command which changes (or edits) an object (i.e. Change, Change Member, etc.), the value entered will set the layer to which the object will be changed regardless of the layer value which appears on the Status palette. If this option is not used or no value is entered for it, the the layer value for the object will not be changed.

The layer value which appears on the Status Palette is set on the palette itself, by the "Layer=" option of the Work command or by the "Work=" option of the Layer command.

Note: This option should not be confused with the "Layers=" option which appears in the Display command. This option has different valid values (i.e. ranges, etc.) and is explained further in this command.

Pen =

A valid value for this option is always a number between 0 and 255.

If this is an option of a command which creates an object (i.e. Line, Circle, Dimension, etc.), the value entered will set the pen in which the object is created and will temporarily override the pen value which appears on the Status palette. If this option is not used or no value is entered for it, the object will be created in the pen value which appears on the Status Palette.

If this is an option of a command which changes (or edits) an object (i.e. Change, Change Member, etc.), the value entered will set the pen to which the object will be changed regardless of the pen value which appears on the Status palette. If this option is not used or no value is entered for it, the the pen value for the object will not be changed.

The pen value which appears on the Status palette is set on the palette itself, by the "Pen=" option of the Work command or by the "Work=" option of the Pen command.

The Pen value of an object determines in which color it will be displayed as determined by the Colors command and what line weight (width) it will have as determined by the Width command.

COMMON OPTIONS

Style =

A valid value for this option is any one of the following:

Solid	or	SO	or	0
Dotted	or	DO	or	1
Centerline	or	CL	or	2
Dashed	or	DA	or	3
Longdash	or	LD	or	4
Phantom	or	PH	or	5
LONGCenterline	or	LC	or	6
LONGGap	or	LG	or	7

If this is an option of a command which creates an object (i.e. Line, Circle, Dimension, etc.), the value entered will set the line style in which the object is created and will temporarily override the style value which appears on the Status palette. If this option is not used or no value is entered for it, the object will be created in the style which appears on the Status palette.

If this is an option of a command which changes (or edits) an object (i.e. Change, Change Member, etc.), the value entered will set the style to which the object will be changed regardless of the style value which appears on the Status palette. If this option is not used or no value is entered for it, the the style value for the object will not be changed.

The style value which appears on the Status palette is set on the palette itself, by the "Style=" option of the Work command or by the "Work=" option of the Style command. See the Style command for further information.

Priority =

A valid value for this option is always a number between 0 and 255.

If this is an option of a command which creates an object (i.e. Line, Circle Dimension, etc.), the value entered will set the priority in which the object is created and will temporarily override the current priority value as set by the Work command. If this option is not used or no value is entered for it, the object will be created in the priority value last set by the Work command.

If this is an option of a command which changes (or edits) an object (i.e. Change, Change Member, etc.), the value entered will set the priority to which the object will be changed regardless of the current priority value as set by the Work command. If this option is not used or no value is entered for it, the the priority value for the object will not be changed.

The priority of an object may also be changed "on the fly" by entering an integer between 0 and 9 with the crosshairs on the object. See the **Locating Points** section.

The current priority value is set by the "Priority=" option of the Work command or by the "Work=" option of the Priority command.

COMMON OPTIONS

Workset =

A valid value for this option is the name of any previously defined Workset which the user has defined in the via the "Workset=" option in the Work command. A Workset is a combination of Class, Layer, Pen, Style, and Priority (any one or more of them) to which the user has assigned a name.

If this is an option of a command which creates an object (i.e. Line, Circle, Dimension, etc.), the Class, Layer, Pen, Style, and Priority values used will be those of the Workset (with that name) as it was defined and will temporarily override those displayed on the Status Line. If the Workset definition did not include a value for one or more of these, then the values displayed on the Status palette will be used for those unspecified values. Furthermore, entering a valid Workset name for this option will also override any values entered for Class, Layer, Pen, Style and Priority entered in the command unless that value was not specified in the Workset definition. If this option is not used and no individual values are entered for Class, Layer, Pen, Style and Priority in the command, the values used will be those displayed on the Status line.

If this is an option of a command which changes (or edits) an object (i.e. Change, Change Member, etc.), the values to which the object will be changed are those of the Workset definition regardless of the values displayed on the Status palette. Class, Layer, Pen, Style and Priority values not specified in the Workset definition will be unaffected.

Mimic

This is a very powerful "Keyword Only" option as described in Entering Commands earlier in this section. That is, it is active only if the keyword ("Mimic" in this case) is entered among the options for the command.

This option is common to all commands which create objects (i.e. Line, Circle, Dimension,). The purpose of this option is to allow you to duplicate most of the characteristics (i.e. Class, Layer, Pen, Style, ...) of existing objects on a drawing and have them automatically assumed by the object which you are creating. When you use this option, TurboCAD will first ask you to pick an existing object of the type you are creating, then you will go through the steps necessary to create the new object. The new object will assume the characteristics of the one picked regardless of the values displayed on the Status Line. If you pick an object which is not of the type you want to create, an error message will be issued.

For example, executing the command "Circle Style=dashed Mimic" will cause TurboCAD to ask you to pick an existing Circle or Arc on the drawing and then will return to the mode of actually creating the circle. The circle you create will have all of the characteristics of the one you picked except for its style, which will be dashed. Options not specifically set will default to the TurboCAD "default" values.

In some commands, such as Text or Dimension, which have numerous options, this option can save a lot of time. In these cases, the new text or dimension created will have all of the specific characteristics (i.e. Offset, Height, Extension, Leader, etc.) of the object picked as well as the environmental characteristics (Class, Layer, Pen, ...) of the object. This eliminates the need for you to "look up" and re enter the characteristics of existing objects just because you want to create new ones with the same characteristics.

CHAPTER 5 - COMMAND TYPES AND LISTS

Types of Commands

The TurboCAD command language is a collection of over 140 commands, each with a variety of Options. TurboCAD commands can generally be divided into groups of commands which are used to:

1. Create objects
2. Edit or modify or change or copy objects
3. Inquire about or query existing objects
4. Setup the drawing environment
5. Accomplish disk functions such as retrieve or save
6. Control the execution of, or are specific to, TurboCAD Macros

Command List - Alphabetical

(AS)	ALIAS	(DG)	DIGITIZER	(MG)	MERGE	(RTT)	ROTATE THREE
(AL)	ALIGN	(DM)	DIMENSION	(MS)	MESSAGE	(RTW)	ROTATE WORLD
(AN)	ANGLE	(DP)	DISPLAY	(MR)	MIRROR	(RD)	ROUNDING
(AR)	ARC	(DS)	DISTANCE	(MT)	MODIFYTEXT	(SV)	SAVE
(AA)	AREA	(DB)	DOUBLELINE	(MV)	MOVE	(SC)	SCALE
(BL)	BELL	(EL)	ELLIPSE	(MVA)	MOVE ARC	(SCT)	SCALE THREE
(BD)	BOUNDARY	(ER)	ERROR	(MVE)	MOVE EDGE	(SL)	SELECT
(BK)	BREAK	(ET)	EXTEND	(MVM)	MOVE MEMBER	(SU)	SETUP
(CA)	CALCULATOR	(EX)	EXTRUDE	(MVN)	MOVE NODE	(SK)	SKETCH
(CF)	CHAMFER	(FD)	FIELD	(MVP)	MOVE PARALLEL	(SP)	SPLINE
(CG)	CHANGE	(FF)	FILE	(MVS)	MOVE SEGMENT	(SS)	STATUS
(CGG))	CHANGE GROUP	(FL)	FILLET	(MVV)	MOVE VERTEX	(SO)	STOP
(CGM)	CHANGE MEMBER	(FLE)	FILLET EXISTING	(MVW)	MOVE WORLD	(SR)	STRETCH
(CGO)	CHANGE OBJECT	(FT)	FONT	(NW)	NEW	(ST)	STYLE
(CC)	CIRCLE	(GA)	GAP	(OP)	OPEN	(SB)	SUBSTITUTE
(CL)	CLASS	(GAA)	GAP ARC	(OR)	ORIGIN	(SF)	SURFACE
(CZ)	CLEAR	(GAE)	GAP EDGE	(OV)	OVERLAY	(SN)	SYNC
(CX)	CLIP	(GE)	GET	(PA)	PATH	(TG)	TAG
(CZ)	CLOSE	(GO)	GOTO	(PS)	PAUSE	(TX)	TEXT
(CS)	COLORS	(GD)	GRID	(PN)	PEN	(TI)	TIMER
(CD)	COORDINATE	(GP)	GROUP	(PT)	POINT	(TF)	TRANSFORM
(CP)	COPY	(HE)	HELP	(PG)	POLYGON	(TFT)	TRANSFORM THREE
(CPM)	COPY MEMBER	(IF)	IF	(PF)	PREFERENCES	(UG)	UNGROUP
(CPP)	COPY PARALLEL	(IN)	INSERT	(PR)	PRIORITY	(VW)	VIEW
(CPR)	COPY RADIAL	(INN)	INSERT NODE	(PJ)	PROJECT	(WA)	WALL
(CT)	COUNT	(INV)	INSERT VERTEX	(PM)	PROMPT	(WT)	WEIGHT
(CR)	CROSSHATCH	(LA)	LAYER	(PU)	PUT	(WH)	WHERE
(DL)	DELETE	(LL)	LEADERLINE	(QT)	QUIT	(WD)	WINDOW
(DLE)	DELETE EDGE	(LI)	LINE	(RL)	READLINE	(WF)	WIREFRAME
(DLG)	DELETE GAP	(LS)	LIST	(RC)	RECORD	(WK)	WORK
(DL3)	DELETE GAP3	(LD)	LOAD	(RE)	RECTANGLE	(WP)	WORKPLANE
(DLM)	DELETE MEMBER	(LG)	LOG	(RP)	REPLACE	(WL)	WRITELINE
(DLN)	DELETE NODE	(LU)	LOOKUP	(RN)	RETURN	(ZD)	ZDEPTH
(DLS)	DELETE SEGMENT	(MA)	MACRO	(RV)	REVOLVE	(-)	-
(DLV)	DELETE VERTEX	(MU)	MENU	(RT)	ROTATE		

Command List - Functional

The following is a functional list of all of the TurboCAD commands:

CREATE		(DLM)	DELETE MEMBER	QUERY	(LG)	LOG	
(AR)	ARC	(DLN)	DELETE NODE	(AA)	AREA	(MU)	MENU
(BD)	BOUNDARY	(DLS)	DELETE	(AN)	ANGLE	(OR)	ORIGIN
(CF)	CHAMFER	(DLV)	DELETE VERTEX	(CA)	CALCULATOR	(PN)	PEN
(CC)	CIRCLE	(EX)	EXTRUDE	(CT)	COUNT	(PR)	PRIORITY
(CR)	CROSSHATCH	(ET)	EXTEND	(DS)	DISTANCE	(PF)	PREFERENCE
(DM)	DIMENSION	(FD)	FIELD	(ER)	ERROR	(PJ)	PROJECT
(DB)	DOUBLELINE	(FLE)	FILLET EXISTING	(HE)	HELP	(RD)	ROUNDING
(EL)	ELLIPSE	(GA)	GAP	(LS)	LIST	(RTW)	ROTATE
(FL)	FILLET	(GAA)	GAP ARC	(LU)	LOOKUP	(SU)	SETUP
(LI)	LINE	(GAE)	GAP EDGE	(SS)	STATUS	(ST)	STYLE
(LL)	LEADERLINE	(GP)	GROUP	(TI)	TIMER	(VV)	VIEW
(PT)	POINT	(IN)	INSERT	(WH)	WHERE	(WD)	WINDOW
(PG)	POLYGON	(INN)	INSERT NODE			(WT)	WEIGHT
(RE)	RECTANGLE	(INV)	INSERT VERTEX	INPUT/OUTPUT		(WK)	WORK
(SK)	SKETCH	(MV)	MOVE	(CZ)	CLEAR	(WP)	WORKPLANE
(SP)	SPLINE	(MVA)	MOVE ARC	(CZ)	CLOSE	(ZD)	ZDEPTH
(SF)	SURFACE	(MVE)	MOVE EDGE	(GE)	GET	(_)	
(TX)	TEXT	(MVM)	MOVE MEMBER	(LD)	LOAD		
(WA)	WALL	(MVN)	MOVE NODE	(MG)	MERGE	MACRO	
(WF)	WIREFRAME	(MVP)	MOVE PARALLEL	(NW)	NEW	(CD)	COORDINATE
		(MVS)	MOVE SEGMENT	(OP)	OPEN	(FF)	FILE
		(MVV)	MOVE VERTEX	(PA)	PATH	(GO)	GOTO
MODIFY		(MVW)	MOVE WORLD	(PU)	PUT	(IF)	IF
(AL)	ALIGN	(MR)	MIRROR	(QT)	QUIT	(MA)	MACRO
(BK)	BREAK	(MT)	MODIFYTEXT	(SV)	SAVE	(MS)	MESSAGE
(CG)	CHANGE	(OV)	OVERLAY	(SN)	SYNC	(PS)	PAUSE
(CGG)	CHANGE GROUP	(RP)	REPLACE			(PM)	PROMPT
(CGM)	CHANGE	(RV)	REVOLVE	ENVIRONMENT		(RC)	RECORD
(CGO)	CHANGE	(RT)	ROTATE	(AS)	ALIAS	(RL)	READLINE
(CP)	COPY	(RTT)	ROTATE THREE	(BL)	BELL	(RN)	RETURN
(CPM)	COPY MEMBER	(SC)	SCALE	(CL)	CLASS	(SL)	SELECT
(CPP)	COPY PARALLEL	(SCT)	SCALE THREE	(CS)	COLORS	(SO)	STOP
(CPR)	COPY RADIAL	(SR)	STRETCH	(DP)	DISPLAY	(WL)	WRITELINE
(CX)	CLIP	(SB)	SUBSTITUTE	(DG)	DIGITIZER		
(DL)	DELETE	(TG)	TAG	(FT)	FONT		
(DLE)	DELETE EDGE	(TF)	TRANSFORM	(GD)	GRID		
(DLG)	DELETE GAP	(TFT)	TRANSFORM	(LA)	LAYER		
(DL3)	DELETE GAP3	(UG)	UNGROUP				

CHAPTER 6 - COMMAND DESCRIPTIONS

ALIAs or AS

2D & 3D

The Alias command allows you create your own commands (of sorts). An alias is the name you assign to a character string of up to 60 characters. This string contains one or more TurboCAD commands which is (are) executed when you type in the alias name on the command line. The Alias command can also list and remove existing aliases.

Aliases differ from Macros in that Aliases are searched for automatically when you type a command entry (just like real TurboCAD commands), whereas Macros must be called for specifically by executing the Macro command. Aliases are also loaded into RAM which makes them execute much more quickly than Macros, which are read from disk. They are limited, however, to 60 characters, are only remembered during your current TurboCAD session, and have to be re-created if you need to modify them.

Because the Alias command allocates memory, aliases should not be used as a replacement for Macros in all cases. The Alias command allocates memory in chunks of 1K byte. Each 1Kbyte is enough memory to hold 10 aliases.

Aliases are searched for and executed before regular TurboCAD commands. This means that if you name an alias the same as a TurboCAD command you are limiting your access to that TurboCAD command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Function	= List or Add or Delete	1	List	%AS.F	=List
Name	= name	2	none	%AS.N	= ``
Statement	= string	3	none	%AS.S	= ``
Active	= switch	4	last	%AS.A	=Yes

Definition of Options

Function =

This option sets the function that you want the Alias command to perform. Valid values are: List, Add and Delete.

List will list the existing aliases. The Name and Statement options are not needed and will be ignored when the List function is specified.

A Commands

Add will add a new alias to those already existing. The Name and Statement options are required when this function is specified.

Warning: Adding an alias will automatically overwrite an existing alias of the same name with no questions asked (or prompts issued).

Delete will remove an alias from the list of existing aliases. Aliases are removed by name. Therefore, the Name option is required, but the Statement option is not.

Name =

This option defines the name of the alias to be added or deleted. Alias names are limited to 60 characters. Case is not important when defining and executing aliases. Example: BrickWall, brickwall and BRICKwall are all the same alias. Alias names cannot be abbreviated when executed like TurboCAD commands can - they must be entered exactly as defined. A name containing spaces or special characters must be quoted ('like this' or \like this) or the name may be interpreted as separate options of the Alias command.

Statement =

This option defines the character string that will be read and processed when an alias is executed. Alias strings are limited to 60 characters. The string can contain any TurboCAD graphics command, Macro or calculation. The string cannot contain another alias. Several TurboCAD commands or Macros can be entered into the string if they are separated with semicolons (;). A semicolon on the TurboCAD command line is read the same as a <return>. A string containing spaces, commas or tabs must be enclosed in single back quotes ('like this') or slashes (\like this) or the string will be interpreted as separate options of the Alias command.

Active =

This option makes the alias list active or inactive. The Active value must be a switch value. Switch values are: True or False, Yes or No, On or Off, All or None.

Examples

ALIAS

A list of existing alias names and their associated statements will be displayed in a window on your screen. Here's an example of what a list might look like:

<u>Name</u>	<u>Statement</u>
Wall8	Doubleline Width=8" Style=Solid Layer=45
BrickWall	CR, Brick,0,1/2, Pen=4
Jeff	/JeffStart {Jeff's startup Macro}
rct	Rectangle; Circle; Text

AS Function=Delete Name=BrickWall

The alias "BrickWall" will be removed from the list of existing aliases.

as a widget `re w=2 h=3 [i|];cc s=1 [r,0,0];`

The "widget" alias will be added to the existing list of aliases. When executed, this alias will execute the TurboCAD commands shown in quotes above. Quotes are necessary so the command processor does not interpret each word on the command line as a separate option of the Alias command.

Alias Active=Off

The current alias list will not be used. The alias list is retained for later reactivation.

Align or AL

2D

The Align command moves objects on your drawing so that they will be "aligned" to an alignment line which you define. The movement of each object will be along the perpendicular to the alignment line.

After entering the command, the desired options and a return, you will be prompted to locate the alignment point. If the "Angle=Points" option was selected (or defaulted to) you will be prompted to locate a second point to define the alignment angle. The alignment line is the line that subsequently picked objects will be aligned to. Next you will be prompted to pick objects or define a box to select the objects to be aligned. If you pick an object, the point picked on the object is the point that will be aligned to the alignment line. If objects are selected by box, their origins are aligned to the alignment line.

Each undo (<%> key) will return the last aligned object or box of objects to its original location. The Undo buffer will remember the last 128 objects aligned with this use of the command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Angle	= angle or Points or Horizontal or Vertical	1	Points	%ALA	=Points
Select	= select	2	Pick	%ALS	=Pick
Object	= object	3	Any	%AL.O	=Any
coordinates		none	none		

Definition of Options

Angle =

This option sets the angle of the alignment line. A valid Angle value is any number between -360 and 360 degrees, or -2¹ and 2¹ radians, or the words Points, Horizontal or Vertical. See the **Common Options** section for more information on angle values.

Points means you wish to locate a second point to define the angle of the alignment line.

Horizontal means the angle of the alignment line will be set to zero degrees. Only one point need be located to define the location of the alignment line on your drawing.

Vertical means the angle of the alignment line will be set to a positive ninety degrees. Only one point need be located to define the location of the alignment line on your drawing.

Select =

See **Common Options** for more information.

Object =

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

ALIGN

The command options will default to Angle=Points, Select=Pick and Object=Any. You will be prompted to locate two points to determine the alignment line and its angle and then to pick the objects you wish to have aligned. Each object will be moved perpendicular to the alignment line until the point picked on the object lands on the alignment line.

ALI 30°12' [R,0,0]

The angle of the alignment line will be set to a positive thirty degrees, twelve minutes. The Select and Object options will default to Pick and Any, respectively. The alignment line will originate at the last point located on your drawing (Relative coordinate of zero horizontal and zero vertical distance from the last point). You will be prompted to pick the objects you wish to have aligned. Each object will be moved perpendicular to the alignment line until the point picked on the object is on the alignment line.

align object=text select=inside angle=vertical

The angle of the alignment line will be set to ninety degrees (vertical). The Select option will be set to Inside, and the Object option will be set to Text. You will be prompted to locate a point to locate the alignment line. You will then be prompted define a box by locating its diagonally opposite corners. Only ungrouped Text primitives that are completely inside the box will be aligned. The origin of each text paragraph will be aligned with the alignment line.

AL,V,I,T

This command is exactly the same as the one above. The option values were entered in positional order, eliminating the need to enter the option keywords. Commas were used as delimiters instead of spaces.

ANgLe or AN**2D**

The Angle command calculates and displays the absolute angle of the imaginary line defined by two points, the absolute angle of an existing line segment, the included angle between three points, or the included angle between two existing line segments. The display shows the absolute angle of the first segment, the absolute angle of the second segment (if any), the included angle (if any), and the total included angle. The format and accuracy of the results will depend on the current "Positive Angle =", "Angle unit =", "Base Angle =", "Format =" and "Rounding =" values of the Preferences command.

After entering the command and a <Return>, you will be prompted to define the angle to be calculated by locating points and/or picking existing lines on your drawing, depending on the options you select for this command. You can use any of the appropriate Locate keys to locate points by snapping to existing objects in your drawing. To select an entire line segment as part of the angle calculation, hold the <Shift> key down while picking the segment with the mouse key. Pressing the (boundary) key will do the same thing. The segment can be part of a multi segment line within a group, or part of one which is not grouped with anything else on the drawing. Existing line segments do not have to intersect in order for the angle between them to be calculated correctly.

As the angle is being created, TurboCAD will show you which end points (vertices) are being used to define the angle by placing temporary point markers on the screen. Each time an angle is defined, the result will be displayed. Each angle is calculated independently. You can continue to define additional angles, which will either be added to or subtracted from the total, until you exit the command by entering a <Return>. If you wish to subtract one angle from the total, you must End it (<E> key), press the <Esc> key to toggle from adding to subtracting areas, and then define the next angle. This is also accomplished by pressing the appropriate button on the angle calculation dialog on computers which provide such dialogs.

Each Undo (<%> key) will remove the last point located or line segment picked for the current calculation. The calculation for the last angle can be removed from the total by selecting Undo immediately after clicking "OK" on the calculation dialog.

The values calculated by the Angle command are remembered in the following Current Status Words until the next use of the command:

Value	CSW
Angle of last segment	%AN.Angle

A Commands

Angle of previous segment	%AN.Previous
Included Angle	%AN.Included
Difference between last two angles	%AN.Difference
Total Angle	%AN.Total

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Type	= Segment or Included	1	Segment	%AN.T	=Segment
Silent			none	%AN.S	
coordinates			none	none	

Definition of Options

Type =

This option specifies whether you want to calculate the absolute angle of a segment or the included angle between two segments.

Segment means two points can be located or an existing segment picked (or <Shift> key) to define the segment for an absolute angle calculation.

Included means three points can be located or two existing segments can be picked to define the segments for calculation of the included angle between them.

Silent

If this keyword-only option is used, the results of the calculation will not be displayed. However, the Current Status Word values will still be set. This option is most commonly used when angle calculations are performed in a Macro.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

AN

You will be prompted to locate two points to define the segment whose absolute angle you wish to have calculated. This segment can also be defined by picking an existing segment with the or <Shift> key.

an, i, s

You will be prompted to locate three points to define the two segments whose included angle you wish to have calculated. These segments can also be defined by picking two existing segment with the or <Shift> key. The absolute angle of each segment will be displayed, along with the included angle between them. Since the "Silent" option was used, the results

will not be displayed on the screen, but the corresponding Current Status Words will be set for future use.

Arc or AR

2D

The Arc command creates circular arcs. Arcs can be defined by Radius, Diameter, Circumference, or Corner. The Size of the arc (radius or diameter) can be specified by locating points or typing in a known value.

After entering the command, the desired options and a return, you will be prompted to locate one, two or three points, depending on your choice of options.

Pressing the <Esc> immediately after drawing an arc will toggle to the complementary arc.

Each Undo (<%> key) will remove the last point located on the current arc. Each Undo past the current arc will remove one complete arc. The Undo buffer will remember the last 102 arcs drawn with this use of the command.

Arcs can be gapped using the Gap Arc command. The included angle of an arc can be changed by using the Move Arc endpoint command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Construction	= Radius or Diameter or Circumference or Corner	1	Radius	%AR.C	=Radius
Size	= number	2	none	%AR.S	=1.0
Class	= 0 thru 255	3	%WK.C	%WK.C	=0
Layer	= 0 thru 255	4	%WK.L	%WK.L	=0
Pen	= 0 thru 255	5	%WK.P	%WK.P	=0
STyle	= style	6	%WK.S	%WK.S	=Solid
PRiority	= 0 thru 255	7	%WK.PR	%WK.PR	=125
WOrkset	= workset	8	%WK.W	%WK.W	=none
Complement			none	%AR.COM	
Mimic			none	%AR.M	
coordinates			none	none	

Definition of Options

Construction =

This option specifies how the arc will be constructed. Valid values are: Radius, Diameter, Circumference and Corner.

Radius will construct an arc by defining its radius. The radius is the linear distance between the center (origin) and a point on the arc. Three points are needed to define each arc. They are the center point, start point and endpoint. The arc will be drawn between the start point and the endpoint and will always be less than 180 degrees (we call this the minor arc). If the major arc is desired, press the <Esc> or <clear> key on your keyboard after defining the arc. If the Size option is not used, the distance between the center point and the start point will be used as the radius value.

Diameter will construct a semicircle by defining its diameter. The diameter is the linear distance between the start point and endpoint of the semicircle. Two points are required to define each arc. If the Size option is used, the arc center point and start point are required. Otherwise, each semicircle will be drawn in a counterclockwise direction from the start point to the endpoint. If the opposite semicircle is desired, press the <Esc> or <clear> key after defining the arc.

Circumference will construct an arc by defining points along the circumference of the arc. Three points are required to define each arc. The arc will be drawn from the start point through an intermediate point to the endpoint. If the Size option is used, the arc center point, start point and endpoint are required. In this case, TurboCAD fixes the radius at the circumference value (size) entered divided by 2¹. If the complementary arc is desired, press the <Esc> or <clear> key on your keyboard after defining the arc.

Corner will construct an arc which is inscribed in the "corner" defined by two imaginary line segments which intersect at the corner. The resulting arc will be tangent to the two line segments. This option is similar to that of "filleting" a corner. The corner is defined by three points. The first point defines the start of the arc, the second point is the corner point, and the third point defines the end of the arc. The radius and included angle of the arc are influenced by both the length of the first imaginary segment and the included angle between the two segments. If the Size option is used, the radius will be fixed at the value entered.

Size =

This option sets the size of the arc. If not used (it is not necessary to enter a size), the size is determined solely by the points located on the screen. If the construct option is either Radius or Corner, the size means the radius of the arc. If it is Diameter, it means diameter, and if it is Circumference, it means the circumference of the complete circle, thereby defining a radius (i.e. radius=circumference/2¹).

Class =

See **Common Options** for more information.

Layer =

See **Common Options** for more information.

Pen =

See **Common Options** for more information.

STyle =

See **Common Options** for more information.

PRiority =

See **Common Options** for more information.

WOrkset =

See **Common Options** for more information.

Complement

This option causes the complementary arc to be drawn. That is the opposite arc than is normally drawn. This option does not negate or change the effect of the <Esc> key.

Mimic

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

ARC

Arcs will be drawn by defining three points. The distance between the first and second points determines the radius of the arc. The Class, Layer, Pen, and Style options will default to the values shown on the Status palette at the bottom of your screen.

AR Construct=Diam Size=1'3" STyle=Dot Pen=15

One hundred and eighty degree arcs will be drawn with a diameter of one foot, three inches. You will be prompted to locate a center point and start point for each semicircle. The arc will be dotted in style and drawn with pen fifteen. The Class, Layer and Priority will default to Status palette values.

a c 1 2 3 4 5 6 c

In this example the option values have been entered in strict positional order. Arcs with the circumference of one will be drawn. Class 2, Layer 3, Pen 4, Style 5 (Phantom line) and Priority 6 will be used. The Complement option at the end of the line means the opposite arc than would normally be drawn will be drawn.

Arc Mim St=Dot

You will be prompted to pick an existing arc on your drawing. The properties of the arc to be drawn will be set to those of the arc picked, except for Style, which will be set to Dotted. The new arc will be defined on the screen by center, start point and end point.

The Area command calculates the area and perimeter of an irregular polygonal shape. A cumulative total is displayed so that multiple areas can be added to or subtracted from one another. The format and accuracy of the results will depend on the current "Format =" and "Rounding =" values of the Preferences command.

After entering the command and a <Return>, you will be prompted to define the enclosed area to be calculated by locating points and/or picking objects on your drawing. You can use any of the appropriate Locate keys to locate points by snapping to existing objects in your drawing. To select an entire object for inclusion in the area enclosure, hold the <Shift> key down while picking the object with the mouse key. Pressing the (boundary) key will do the same thing. Only primitives (not entire groups) can be used as part of the area enclosure. The primitive can be part of a group, but only the primitive picked will be used.

As the area enclosure is being created, a temporary line will trace over it and rubberband to the crosshairs to show you the extent of the enclosure thus far. To close and calculate an area press the <E> (end) key. If you would like to define several areas before performing the calculation, press the <K> (close) key to close the current area and begin defining another. Each area is calculated independently. This means that overlapping and nested areas will be calculated twice. If you wish to subtract one area from another you must End it (<E> key), press the <Esc> key to toggle from adding to subtracting areas, and then define the next area. An example of this would be calculating a donut shape.

The <Return> key can also be used to close and end an area enclosure. However, this will also terminate the command.

Hot Tip: If you expect to be calculating an area several times during the course of working on your drawing, it is a good idea to draw a Line or Boundary primitive enclosing the area on a dedicated Layer or Class. It's quicker to pick a single primitive than to repeatedly recreate the area enclosure. The Layer or Class can be made invisible when not being used, and can also be turned off for plotting, if desired.

Each Undo (<%> key) will remove the last point located or object picked on the current enclosure. The calculation for the last area can be removed from the total by selecting Undo immediately after clicking "OK" on the calculation dialog. Each enclosed area cannot have more than 124 segments including the segment that closes it. A maximum of 50 areas can be calculated with each use the Area command.

The values generated by the Area command are remembered in the following Current Status Words until the next use of the command:

A Commands

<u>Value</u>	<u>CSW</u>
Area	%AA.Area
Total Area	%AA.TArea
Perimeter	%AA.Perimeter
Total Perimeter	%AA.TPerimeter

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Type	= Points or Objects	1	Points	%AA.T	=Points
Silent			none	%AA.S	
coordinates			none	none	

Definition of Options

Type =

This option specifies how the area enclosure will be defined.

Points means points can be located and/or objects picked to define the enclosure. A vectorized equivalent for objects like Splines, Arcs and Circles is used to create the enclosure.

Objects means only single objects will be picked to define the enclosure. If the object is not a closed object, like an arc, it will be closed automatically. This option is provided to obtain more accurate calculations on shapes like Circles, Arcs, Splines and Ellipses. When using the Points value, calculations can have a small error due to the fact that these primitives are vectorized to form the enclosure.

Silent

If this keyword-only option is used, the results of the calculation will not be displayed. However, the Current Status Word values will still be set. This option is most commonly used when area calculations are performed in a Macro.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

AA

You will be prompted to pick object or locate points to define an enclosed area you wish to have calculated. The area, perimeter, total area and total perimeter will be displayed for each area defined.

are, o, s

You will be prompted to pick a single object to define an enclosed area you wish to have calculated. The area, perimeter, total area and total perimeter will be calculated, but not displayed, for each object picked.

BELL or BL

2D & 3D

The Bell command sounds the bell at your terminal or computer (ding!).

Enter the command and a carriage return to execute and exit the command.

This command can be useful in Macros to attract your attention to some operation. You can string several Bell commands together to really drive yourself crazy.

Options

This command has no options.

Examples

BL;BE;BELL

The bell will ring three times (ding-ding-ding). The semicolon (;) is used to separate commands on a single command line (it is read as a <Return> by the TurboCAD command interpreter).

BOundary or BD

2D

The Boundary command creates irregular polygons (enclosed areas) that can be optionally filled with a bitmap (non vector) pattern.

After entering the command and a carriage return, you will be prompted to define an enclosed area by locating points and/or picking objects on your drawing. You can use any of the appropriate Locate keys to locate points by snapping to existing objects in your drawing. To select an entire object for inclusion in the boundary, hold the <Shift> key down while picking the object with the mouse key. Pressing the (boundary) key will do the same thing. Only primitives (not entire groups) can be used as part of the boundary. The primitive can be part of a group, but only the primitive picked will be used.

The vertices (corners) and segments (edges) of the resulting boundary can be edited (i.e. moved, deleted) just like the vertices and segments of a multi-segment line. The fill pattern will adjust to the new shape when one of these types of edits is performed.

As the boundary is being created, a line will rubberband to the crosshairs to show you the extent of the boundary thus far. To close the boundary and start a new one, press the <E> (end) key. If you would like to define several boundaries before the boundary is filled, press the <K> (close) key to close the current boundary and begin defining another. Several

boundaries may be nested and/or overlapped this way before they are filled. Nested and overlapped boundaries will alternate between being filled and not filled starting with the outermost boundary. An example might be filling a donut shape.

The <Return> key can also be used to close and end a boundary. However, this will also terminate the Boundary command.

Each Undo (<%> key) will remove the last point located or object picked to form the current boundary. A boundary cannot have more than 124 segments including the segment that closes it.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Fill	= 0 thru 127 or None	1	last	%BD.F	=0
Class	= 0 thru 255	2	%WK.C	%WK.C	=0
Layer	= 0 thru 255	3	%WK.L	%WK.L	=0
Pen	= 0 thru 255	4	%WK.P	%WK.P	=0
Style	= style	5	%WK.S	%WK.S	=Solid
PRiority	= 0 thru 255	6	%WK.PR	%WK.PR	=125
WOrkset	= workset	7	%WK.W	%WK.W	=none
FRame			none	%BD.FR	
Mimic coordinates			none	%BD.M	

Definition of Options

Fill =

This option selects the pattern that is to be used to fill the enclosed area. The Pattern value must be a number from 0 through 127 or the word "None". None means that the Boundary will be hollow (not filled with any bitmap pattern). Currently there are only 40 available patterns on the Macintosh.

Class =

See **Common Options** for more information.

Layer =

See **Common Options** for more information.

Pen =

See **Common Options** for more information.

B Commands

STyle =

See **Common Options** for more information.

PRiority =

See **Common Options** for more information.

WOrkset =

See **Common Options** for more information.

FRame

If this keyword-only option is used, TurboCAD will create a line around the perimeter of the boundary in addition to filling it with the selected pattern. This line will go in as a separate primitive and is not part of the boundary itself. This "frame" can be very useful if you later want to calculate the area enclosed in the boundary easily. It can also be deleted, moved to a separate Class or Layer at a later time, or edited like any other multi-segment line. Be careful, though, because editing this line (i.e. moving one of the vertices) will not automatically edit the boundary.

Mimic

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Bound

You will be prompted to pick objects or locate points to define a boundary you wish to have filled. The Pattern will default to the last used pattern and the FRame option will be inactive. The Class, Layer, Pen, and Style values will default to the values shown on the Status Palette at the bottom of your screen.

BD,none FR [a,2,2|4|,,2|-4|e|;]

The Boundary command will be entered and points will be located at the absolute coordinate 2,2, then 4 units to the right, then two units up, and then four units to the left. The <E> (end) key will then be entered, ending the boundary. The last coordinate is a carriage return (;) which will exit the command. What will be created is a non-filled rectangular area that is four units wide by two units high with a Frame around it. The Class, Layer, Pen, and Style values will default to the values shown on the Status Palette at the bottom of your screen. See the **Entering Coordinates** section for more information.

bo,,200,5

You will be prompted to pick objects or locate points to define a boundary you wish to have filled with the last pattern you used in this command. The boundary will be drawn on layer 200 with pen 5. The Class, and Style values will default to those shown on the Status Palette at the bottom of your screen.

bd m c=12

You will be prompted to pick an existing Boundary on your drawing (i.e. "m" stands for the Mimic option). The properties of the Boundary to be drawn will be set to those of the Boundary picked, except for Class, which will be set to 12.

Break or BK**2D & 3D**

The Break command breaks apart multi-segment Lines, Doublelines and Wireframes into two or more primitives of the same type. The resulting "pieces" are independent of one another. If an object other than a Line, Doubleline or Wireframe is picked (like a Circle, Rectangle, etc.) you will be asked if you would first like it converted into a line so you are able to break it. The object can be broken at all gaps, all vertices or selected vertices.

After entering the command and desired option, you will be prompted to pick the objects you wish to have broken. If you are breaking an object at vertices picked, the object will first be highlighted and then you will be prompted to pick the vertices at which you want the object broken. Temporary point markers will appear at the vertices you picked. If you are breaking an object at all gaps or all vertices, the object will not be highlighted, but temporary point markers will appear at either all the gaps or all the vertices of the object.

Once the desired objects and/or vertices have been selected, the <E> (end) key will perform the break without exiting the command. The <Return> key can also be used. However, this will also terminate the command.

Each Undo (<%> key) will remove the last object or vertex picked. You cannot Undo a break once it has been performed.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Type	= Vertex or All or Gaps	1	Vertex	%BK.T	=Vertex
Convert coordinates			none none	%BK.C	

Definition of Options**Type =**

This option selects the type of break you wish to perform.

B Commands

Vertex means that the picked object will be broken at selected vertices. After selecting an object, select the vertices at which to break it.

All means that the picked objects will be broken at all vertices.

Gaps means that the picked objects will be broken at all gaps. Gaps are invisible segments in the object. Pick the objects you wish to have broken.

Convert

Using this keyword-only option will bypass the safety question and automatically convert objects picked into lines suitable for breaking.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Break

The Type option will default to Vertex (i.e. vertices picked). You will be prompted to pick an object and then to select the vertices on that object at which you would like the object broken.

BR T=A Convert

Objects will be first converted into lines, then broken at all vertices. You will be prompted to pick the objects you wish to have broken.

B G

Objects will be broken at all gaps. You will be prompted to pick the objects you wish to have broken.

Calculator or CA

2D, 3D, & Macro

TurboCAD has a built-in, full-featured Calculator which is always active. This means it will activate itself any time TurboCAD recognizes that you are trying to calculate or evaluate something. Anything that the calculator evaluates or calculates is called an Expression. Expressions are "Processed" by the calculator. In order for expressions to be processed successfully, they must be valid. That is, they must be complete and clear, contain the right types of items, and be in an acceptable format.

The calculator will process both Numeric and String expressions. Numeric expressions are the ones you are most familiar with. That is, a numeric value is the result of processing the expression. String expressions deal with character strings. As you will see later, the result of a string expression may be a string or a numeric value. String expressions are very useful in TurboCAD Macros. When either type of expression is processed successfully, the calculator "Returns" the result to you for your use.

In addition to processing expressions as above, the calculator will also process Assignment Statements, which is when two expressions are separated by an "equals" (=) sign. These are typically used when you want to remember the results of an expression for later use. Assignment Statements store the results in memory locations which are called "Variables" in TurboCAD, and are similar to "memory" in hand-held calculators. You will typically use Assignment Statements in Macros.

You may enter a valid calculator expression directly on the command line, as a line in a Macro, or as the value of an option in any command which you execute directly or include in a Macro. If you are working on a Macintosh or other system which allows for options to be entered via dialogs, this means you can enter a valid expression any time you need to enter a numeric or string value on the dialog. The calculator will automatically process the expression and use the results as the value.

When you want to use the calculator simply as a calculator from the command line, enter the command (or its abbreviation or code) followed by a valid expression. Entering a <Return> will cause the calculator to process the expression and the result will appear in a window on the screen. If you enter an Assignment Statement on the command line, the statement will be processed and no results will be shown.

When the Calculator command is entered without an expression (i.e. CA followed by a <Return>), TurboCAD will display the current values of all Current Arguments and System Variables. See the Current Arguments and Variables sections for more information.

In order for calculator expressions to be acceptable as values for options in commands, we made the command itself optional. The calculator command is the only TurboCAD command with this facility. This means that the numerical expressions "CA (3 + 2)" and "(3 + 2)" will both result in (<Return>) the number "5" when entered on the command line. When you choose to omit the command, the first character in the expression must be an "open

parenthesis" as above. This will eliminate the possibility of TurboCAD interpreting your entry as some other command. Remember to close the parentheses at the end of your expression.

The following paragraphs define the format, permissible entries, functions, and operators available to you in the TurboCAD calculator.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
expression		1	none	none	

Definition of Options

Expression

An expression is any alphanumeric character string that makes sense as something which can be processed (evaluated) to return (result in) either a number or string. It can range from a single number or character to a complex algebraic equation containing both numbers and strings. The important thing is that it make sense and be logical. When TurboCAD processes a valid expression, the results are remembered (until the next time the calculator is accessed) in the TurboCAD variables (called Current Status Words) %CA.String and %CA.Number, depending on whether a string or number was returned.

Expressions can be continued across lines and parentheses may be used to change the order of execution. An expression can contain the following types of items:

DATA

VARIABLES and CURRENT STATUS WORDS

OPERATORS

FUNCTIONS

Expressions which contain an equals (=) sign are

ASSIGNMENT STATEMENTS

Operators and Functions work according to a

PRECEDENCE TABLE

Each of the capitalized headings above are described in the following paragraphs.

Data

Data (also called operands) are the actual numbers and character strings that are to be processed by the calculator in conjunction with functions and operators used in the expression. The types of data that the calculator can use are listed here:

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<u>Data Type</u>	<u>Examples</u>
Numbers	34, 1000.5, .0001
Fractions	1/2, 3.5/7, 2.5/7.5
Feet/Inches entries	12'3". -3' 4.5", 3.5', 12'3+1/2"
Deg/Min/Sec entries	45°16'10", -0°2'45"
Strings	abcde, Watusi
Quoted Strings	\Total Area = %AA.ARea\ 'Enter desired size:'

Numbers and Fractions

Numbers are just numbers and can be entered in regular decimal, fractional, scientific, feet/inches, and degrees/minutes/seconds notation.

Decimal notation is standard decimal notation as in the examples above. Fractional notation is also permissible, such as "1/2" or "-3.5/7". As you can see, fractional notation is really an expression containing decimal numbers and some Operators, and you do not have to limit yourself to whole numbers in the fraction. Standard scientific notation is also allowable. For example, 1.234e5 for 123400, and 123e-3 for .00123. Remember, no commas to depict thousands and no spaces, because commas and spaces will cause the number to be interpreted incorrectly.

Feet/Inches Entries

If the Work units of your drawing is set to Feet or Inches, the calculator will process numbers in feet/inches format. Again, these are actually expressions but are covered here because they can be interpreted as a specific format for numbers.

This format uses the the foot (') and inch (") marks along with numbers and fractions. You should already know that TurboCAD will accept either Feet or Inches separately by using these marks (Basics, Entering Coordinates). Feet/inches entries are used when you want to enter both feet and inches together and require that you use both marks in the expression.

You start by entering a number for feet followed by a foot mark ('), then a number for inches followed by an inch mark ("). The inches will be added to the foot value (multiplied by twelve, of course). The entire expression is then converted into the current Work Units (either feet or inches).

If the number following the foot mark is not followed by an inch mark, it is assumed to be a separate and unrelated number. If the foot value is a negative number, the whole feet/inches expression is made negative. If fractions of an inch are entered after the whole inches, a plus sign (+) must be used to tie the two numbers together. In fact, this allows several inch and/or fractional inch values to be strung together. As you can see in the examples below, decimal and fractional entry can be mixed with feet/inches entry. Also note that spaces do not affect the feet/inches entry - spaces are optional.

Examples of valid feet/inches entry:

<u>Example</u>	<u>Meaning</u>
12'3"	Twelve feet, three inches.
12'3+1/2"	Twelve feet, three and one half inches.
2'3+3/8+1/4"	Two feet, three and five eighths inches.
12' 1/2 + 3"	Twelve feet, three and one half inches.
-3' 4"	Negative three feet, four inches.
10'	Ten feet.
3.5'	Three and a half feet. (Three feet, six inches)
-0'1.25"	Negative one and one quarter inches.
0'1+1/4"	One and one quarter inches.
-3/4"	Negative three quarter inches.
-0'3/4"	Negative three quarter inches.

Examples of invalid feet/inch entry:

<u>Example</u>	<u>Won't work because</u>
12'3	Missing inch mark will cause an error.
3'-5 8"	Space is separator. 3'-5 and 8" will be read as two separate numbers.
3"	Missing inch value.
-'3/4"	Missing foot value.

When you want to add or subtract two or more entries in this notation, each should be enclosed within parentheses to insure that the calculator will process everything in the right order. For example, $(1'2+3/4") + (2.5')$ is the best way to add one foot, two and three quarter inches to two and a half feet.

Remember that all math functions are performed before the feet and inch conversions are performed. This is because the ' and " marks have very low precedence (see Precedence Table), meaning that the calculator evaluates them last, just before assignment. In an expression like $2+3/4"$, the $3/4$ will be divided first and then added to the 2 before the inch conversion is done. Otherwise, the calculator would divide 3 by 4 inches and add the result to 2, which would be read as $2'3/4"$ if your current Work units were feet.

degrees/minutes/seconds entries

Although TurboCAD deals with all angle values internally as decimal numbers, you have the option of entering angle values in a degrees/minutes/seconds format. The decimal format looks like: 12.5, 5, .02. The degrees/minutes/seconds format looks like: 12~24'45". The tilde (~) is used as the degree sign because not all computer keyboards have a degree character. (On the Macintosh, the degree sign can also be used. It is Shift-Option-8 on the keyboard.)

You don't have to enter minutes or seconds if they are zero, but if you enter a number you must follow it with a unit type so it knows what the number is. The inverse is also true. You cannot show a unit type without a value. The ~ must be entered even if the degrees are zero. That's because entering 24'45" would be processed as feet and inches. Like feet and inches, if the degrees value is negative the entire expression is made negative.

Valid examples of degrees/minutes/seconds entry:

<u>Example</u>	<u>Meaning</u>
45°16'10"	Forty-five degrees, sixteen minutes and ten seconds.
-0°0'45"	Negative forty-five seconds.
30.5°30'	Thirty-one degrees.
15°	Fifteen degrees
5°0'45"	Five degrees, forty-five seconds.

Examples of invalid degrees/minutes/seconds entry:

<u>Example</u>	<u>Won't work because</u>
45°16	No trailing unit type (minutes)
4'5"	Will be read as feet and inches.
°15'	No value for degrees.
3°60"	No value for minutes.
60°"	No value for minutes and seconds.

If you enter a degrees/minutes/seconds value while your current Angle unit (%PF.Angle) is set to Radians, the conversion will be done automatically.

Strings

Strings can contain any character, including spaces, tabs, commas, special characters, and numbers. When numbers are mixed with other characters, then will be interpreted as part of the string rather than numbers(except for scientific notation as described above). Strings which contain spaces must be enclosed in quotes in order for them not to be interpreted as separate options of the command in which they are used.

Quoted Strings

There are two types of Quoted Strings. The first uses the backslash character (\) and the other uses the single back quote mark (`). The back quote is not the single quote on the same key as the double quote mark, but rather the one that is typically on the same key as the " ~ ". Remember that there must be one of these at the beginning and another at the end of the string.

Strings quoted with the backslash (\) will substitute any Variable or Current Status Word within the string with its current value before processing the string For example, in the expression\%TX.Height + .5\, the calculator would first replace the Current Status Word %TX.Height with its current value (the current Text Height) and then add .5 to it. Strings quoted with the single back quote character (`) will be read exactly as entered (literally) and no variable substitution will be performed.

Variables and Current Status Words

TurboCAD always remembers the current settings of options in commands, distances entered by you, external files read by Macros, and so forth. All of these things are remembered in

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special variables called Current Status Words (CSWs) and Current Arguments. Furthermore, TurboCAD allows you to store points, numbers, and strings in variables whose names are pre-defined or to which you assign a name. These types of variables are called Coordinate Variables, System Variables, Global Variables, and Local Variables.

Any and all of these variables can be used in a calculator expression. A variable is first replaced with its current value and then treated the same as other data, unless it is quoted as described above. See the Variables, Current Status Words, and Current Arguments sections for more information on variables.

<u>Variable Type</u>	<u>Examples</u>
Current Status Words	%CC.Size, %DS.T, %GD.XLarge
Current Arguments	ARG1, arg4
Coordinate Variables	Point.X, loc.k, a
System Variables	A, x
Global Variables	name , Width
Local Variables	&Length, &count

The last two types of variables, Global Variables and Local Variables, cannot be used from the command line. They can only be used in a Macro.

Operators

Operators are used in expressions to perform calculations, evaluate the relationship between values, or to associate items in logical expressions. The TurboCAD calculator uses the following types of Operators, and each is defined in the paragraphs which follow.

MATH OPERATORS

RELATIONAL OPERATORS

LOGICAL OPERATORS

Math Operators

Math Operators perform mathematical operations to the data associated with them. Valid math operators are:

<u>Math Operator</u>	<u>Function</u>	<u>Precedence</u>
+	Unary Plus (Absolute value)	1
-	Unary Minus (Negation)	1
**	Exponentiation	2
*	Multiplication	3
/	Division	3
+	Addition	4
-	Subtraction	4

Precedence simply means who goes first. Often expressions involve more than one operator, and the result will depend on the order in which the operations are performed. We have established "precedence" rules to determine what gets done first. You will notice in the above table that multiplication and division have the same precedence, as do addition and subtraction. When adjoining operators have the same precedence, they are processed left to right. In the example:

$$-41 - 6 + 12 * 4$$

41 would first be made negative, 12 would be multiplied by 4 to get 48, 6 would be subtracted from -41 to get -47, which would then be added to 48 to get the result 1. The negation of 41 is done first because it has the highest precedence, followed by multiplication. Subtraction and addition are processed left to right because they have the same precedence.

A unary operator is a sign that operates on a single value or operand. The most common unary operator is a minus sign preceding a single number thereby making it a negative number. Unary plus is also supported. To be sure that unary plus or minus is not executed as addition or subtraction it must be the first item in an expression or be enclosed in parentheses within the expression. For example, $(- \%RE.Width)$ would be a negative rectangle width and $-17 + 5$ would be a negative twelve.

In exponentiation, the "left operand" is raised to the "right operand" power. In multiplication, the operand on its left will be multiplied by the operand on its right. In division, the operand on its left will be divided by the operand on its right.

The addition (+) and subtraction (-) operators may be used to either assign a "sign" to a number or to add or subtract two numbers. When used to assign a "sign" (as unary operators), they have the highest precedence. When used to add or subtract, the operand on its right will be added to or subtracted from the operand on its left. Note that the unary expression must be enclosed in parentheses or be at the start of an expression.

Relational Operators

Relational operators establish a relationship between one value and another. When used in a calculation, the expression is evaluated and the calculator returns a true (1) or false (0) answer. For example, the expression $45 \geq 30$ is true and the result would therefore be 1.

Relational operators are most commonly used in the "If" Macro command. See the If command for more information.

<u>Relational Operator</u>	<u>Meaning</u>
<	Less Than
<=	Less Than or Equal To
>	Greater Than
>=	Greater Than or Equal To
==	Equal To
^=	Not Equal To (Inequality)

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Note: The <= and >= operators cannot be entered as =< and =>. They must be entered as shown above. The == sign should not be confused with the assignment statement (=). The assignment statement is used to assign a value to a variable, while the 'Equal To' operator is used to test for equality.

Logical Operators

Logical operators allow several relational expressions to be joined together to form more complex conditions.

Logical operators are most commonly used in the "If" Macro command. See the If command for more information.

<u>Logical Operator</u>	<u>Meaning</u>
^	Not (Reverses Value)
&&	And
	Or

The operators are listed in their order of precedence. All logical operators have a lower precedence than relational operators. The logical operators && (and) and ||(or) must have a value before them and a value after them to make sense. The ^ (not) operator affects only the value which follows it.

Logical expressions may use parentheses to group conditions into specific relationships. For example, the expression: (A == B && (C == 0 || C == 2)) will only be true if A is equal to B and if C is equal to either zero or two.

Functions

Functions are used in expressions to clarify processing order, perform standard mathematical functions, convert units, and to perform string functions. The TurboCAD calculator uses the following types of Functions, and each is defined in the paragraphs which follow.

GROUPING FUNCTIONS

MATH FUNCTIONS

CONVERSION FUNCTIONS

STRING FUNCTIONS

Grouping Functions

Grouping gives you the ability to establish the order in which an expression will be processed. Grouping is done by using the left and right parentheses "()".

<u>Grouping Function</u>	<u>Meaning</u>
(Starts the group
)	Ends the group

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Groups may be nested by entering one set of parentheses within another. The part of an expression that is in the innermost pair of parentheses will be processed first, then the next innermost, and so on throughout the expression. Non-nested pairs of parentheses will be processed left to right. We can alter the result of the previous example by adding parentheses like this:

$$(-41 - (6 + 12)) * 4$$

In the above, 6 would first be added to 12 to get 18, 18 would then be subtracted from -41 to get -59, which would then be multiplied by 4 to get the result -236. A far cry from the result of 1 that we would have gotten if there were no parentheses altering the processing order.

Math Functions

The calculator provides many built-in math functions for frequently performed mathematical operations. Case is unimportant when entering math functions. Math function names cannot be abbreviated, they must be spelled out as shown. *number*, *number1* and *number2* represent an operand or numeric expression that must be in parentheses.

<u>Math Function</u>	<u>Meaning</u>	
SIN	(number)	Trigonometric sine of angle number
COS	(number)	Cosine of angle number
TAN	(number)	Tangent of angle number
ASIN	(number)	Arcsine of number
ACOS	(number)	Arccosine of number
ATAN	(number)	Arctangent of number
ATAN2	(number2)	Arctangent of number1 and number2
SQRT	(number)	Square root of number
LOG10	(number)	Common log (base 10 logarithm) of number
LOGE	(number)	Natural logarithm of number
FIX	(number)	Truncates fractional portion of number
RND	(number)	Rounds number to whole number (.5 rounds up)
MOD	(number1 number2)	Modulus - Remainder of <i>number1</i> divided by <i>number2</i>

The functions SIN COS and TAN expect the units of number to be the current type as set by the Angle option of the Setup command (ie: Degrees or Radians). The Arc functions will return an angle value in the same units. Functions may be nested to any depth and have the same precedence as exponentiation. They also execute right-to-left. The two numbers required for ATAN2 and MOD must be separated with a space, not a comma.

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Conversion Functions

The calculator can assist in the conversion between different units. This allows you to enter distance values in the most convenient unit regardless of the the Work units of the drawing you are working on. Case is unimportant when entering conversion functions. Function names cannot be abbreviated and they must be spelled out as shown. Items in italics represents an operand or numeric expression which you enter. There are only two conversion functions, one is the generic conversion function, and the second is a special one provided for your convenience.

Conversion Function Meaning

UNIT (number units from units to) Converts number from "units from" to "units to"
FTIN (number) Converts number into feet/inches format

The general UNIT() function above will convert between any two units regardless of the current Work Units. Allowable Units in TurboCAD are:

<u>Abbreviation</u>	<u>Units</u>
ML	Mils
IN	Inches
FT	Feet
MI	Miles
MM	Millimeters
CM	Centimeters
M	Meters
KM	Kilometers
DEG	Degrees
RAD	Radians
GRAD	Gradians

Always use the abbreviation. For example, if you are working on a drawing whose Work units are Inches and specified a line to be drawn "unit(20 cm in)" long, the line would be approximately 7.874016 inches on the drawing. Twenty centimeters is converted into our current work unit - inches.

It is important to remember that the result of a conversion will be interpreted in the current Work Units when used to specify the value of an option. In the example above, assume that the current Work Units are Feet instead of Inches and that you entered the conversion expression in the following context:

Rectangle Width=unit(20 cm in) Height=unit(40 cm in)

The resulting rectangle would be 7.874016 feet wide and 15.748032 feet high because the conversion function merely applies a multiplier and returns a value which then is processed in

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the current Work Units. If you really wanted these values to be in inches, then the correct command would be:

```
Rectangle Width=unit(20 cm in)" Height=unit(40 cm in)"
```

To be safe, generally enter the current Work Units as the "units to" value, or enter the Current Status Word, %SU.Work, for this value.

The second conversion function, FTIN() merely changes the format of the number into Feet/Inches format. For example, the expression "FTIN(6.5)" will return the value 6'6" if your Work Units are Feet and 6 1/2" if your Work Units are Inches.

String Functions

The calculator will perform the following string functions. Case is unimportant when entering string functions. Function names cannot be abbreviated, they must be spelled out as shown. string, string1 and string2 represent an alphanumeric character string, and number, number1 and number2 represent an operand or numeric expression that must be in parentheses.

<u>String Function</u>	<u>Meaning</u>	
LEN	(string)	Counts the number of characters in string
SUB	(string number)	Returns the string from number thru the end of string
SUBN	(string number1 number2)	Returns the string from number1 thru number2 of string
CAT	(string1 string2)	Concatenates string2 to string1
FIND	(string1 string2)	Finds numeric start of string2 within string1
NUM	(string)	Converts string into a number
STR	(number)	Converts number into a string

LEN returns the number of characters in string. If string has not been assigned a value or it contains all blanks, a zero is returned. Trailing blanks are counted. For example, LEN (How soon? `) would be 12.

SUB returns the portion of string beginning at the character position number and ending at the end of the string. Character positions are numbered from left to right beginning at 1. For example,

```
SUB (around` 2) would return `round`.
```

SUBN returns the portion of string beginning at the character position number1 and ending at number2. Character positions are numbered from left to right beginning at 1. For example, SUBN (starting` 3 5) would return `art`.

CAT will concatenate (join together) string1 and string2. For example, if &Temp = `A`, the result of CAT (Part` &Temp) would be `Part A`.

FIND searches for string2 within string1. If string2 is found, FIND returns the number of the character position in string1 where the match began. If the search is not successful a zero is returned. For example, FIND ('Hi, Mom!' '!') returns 8.

NUM takes string and converts it into a number. string is expected to contain only digits, blanks and no more than one decimal point (unless scientific notation is used). Blanks are interpreted as zeros. The value of NUM ('375') would be 375.

STR converts number into a character string. If the number value was negative the character string will be preceded with a minus sign (-). Up to six places of accuracy will be displayed after the decimal point.

Note: For functions that require more than one string or number value, the value must be separated with a space or tab. Commas will cause an error.

Assignment Statements

The assignment statement remembers the value returned by an expression in a variable.

<u>Assignment</u>	<u>Meaning</u>
=	Places value on right into variable on left

The variable must be on the left of the equals sign and the expression on the right. Using the previous example, we can assign its result to the TurboCAD System Variable A using the following statement:

$$A = (-41 - (6 + 12)) * 4$$

The result -236 is remember in A for later use. Several variables can be set in a single assignment statement. It might look like this:

$$A = c = \text{answer} = 47 + \text{sqrt}(36)$$

The variables A, c, and answer will all be set to 53. Because the variable answer is a user global variable, it cannot be used from the command line. This example would have to be run in a Macro. See the section on Variables for more information on variable types and uses.

Precedence Tables

Here is the precedence table for all of the above functions and operators.

C Commands

Operator/Function	Associativity	Precedence
()	Left-to-Right	1
+ - (unary)	Right-to-Left	2
^	Right-to-Left	3
**	Right-to-Left	4
* /	Left-to-Right	5
+ - (unary)	Left-to-Right	6
All Math Functions	Left-to-Right	7
All Conversion Functions	Left-to-Right	7
All String Functions	Left-to-Right	7
< <= > >=	Left-to-Right	8
== ^=	Left-to-Right	9
&&	Left-to-Right	10
	Left-to-Right	11
"	Right-to-Left	12
'	Right-to-Left	13
~	Right-to-Left	14
=	Right-to-Left	15

Examples

For the following examples, Me has a value of 4, You has a value of 3 and the current Angle unit is set to Degrees.

CALC Me + SQRT (25)

The result of this expression is 9. The SQRT function is executed first, and its result added to the value of Me.

Sin (sqrt(45**(4/2)))

The result of this expression is .707107. First, four was divided by two to get two, and then 45 was raised to the power of two. Then the square root of this value (2025) was taken (45). The sine of 45 degrees is .707107. One additional overall set of parentheses would have to be added in order to enter this expression on the command line.

sqrt ((%CD.X - You) ** 2 + (%CD.Y - Me) ** 2)

This is the formula for calculating the distance between two points. The X and Y axis components of the points being represented by %CD.X, %CD.Y and You, Me. (%CD.X and %CD.Y are the Current Status Words for the X and Y axis components of the last point located.) The quantities are first subtracted and the results squared, then added together, and finally the square root of the sum is taken. If %CD.X and %CD.Y are both equal to 7, the expression would be processed result in 5.

The Chamfer command creates multi-segment, rectilinear lines with chamfered corners. Rectilinear means that all of the segments of the lines are at right angles (perpendicular) to each other. Chamfered means that the corners are diagonally cut. This command can also convert existing Line primitives into Chamfers. Existing lines which are converted do not have to be rectilinear.

In this command, you have to define the angle for the first segment. Subsequent segments will alternate between being drawn perpendicular to the first and in line with the first.

Once you define the angle, each point located will define two segments. The first segment will be the component along the angle line and the second will be the component perpendicular to the angle line. Pressing the <Esc> or <clear> key will alternate the order of these components. The corner between the two segments will be chamfered.

The <K> key will close the current chamfer by returning to its starting point (origin). If the first and last point of a chamfer are in the same location then the resulting corner will also be chamfered.

Each Undo (<%> key) will remove the last point located.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Xsize	= number	1	last	%CF.X	= 1
Ysize	= number	2	last	%CF.Y	= 1
Angle	= angle or Point (or Pt)	3	Point	%CF.A	= Point
Class	= 0 thru 255	4	%WK.C	%WK.C	= 0
Layer	= 0 thru 255	5	%WK.L	%WK.L	= 0
Pen	= 0 thru 255	6	%WK.P	%WK.P	= 0
STyle	= style	7	%WK.S	%WK.S	= Solid
PRiority	= 0 thru 255	8	%WK.PR	%WK.PR	= 125
WOrkset	= workset	9	%WK.W	%WK.W	= 125
COnvert			none	%CF.CO	
coordinates			none		

Definition of Options

Xsize =

This option sets the X-component of the chamfer. Imagine a right triangle whose hypotenuse is the chamfer and whose sides are along the segments which define the corner to be chamfered (these segments are always at 90 degrees). The X-component is along the first of the two segments. The Xsize value can be any non-negative number or zero. Entering a value of zero will cause the command to create rectilinear lines without chamfered corners.

Ysize =

This option sets the Y-component of the chamfer. Imagine a right triangle whose hypotenuse is the chamfer and whose sides are along the segments which define the corner to be chamfered (these segments are always at 90 degrees). The Y-component is along the second of the two segments. The Ysize value can be any non-negative number or zero. Entering a value of zero will cause the command to create rectilinear lines without chamfered corners.

Angle =

This option sets the angle of the first segment of the chamfered line. All subsequent segments will either be in line or perpendicular to the first segment. Any angle value between -360 and 360, or the word Point (or Pt) are permissible. A value of zero means horizontal, unless the Base Angle has been changed by using the Preferences command. If Points is used for this option, the first point located after the starting point will define the angle.

Class =

See **Common Options** for more information.

Layer =

See **Common Options** for more information.

Pen =

See **Common Options** for more information.

SStyle =

See **Common Options** for more information.

PRiority =

See **Common Options** for more information.

WOrkset =

See **Common Options** for more information.

COnvert

This option allows you to convert existing Lines on your drawing into Chamfered lines. You will be prompted to pick an existing Line on your drawing instead of locating points to create the Chamfered line.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

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Examples

Chamfer X=1/2 Y=1/2

A multi-segment, rectilinear line will be drawn with chamfered corners. Each chamfer will have X and Y components of 1/2 Work Units, causing them to be at 45 degrees. After locating the starting point, you will be prompted to locate a point to define the angle of the first segment. All subsequent segments will be perpendicular to the previous one. Class, Layer, Pen, and Style options will default to the Status Palette values.

Cf,,,30 p=4

A multi-segment, rectilinear line will be drawn with chamfered corners. Each chamfer will have X and Y components equal to those used the last time this command was executed. The angle of the first segment will be fixed at 30 degrees. All subsequent segments will be perpendicular to the previous one. The chamfer will be drawn in Pen 4. Class, Layer, and Style options will default to the Status Palette values.

CHAM co

You will first be prompted to pick an existing multi-segment line on your drawing. It will be converted into a Chamfered line with X and Y components equal to those used the last time this command was executed. The Class, Layer, Pen, and Style options will assume the values of the line picked. If what you pick is not a line (i.e. a Rectangle), you will be asked whether TurboCAD should convert the object into a line prior to chamfering the corners.

CHange or CG

2D & 3D

The Change command lets you change just about all of the "properties" of objects on a drawing. Properties is the word we use to describe the parameter that define an object. Properties include general properties such as the object's Scale, Size, Layer, Font, Style, Angle, plus all of the specific properties particular to a given object. Different types of objects have different properties. For example, a Circle has a Size property, while Text does not. Text has a Font property, while a Circle does not.

After entering the command and the properties you would like changed, you will be prompted to pick the objects on your drawing you want changed. If you pick an object that does not carry all of the properties you have entered, the changes will not be made.

Properties fall into two main categories: Group properties and Member (or Primitive) properties. A single, ungrouped, primitive is also referred to as a Member. Changing a group property affects all of the members of the selected group, where changing a member property affects only the selected primitive or group member. Group properties consist of: Tag, Class, Priority, X, Y and Z axis Location on the drawing, X and Y axis Scale and rotation Angle. All other properties are primitive properties. Changing a property of a group member affects the

structure of the group. If there are copies of the group, you will be asked if you wish to make a global change. Changing a group property cannot be done globally.

You may want to review other Change commands available in TurboCAD, such as Change Group, Change Member, and Change Object to find the one which best suits a particular need.

Undo (<%) key) can only undo the change made to the last object picked while you are still in the command. This is because this command has so many options to keep track of. In other commands, you can typically undo everything you have done while in the command. This is the trade off for having such a powerful command available.

This command has so many options that positional order is impractical. Use the "Keyword=" format when using this command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Select	= select	-	%PF.S	%CG.S	=Pick
Object	= object	-	%PF.O	%CG.O	=Any
Global	= global	-	%PF.G	%CG.G	=Ask
Class	= 0 thru 255	-	none	none	
Layer	= 0 thru 255	-	none	none	
Pen	= 0 thru 15	-	none	none	
STyle	= style	-	none	none	
PRiority	= 0 thru 255	-	none	none	
Extstyle	= style	-	none	none	
CAp	= None	-	none	none	
	or End				
	or Both				
	or Gaps				
	or All				
	or Lines				
Marker	= Dlamond	-	none	none	
	or Dot				
	or Period				
	or Plus				
	or Asterisk				
	or O				
	or X				

C Commands

	or Square			
	or DOTSquare			
	or DOTDiamond			
	or XSquare			
Placement	= None	-	none	none
	or Start			
	or End			
	or Both			
Slides	= 3 thru 120	-	none	none
Justification	= justification	-	none	none
Font	= 0 thru 7	-	none	none
FONTName	= font	-	none	none
ATtribute	= 1 thru 127	-	none	none
SLant	= -45 thru 45	-	none	none
Leader	= Arrow	-	none	none
	or Openarrow			
	or Closedarrow			
	or Dot			
	or Up			
	or Circle			
	or Square			
	or Down			
	or Loop			
	or Box			
	or Slash			
	or Squiggle			
Type	= Incremental	-	none	none
	or Baseline			
	or Left			
	or Right			
	or Centered			
	or String			
	or Datum			

Dimension	= number	-	none	none
SHow	= switch	-	none	none
Units	= unit	-	none	none
FORmat	= Fractional or Integer or One or Two or THree or Four or Flve or Six or Float	-	none	none
Fill	= 0 thru 127	-	none	none
Height	= number	-	none	none
Width	= number	-	none	none
LLength	= number	-	none	none
LWidth	= number	-	none	none
THeight	= number	-	none	none
TWidth	= number	-	none	none
HOrizontal	= number	-	none	none
Vertical	= number	-	none	none
OFFset	= number	-	none	none
SIze	= number	-	none	none
MAJor	= number	-	none	none
MInor	= number	-	none	none
ROunding	= number	-	none	none
SPacing	= 0 thru 1	-	none	none
EXTension	= number	-	none	none
GAp	= number	-	none	none
OVerrun	= number	-	none	none
ANgle	= angle	-	none	none
STArting	= angle	-	none	none
Included	= angle	-	none	none

C Commands

TAg	= string	-	none	none
coordinates			none	none

Definition of Options

Select =

See **Common Options** for more information.

Object =

See **Common Options** for more information.

Global =

See **Common Options** for more information.

All Other Options

All of the remaining options of this command are properties of specific objects that you can change. Instead of using twenty pages of the manual here to describe them all, I am going to refer you to the appropriate command. Listed below is the name of the Change command option and the commands that describe the possible settings for the option. This list is also handy for seeing which properties are changeable for a given object type.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

C Commands

<u>Option</u>	<u>See Command</u>		
Class	Line	Chamfer	Fillet
	Leaderline	Extstyle	Dimension
	Fillet	CAP	Doubleline
	Doubleline		Wall
	Rectangle	Marker	Point
	Leaderline	PLacement	Leaderline
	Spline	Sides	Polygon
	Text	Justification	Text
	Arc	Font	Text
	Circle	FONTName	Text
	Polygon	ATtribute	Text
	Ellipse	SLant	Text
	Points	Leader	Leaderline
	Crosshatch		Dimension
	Dimension	Type	Dimension
	Boundary	Dimension	Dimension
	Group	SHow	Dimension
Wireframe	Units	Dimension	
<u>Layer</u>	Same as Class option above	FORmat	Dimension
	(Except Group)	Height	Rectangle
Pen	Same as Class option above	Width	Rectangle
	(Except Group)		Doubleline
STyle	Same as Class option above	LLength	Dimension
	(Except Group, Text and Point)	LWidth	Leaderline
			Dimension
PRiority	Same as Class option above	THeight	Leaderline
			Dimension (Text)
		TWidth	Text
			Dimension (Text)
			Text

HOrizontal	Dimension (Text) Text	SPacing	Dimension Dimension
VerTical	Dimension (Text) Text	EXTEnsion	Dimension
OFfset	Dimension (Text) Text	GAp	Dimension
		OVerrun	Dimension
		ANgLe	Same as Class above (Except Wireframe)
SIZe	Arc Circle	STArting	Ellipse Arc
MAJor	Ellipse	Included	Ellipse
MInor	Ellipse		Arc
ROunding	Dimension		

Examples

Change Style=CL

You will be prompted to pick the objects you wish to change. The Style of each object picked will be changed to Centerline. If a picked group has copies, the appropriate global change will be made based on the current setting of the Global option of the Setup command.

cg sides=8

You will be prompted to pick the objects on your drawing you wish to have changed. Because A polygon is the only primitive type that has a Sides property, picking any other type of object will be ignored.

CG CL=45 SEL=IN

All objects completely inside a rectangular area that you define by locating its diagonally opposite corners will be changed to Class 45.

CG OBJ=RECT SEL=ALL PEN=3 WIDTH=13

All Rectangles on the drawing will be changed to Pen 3 and their Width will become 13 Work Units.

CG THEI=.5 SEL=IN

All text within a rectangular area will be changed to text height value of .5 Plot Units. Remember that Text Height is defined in Plot Units.

CHange Group or CGG

2D & 3D

The Change Group command lets you change the Class and Priority values of groups on your drawing.

After entering the command and the Class and/or Priority you would like changed, you will be prompted to select the objects on your drawing you want changed.

Each Undo (<%> key) will return the last changed object or box of objects to its original properties. The Undo buffer will remember the last 170 objects changed with this use of the command.

This command can also be entered as "CHANGE_Group" or "CG Group".

Although the Change command can also change these properties of a group, by limiting the properties that can be changed to just Class and Priority the Undo buffer can contain many more objects. This makes it possible to Undo changes made when selecting objects by box; something the Change command cannot do.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Class	= 0 thru 255	1	none	%CGG.C	=0
Priority	= 0 thru 255	2	none	%CGG.P	=125
Select	= select	3	%PF.S	%CGG.S	=Pick
Object	= object	4	%PF.O	%CGG.O	=Any
coordinates			none	none	

Definition of Options

Class =
See **Common Options** for more information.

Priority =
See **Common Options** for more information.

Select =
See **Common Options** for more information.

Object =
See **Common Options** for more information.

coordinates
See **Locating Points** and **Entering Coordinates** for more information.

Examples

Change Group Class=200

You will be prompted to pick the objects you wish to change. The Class of each object picked will be changed to a value of 200.

cgg 1 pr=40 s=in

All objects completely inside a box that you define by locating its diagonally opposite corners will be changed to Class 1 and Priority 40.

CG G,90,,A,T

All ungrouped Text on your drawing will be changed to Class 90. Text that is part of a group will not be changed.

CHange Member or CGM

2D & 3D

The Change Member command lets you change the Layer, Pen and Style values of group members or ungrouped primitives on your drawing.

After entering the command and the Layer, Pen and Style you would like changed, you will be prompted to select the objects on your drawing you want changed.

Each Undo (<%> key) will return the last changed object or box of objects to its original properties. The Undo buffer will remember the last 102 objects changed with this use of the command.

This command can also be entered as "CHANGE_Member" or "CG Member".

Although the Change command can also change these properties of a member, by limiting the properties that can be changed to just Layer, Pen and Style the Undo buffer can contain many more objects. This makes it possible to Undo changes made when selecting objects by box; something the Change command cannot do.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Layer	= 0 thru 255	1	none	%CGM.L	=0
Pen	= 0 thru 255	2	none	%CGM.P	=0
Style	= style	3	none	%CGM.S	=Solid
SElect	= select	4	%PF.S	%CGM.SE	=Pick
Object	= object	5	%PF.O	%CGM.O	=Any
Global	= global	6	%PF.G	%CGM.G	=Ask
coordinates			none	none	

Definition of Options

Layer =

See **Common Options** for more information.

Pen =

See **Common Options** for more information.

Style =

See **Common Options** for more information.

SElect =

See **Common Options** for more information.

Object =

See **Common Options** for more information.

Global =

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Change Member Layer=200

The Layer of each object picked will be changed to 200. If a picked group has copies, the appropriate global change will be made based on the current setting of the Global option of the Preferences command.

cgm l=1 p=40 st=d se=in obj=li

All line primitives completely inside a box that you define by locating its diagonally opposite corners will be changed to Layer 1, Pen 40 and line Style Dotted. Lines which are members of groups will not be affected.

CHange Object or CGO

2D & 3D

The Change command lets you change the "properties" of an object on your drawing. Properties are the parameters that define an object. Properties of primitives (members) include such things as the object's Radius, Layer, Font, Style, Width, and more. Properties of groups include such things as the group's Class, Rotation Angle, Scale, etc. Primitives which are not part of a group also have these "Group" properties.

The difference between this Change command and the other Change commands (CHANGE, CHANGE MEMBER, CHANGE GROUP) is that you do not have to decide what changes you will make before you pick the object. This is a very powerful tool. When you pick the object, TurboCAD displays a dialog which contains the current properties of the object and lets you change any of them.

The only thing you have to decide while entering this command is whether you want to change the object's member properties (i.e. primitive properties) or its group properties. This is set by the "Type" option as defined below.

You cannot Undo changes made with this command.

This command can also be entered as "CHANGE_Object" or "CG Object".

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Type	= Member or Group	1	Member	%CGO.T	=Member
Object	= object	2	%PF.O	%CGO.O	=Any
Global	= global	3	%PF.G	%CGO.G	=Ask
coordinates				none	

Definition of Options

Type =

Member means the properties of the primitive type picked will be displayed for change. Member properties are Layer, Pen, Style and other properties that are unique to the type of primitive picked.

Group means the properties of the group picked will be displayed for change. Properties for 2-dimensional groups are Tag, Class, Priority, X and Y Location, X and Y Scale and Rotation Angle. For 3D groups its Tag, Class, Priority, and X,Y & Z Location.

Object =

See **Common Options** for more information.

Global =

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

cgo

You will be prompted to pick the object you wish to change. Upon selecting an object, it's member properties will be displayed in a dialog box. Make the desired changes and click the "OK" button. All done.

Circle or CC

2D

The Circle command creates circles. Circles can be defined (constructed) by Radius, Diameter or Circumference. The size of the circle can be specified by locating points or typing in a known value.

After entering the command, the desired options and a return, you will be prompted to locate one, two or three points, depending on your choice of construction option.

Each Undo (<%> key) will remove the last point located on the current circle. Each Undo past the current circle will remove one complete circle. The Undo buffer will remember the last 102 circles drawn during the current use of the command.

Circles can be gapped using the Gap Arc command. Gapping a circle converts it into an arc.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Construction	= Radius or Diameter or Circumference	1	Radius	%CC.C	=Radius
Size	= number	2	none	%CC.S	=1
CLass	= 0 thru 255	3	%WK.C	%WK.C	=0
Layer	= 0 thru 255	4	%WK.L	%WK.L	=0
Pen	= 0 thru 255	5	%WK.P	%WK.P	=0
STyle	= style	6	%WK.S	%WK.S	=Solid
PRiority	= 0 thru 255	7	%WK.PR	%WK.PR	=125
WORkset	= workset	8	%WK.W	%WK.W	=none
Mimic			none	%CC.M	
coordinates			none	none	

Definition of Options

Construction =

This option specifies how the circle will be constructed. Valid values are: Radius, Diameter and Circumference.

Radius will construct a circle by defining its radius. The radius is the linear distance between the center (origin) and a point on the circle. If the Size option is not used, two points are required to define each circle. They are the center point and a point on the circle. If the Size option is used, only the center point is required.

Diameter will construct a circle by defining its diameter. The diameter is the linear distance from one side of the circle to the other passing through the center. If the Size option is not used, two points are required to define each circle. They are located on opposite sides of the circle. If the Size option is used, only the center point is required.

Circumference will construct a circle by defining its circumference. The circumference is the radial distance around the circle. If the Size option is not used, three points are required to define each circle. If the Size option is used, only the center point is required.

Size =

This option sets the size of the circle. The Size value must be a number greater than zero. The size is either the Radius, Diameter or Circumference as set by the Construction option.

Class =

See **Common Options** for more information.

Layer =

See **Common Options** for more information.

Pen =

See **Common Options** for more information.

STyle =

See **Common Options** for more information.

PRiority =

See **Common Options** for more information.

WOrkset =

See **Common Options** for more information.

Mimic

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Circle

Circles will be drawn by defining two points. The distance between the first and second points determines the radius of the circle. The Class, Layer, Pen, and Style will all default to the values shown on the Status Palette at the bottom of your screen.

CC Construct=Diam Size=1'3" ST=Dot P=15 Wo=circles

Circles will be drawn with a diameter of one foot, three inches. You will be prompted to locate a center point for each circle. The circle will be dotted in style and drawn with pen fifteen. The Class, Layer and Priority will default to those defined in the workset "circles."

ci c 1 2 3 4 5 6

In this example the option values have been entered in strict positional order. Circles with the circumference of one will be drawn. Class 2, Layer 3, Pen 4, Style 5 (Phantom line) and Priority 6 will be used.

Class or CL

2D & 3D

The Class command sets the Class that you wish to work on. It also sets the Classes that are to be displayed and editable. The current work class is the class that new objects will be drawn on unless a different class is specified in the specific command for a new object. Any combination of Classes can be turned on or off for display or edit, with the restrictions that a class must be visible to be editable, and the current working class must be displayed.

There are 256 Classes on each drawing, numbered 0 through 255. Every object on your drawing has a Class assignment, just like it has a Layer, Pen, Style and Priority assignment. Classes differ from Layers in that Class is a property of a group, where Layer is a property of a primitive.

A primitive that has not yet been made part of a group is, for all intents and purposes, a group of only one item. Therefore, it has both a Layer and Class assigned to it. But, when several primitives are grouped together, they all assume the same Class. All members of a group have the same Class assignment because the Class is assigned to the entire group, not the individual members. If the group was later broken, the members would still retain the Class assignment they had when part of the group.

The Classes that contain data on your drawing are remembered in the Current Status Word %CL.U.n.

C Commands

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Work	= 0 thru 255	1	last	%CL.W	=0
Display	= classes or All or Display or Edit or Used	2	last	%CL.D.n	=All
Edit	= classes or All or Display or Edit or Used	3	last	%CL.E.n	=All

Definition of Options

Work =

Sets the current work Class that is displayed on the Status palette at the bottom of your screen. The work Class is the default class that new objects will be drawn on if the Class option is not used in a command that creates a primitive. You must enter a whole number from 0 through 255.

Display =

Sets the classes that are displayed. A class must be visible to be editable. If a class is made editable, it will automatically be made visible. Similarly, if a class is turned off for display, it will also be turned off for edit. Remember, though, that it is possible for a Class to be visible when it is not editable.

classes is a list of the classes that you wish to have turned on and off from the classes that are currently being displayed. There are 256 classes on each drawing, numbered 0 through 255. List the classes you wish to have turned on, separating each number with a space or comma. Ranges on classes can be entered by separating the starting class and ending class with an ellipsis (three periods, like this: 10...20). On Macintosh computers, the single ellipsis character can also be entered - it's Option-; on the keyboard. Individual classes and ranges of classes are turned off by preceding them with an up arrow (<Shift>-6 on most keyboards, like this: ^).

Here is an example how you would turn on classes 23, 43, 100 through 200 for display, and turn off classes 150 through 175 for display: "Class Display = 23 43 100...200 ^150...175".

All means that all classes will be turned on for display.

Display means the classes currently displayed will be displayed.

Edit means that just the classes that are currently editable will be displayed.

Used means that just the classes that contain data will be displayed.

Edit =

Sets the classes that are editable. Otherwise, the values for this option work exactly like the Display option.

Examples

CLASS

Executing the Class command this way will do nothing. All of the command options will default to last used, which will not change the Work, Display or Edit settings.

cl 4 a

Your work Class will be set to four and all Classes will be displayed.

Class Edit=None

All Classes, except the work Class, will be made not editable. The Classes being displayed will be unaffected.

cla,,,125, 200...220, 230...235, ^6, ^10...20

Classes 125, 200 through 220, and 230 through 235 will be made editable. Classes 6, and 10 through 20 will be made uneditable. The Work and Display Classes will be unchanged.

CLEar or CZ

2D & 3D

The Clear command removes the current drawing from your workarea. A new blank workarea is created with the same setup parameters as the cleared drawing. Drawing setup parameters include the drawing Width, Height, Scale, Work Units, Plot Units and Resolution. The settings of the TurboCAD graphics commands (last used values) will also be retained.

If the current drawing had been altered since the last save to disk, you will be asked if you wish to save the changes before clearing.

You cannot Undo the Clear command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Force				%CZ.F	

Definition of Options

Force

This keyword-only option forces the clearing of the existing drawing by bypassing the safety question. This option is most commonly used in Macros where operator intervention is not desired.

Warning: Use of this option can cause loss of data with no warning.

Examples

Clear

A new blank drawing workarea will be created with the same setup parameters as the cleared drawing. A safety question will appear if the current workarea has not been saved.

CZ F

A new blank drawing workarea will be created with the same setup parameters as the cleared drawing. Your drawing will be cleared without asking the safety question, so BE CAREFUL.

CLIP or CX

2D

The Clip command deletes whole or partial objects at the point that they cross an enclosed area perimeter. The enclosed area, called a "fence", can be in the shape of a rectangle, circle or irregular convex polygon. Objects are clipped (like a cookie cutter) where they intersect the fence and the contents inside or outside of the fence are deleted.

Undo (<%) key) will restore (unerase) the objects deleted by the last Clip fence. A second consecutive Undo will restore the objects deleted by all previous fences in this use of the command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Fence	= Rectangles or Circles or Polygons	1	last	%CX.F	=Rectangles
Select	= Touching or Pick	2	last	%CX.S	=Touching
Delete	= Inside or Outside	3	last	%CX.D	=Inside
Convert Frame coordinates	= switch	4	Yes none none	%CX.C %CX.FR	=Yes

Definition of Options

Fence =

This option defines the shape of the clip fence.

Rectangles means the clip fence will be defined by locating any two diagonally opposite corners of a rectangular area.

Circles means the clip fence is a circle defined by locating its center point and a point on the circle.

Polygons means an irregular convex polygon is the clip fence. The polygon can have from 3 through 15 sides. The polygon is created by locating points that are the vertices of the polygon. A line will rubberband from the last vertex located to the starting point of the polygon. Each subsequent vertex located must be between the last vertex and the starting point. The polygon cannot be concave (all internal angles must be less than 180°). An error will be issued if an illegal point is located.

Pressing the <E> key will terminate the polygon fence being defined and perform the clip.

Select =

This option determines how objects are to be selected.

Touching means all objects inside the clip fence or touching the fence will be clipped.

Pick means you will selectively pick the objects which you want clipped.

Delete =

This option determines how objects are to be clipped.

Inside means objects completely inside the fence, or the portions of objects touching the fence which fall inside the fence, will be deleted.

Outside means objects completely outside the fence, or the portion of objects touching the fence which fall outside the fence, will be deleted.

Convert =

This option determines if objects are to be automatically converted into Line primitives so that they may be clipped at the fence line. The Convert value must be a switch value. Switch values are: True or False, Yes or No, On or Off, All or None. A positive value means that objects will be converted, while a negative value means that they will not. In the case of a negative value, only the objects that can be clipped without being converted will be clipped.

C Commands

Frame

This keyword-only option, when used, will cause TurboCAD to draw the fence line as part of executing the command. If the Fence is rectangular, then a Rectangle primitive will be drawn. If the Fence is circular, then a Circle primitive will be drawn. If the Fence is polygonal, then a closed multi-segment line will be drawn. The Class, Layer, Pen, Style, and Priority of the fence will be that of the Status palette at the time of executing the command. Remember that keyword-only options must be entered in order to be active.

Examples

cx,c,t,i

You will be prompted to define a circle by locating its center point and a point on the circle. The circle will be used as the clip fence. All whole and partial objects inside the fence will be clipped at the fence and deleted. Objects will be converted to Lines as necessary to perform the clip.

cli fence=poly sel=touch

You will be prompted to define an irregular polygon by locating between 3 and 15 points. The polygon will be used as the clip fence. All whole objects inside, or objects partially inside, the fence will be deleted. Objects will be converted to Lines as necessary to perform the clip.

clip fence=rect sel=pick del=out Frame

You will be prompted to define the diagonally opposite corners of a rectangle. The rectangle(s) will be used as the clip fence. Objects you pick which are completely outside the fence or the portion of those you pick which are outside the fence will be deleted. Objects will be converted to Lines as necessary to perform the clip.

CLOse or CZ

2D & 3D

This command is a built-in alias for the "Clear" command. See the Clear command for information.

COLors or CS

2D & 3D

The Colors command allows you to assign a color to any pen or pens, the drawing background, status area background, text for dialogs and status area, crosshairs and grids. A color is defined by a name or a combination of Hue, Lightness, and Saturation. Macintosh computers will allow you to specify a color via the standard color "wheel" or palette.

You cannot Undo the Colors command, although it does have a Reset option to get the colors back to their startup values.

Note: Only the colors assigned to the first eleven pens of a drawing are remembered with each drawing. The colors assigned to pens 11 through 255 are set only during the current session. Because of this it is often a good idea to write a Macro that contains the color assignments you like to work with.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Item	= 0 thru 255 or Field or Dialog or Highlight or Small or Large	1	last	%CS.I	=Field
Color	= Blue or Purple or Magenta or Violet or Pink or Red or Orange or Yellow or Olive or Green or Aqua or Cyan or Turquoise or White or Gray or BRown or BLAck	2	none	%CS.C	=Blue
Hue	= 0 thru 360	3	last	%CS.H.n	=0
Lightness	= 0 thru 100	4	last	%CS.L.n	=50

C Commands

Saturation	= 0 thru 100	5	last	%CS.S.n	=100
Reset				%CS.R	=none
Noupdate				%CS.N	=none

Definition of Options

Item =

This option specifies the item whose color you are setting. Valid Item values are:

0 thru 255	Objects drawn in pens 0 through 255.
Field	The background color of your drawing.
Dialog	The text in the status area and dialogs.
Highlight	The background color of the status area.
Small	The small (minor) reference grids.
Large	The large (major) reference grids.

On Macintosh computers, the color of the menu bar and scroll bars will be the same as the Dialog or Highlight, whichever is darker.

Color =

This option sets the color of the item specified. Valid Color values are the color names shown above. These color names are nothing more than predefined Hue, Lightness and Saturation values provided to make color selection a little easier. Color names can be mixed with the Hue, Lightness and Saturation options to modify a color. For example, to get a light blue field you might enter "Color Field Blue Lightness=80".

See the "Color Names" table and "Startup Colors" table shown below for the HLS assignments of a color.

Hue =

This option sets the hue of the color. Hue is a graduation of color advanced by degrees, represented as an angle from 0 through 360 degrees. The angle values of the primary and secondary colors are:

Hue	Color
0	Blue
60	Magenta
120	Red
180	Yellow
240	Green
300	Cyan

Lightness =

This option sets the lightness of the color. Lightness describes the amount of black or white in a color. Lightness is expressed as a percentage with 0 being black and 100 being white.

Saturation =

This option sets the saturation of the color. Saturation describes the extent to which a color differs from a gray of the same lightness. Saturation is expressed as a percentage with 0 being maximum white content and 100 being full color saturation.

Reset

This option sets all color values back to the startup colors. See the "Startup Colors" table shown below.

Noupdate

Because some monitors require a time consuming repaint of the screen with each color change, we have included this option so you can suppress this repaint. This is especially useful in Macros where multiple color changes are being made and you wish to avoid a lot of repaint time.

Color Names

<u>Color Name</u>	<u>Hue</u>	<u>Lightness</u>	<u>Saturation</u>
Blue	0	50	100
Purple	35	70	80
Magenta	60	50	100
Violet	70	40	100
PInk	80	50	100
Red	120	50	100
Orange	150	50	100
Yellow	180	50	100
Olive	210	50	35
Green	240	50	100
Aqua	270	50	100
Cyan	300	50	100
Turquoise	320	60	100
White	0	85	0
Gray	0	55	0
Brown	160	35	100
BLAck	0	0	0

C Commands

Startup Colors

<u>Item</u>	<u>Hue</u>	<u>Lightness</u>	<u>Saturation</u>
Field	0	40	100
Dialog	35	95	0
Highlight	0	25	75
Small	0	30	0
Large	0	45	0
Pen 0	0	85	0
Pen 1	120	50	100
Pen 2	300	50	100
Pen 3	180	50	100
Pen 4	60	50	100
Pen 5	240	50	100
Pen 6	320	60	100
Pen 7	70	40	100
Pen 8	210	50	35
Pen 9	35	70	80
Pen 10 thru 255	150	50	100

Examples

Color 5 Green

The color of pen five will be set to green.

COL,,RED,L=25

The color of the last item will be set to red with a lightness of 25 (dark red).

cs h,,0,30,70

The color of the Highlight (the background color of the command area) will be set to a Hue of 0, a Lightness of 30 and a Saturation of 70. This HLS scheme would produce a light blue color.

cs 1 red no; cs 2 blue

The color of pen one will be set to red. No screen repaint would occur. The semi-colon acts like a carriage return. It is used when you wish to separate multiple TurboCAD commands on a single line. The second command will set the color of pen two to blue.

COOrdinate or CD**2D & 3D**

The Coordinate command allows points to be placed on the TurboCAD point stack where they are remembered for later use. These points can be located with the crosshairs or their coordinates entered through the keyboard. The contents of the point stack can also be cleared (emptied) with this command.

The TurboCAD "point stack" is a buffer for coordinates. All points that are processed by TurboCAD pass through the point stack. Upon execution of a graphics command that requires point input, the contents of the point stack will be used first before allowing points to be entered with the crosshairs or from the keyboard.

A maximum of 256 points can be placed on the point stack. All distance and angle values may be positive, negative or zero. If a distance or angle is zero, the zero need not be entered. If a value is omitted, zero is used. However, delimiters must be used to assure that the entered values are assigned to their proper location in the coordinate string.

Undo (the <%> key) when used in a coordinate, will place an Undo on the stack. Example: [,,,%].

The Status command can be used to look up the contents of the point stack. For a description of how coordinate sets are formatted, see the Entering Coordinates section in Basics.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Active	= switch	1	none	%CD.A	=Yes
Cursor			none	%CD.C	
CLear			none	%CD.CL	
coordinates			none		

Definition of Options**Active =**

This option determines if points are to be read from the point stack. Points can be placed on the stack while the stack is inactive, but they will not be used until the stack is reactivated. The Active value must be a switch value. Switch values are: True or False, Yes or No, On or Off, All or None.

Cursor

This keyword-only option allows points to be placed on the point stack by locating points on your drawing with the crosshairs. Points placed on the stack in this manner will always be absolute coordinates. Snapping to an object (by using the <V> (vertex) key, for example) will

C Commands

record the absolute location of the object snapped to; not the location of the crosshairs when the key was pressed.

CLear

This keyword-only option empties the contents of the point stack. Any points currently on the point stack will be thrown away.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Coordinate [A,2,2|R,1|R,,1|;]

Four points will be placed on the point stack. Running the Status command to reveal the settings of the Coordinate command (Status Coordinate) would show the contents of the point stack to be:

[a,2,2,0,]

[r,1,0,0,]

[r,0,1,0,]

[r,0,0,0,;]

Note that the last point on the stack is equivalent to a carriage return on the command line. That is, it has no relative distances specified. Running the Line command would create a line starting two units to the right and two units up from the drawing origin, then draw one unit long to the right and one unit up. The carriage return on the stack would then be read, terminating the Line command.

CD N CL C

The point stack will be purged and you will then be prompted to locate points to be remember on the point stack. All located points will be remember as absolute coordinates. The point stack has also been made inactive. Therefore, the points that you place on the stack will not be used until the stack is reactivated.

Copy or CP

2D & 3D

The Copy command duplicates objects on your drawing. The objects to be copied can be selected by picking them or by defining the diagonally opposite corners of a box. The resulting location for the copies is determined by dragging the object, entering relative X, Y, and Z distances, locating two points to define the distance and angle (vector), or entering a distance and angle. Multiple copies can be made at a specified incremental distance, or an overall distance divided by the number of copies desired.

Each Undo (<%) key) will remove the last object or box of objects copied. The Undo buffer will remember the last 128 objects copied with this use of the command.

The <E> (end) key will terminate the selection of the current item that is being copied and allow a new one to be selected.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Xdistance	= number	1	none	%CP.X	=0
Ydistance	= number	2	none	%CP.Y	=0
Zdistance	= number	3	none	%CP.Z	=0
Repeat	= 0 thru 127	4	none	%CP.R	=1
Select	= select	5	%PF.S	%CP.S	=Pick
Mode	= Relative or Overall	6	Relative	%CP.M	=Relative
Object	= object	7	%PF.O	%CP.O	=Any
Vector			none	%CP.V	
Distance	= number	8	none	%CP.D	=0
Angle	= angle	9	none	%CP.A	=0
coordinates			none		

Definition of Options

Xdistance =

Sets the distance that objects will be copied along the X axis (horizontally). A positive X axis value is to the right, while a negative value is to the left. If multiple copies are made, this distance is the distance between each copy or the overall distance across all copies as set by the Mode option. Entering a value for this option will cause an error to be issued if you are using the Vector option or the Distance and Angle options below.

Ydistance =

Sets the distance that objects will be copied along the Y axis (vertically). A positive Y axis value is up, while a negative value is down. If multiple copies are made, this distance is the distance between each copy or the overall distance across all copies as set by the Mode option. Entering a value for this option will cause an error to be issued if you are using the Vector option or the Distance and Angle options below.

Zdistance =

Sets the distance that objects will be copied along the Z axis of the Viewplane. A positive Z axis value is toward you, while a negative value is away from you. If multiple copies are made, this distance is the distance between each copy. Entering a value for this option will cause an error

C Commands

to be issued if you are using the Vector or the Distance and Angle options below. This value will be ignored if you select objects which are 2D objects and will affect only 3D wireframes.

Repeat =

This option specifies the number of copies that will be made. A valid Repeat value is a whole number from 0 through 127. A Repeat value of zero is the same as 1 when a distance is entered and the same as 127 when dragging.

Select =

See **Common Options** for more information.

Mode =

This option determines how repeated copies are to be spaced.

Relative means that the copies will be spaced apart by the distance specified by the X, Y, and Z values entered, by the Vector defined, or by the Distance and Angle values entered.

Overall means that the distance specified as above will be divided by the number of copies specified.

Object =

See **Common Options** for more information.

Vector

This keyword-only option allows the distance and angle of the copies to be specified by locating two points with the crosshairs. If multiple copies are made, this distance is the distance between each copy or the overall distance across all copies as set by the Mode option. An error will be issued if this option is used at the same time as the X, Y, or Z distance options or the Distance and Angle options below.

Distance =

This option, along with the Angle option below, allows you to specify the distance vector numerically. Any positive or negative distance value is valid. If multiple copies are made, this distance is the distance between each copy or the overall distance across all copies as set by the Mode option. An error will be issued if this option is used at the same time as the X, Y, or Z distance options or the Vector option.

Angle =

This option works with the Distance option to specify the distance vector numerically. An error will be issued if this option is used at the same time as the X, Y, or Z distance options or the Vector option. See **Common Options** for more information on entering angles.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

C

You will be prompted to pick an object you wish to copy and drag it to the desired new location. Multiple copies of the object can be made.

CP R=4 V

You will first be prompted to locate two points to define the distance and angle between each copy. You will then be prompted to pick the objects you wish to copy. Four copies will be made of each picked object.

CP 10,10 D=10 A=45

You will get an error message because X and Y distances have been entered at the same time as specifying a vector by distance and angle.

CP 40,,,10,,0

You will be prompted to pick objects you wish to copy. Each picked object will be copied ten times to the right (along the X-axis) with four units between each copy (40 units divided by 10 copies).

copy y=2.5 s=all

All objects on your drawing will be copied up two and one half units.

Copy Member or CPM

2D & 3D

The Copy Member command duplicates group members on your drawing. This allows you to copy part of a group without breaking up the group. Once a group member is copied, it is a new object on your drawing. It is not part of the original group from which it was copied. A group member can only be selected by picking it. All other options are the same as in the Copy command.

Each Undo (<%> key) will remove the last member copied. The Undo buffer will remember the last 128 objects copied with this use of the command.

The <E> (end) key will terminate the selection of the current item that is being copied and allow a new one to be selected.

This command can also be entered as "COPY_Member" or "CP Member".

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Xdistance	= number	1	none	%CPM.X	=0
Ydistance	= number	2	none	%CPM.Y	=0

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Repeat	= 0 thru 127	3	none	%CPM.R	=1
Mode	= Relative or Overall	4	Relative	%CPM.M	=Relative
Object	= object	5	%PF.0	%CPM.O	=Any
Vector			none	%CPM.V	
Distance	= number	6	none	%CPM.D	=0
Angle	= angle	7	none	%CPM.A	=0
coordinates			none		

Definition of Options

Xdistance =

Sets the distance that objects will be copied along the X axis (horizontally). A positive X axis value is to the right, while a negative value is to the left. If multiple copies are made, this distance is the distance between each copy or the overall distance across all copies as set by the Mode option. Entering a value for this option will cause an error to be issued if you are using the Vector option or the Distance and Angle options below.

Ydistance =

Sets the distance that objects will be copied along the Y axis (vertically). A positive Y axis value is up, while a negative value is down. If multiple copies are made, this distance is the distance between each copy or the overall distance across all copies as set by the Mode option. Entering a value for this option will cause an error to be issued if you are using the Vector option or the Distance and Angle options below.

Repeat =

This option specifies the number of copies that will be made. A valid Repeat value is a whole number from 0 through 127. A Repeat value of zero is the same as 1 when a distance is entered and the same as 127 when dragging.

Mode =

This option determines how repeated copies are to be spaced.

Relative means that the copies will be spaced apart by the distance specified by the X and Y values entered, by the Vector defined, or by the Distance and Angle values entered.

Overall means that the distance specified as above will be divided by the number of copies specified.

Object =

See **Common Options** for more information.

Vector

This keyword-only option allows the distance and angle of the copies to be specified by locating two points with the crosshairs. If multiple copies are made, this distance is the distance between each copy or the overall distance across all copies as set by the Mode option.

An error will be issued if this option is used at the same time as the X or Y distance options or the Distance and Angle options below.

Distance =

This option, along with the Angle option below, allow you to specify the distance vector numerically. Any positive or negative distance value is valid. If multiple copies are made, this distance is the distance between each copy or the overall distance across all copies as set by the Mode option. An error will be issued if this option is used at the same time as the X or Y distance options or the Vector option.

Angle =

This option works with the Distance option to specify the distance vector numerically. An error will be issued if this option is used at the same time as the X or Y distance options or the Vector option. See **Common Options** for more information on entering angles.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples**Copy Member**

You will be prompted to pick an group member you wish to copy and drag it to the desired new location. Multiple copies of the member can be made.

CPM R=4 V

You will first be prompted to locate two points to define the distance and angle between each copy. You will then be prompted to pick the members you wish to copy. Four copies will be made of each picked member.

COPY_M 40,,10,0

You will be prompted to pick group members you wish to copy. Each picked member will be copied ten times to the right with four units between each copy (40 units divided by 10 copies).

Copy Parallel or CPP**2D**

The Copy Parallel command duplicates the shape of objects such that all segments are parallel to, and equidistant from, the original object. All primitives except Text, Smart Dimensions, and Pattern Fills (Boundaries) can be copied in parallel. The resulting primitive will be of the same type as the original. Objects are selected by picking them. Primitives picked which are part of a group will be copied in parallel as primitives and the resulting primitive will not be associated with the group. The copy distance can be specified by locating a point or entering a distance.

C Commands

Each Undo (<%> key) will remove the last object copied. The Undo buffer will remember the last 128 objects copied with this use of the command.

This command can also be entered as "COPY_Parallel" or "CP Parallel".

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Distance	= number or Points (or Pts)	1	Points	%CPP.D	=1
Repeat	= 0 thru 127	2	1	%CPP.R	=1
Mode	= Relative or Overall	3	Relative	%CPP.M	=Relative
coordinates			none		

Definition of Options

Distance =

Sets the distance that the parallel copy will be from the original. A valid Distance value is a number or the words "Points" or "Pts". If multiple copies are made, the distance is the distance between each copy or the overall distance across all copies as set by the Mode option.

A positive number value copies to the right of the direction the object was originally created, while a negative value copies to the left.

Points or **Pts** means that after picking the object to be copied you will be prompted to locate a point to define the distance the copy is to be from the original object. The distance is measured perpendicularly from the object to the point.

Repeat =

This option specifies the number of copies that will be made. A valid Repeat value is a whole number from 0 through 127. A Repeat value of zero is the same as 1.

Mode =

This option determines how repeated copies are to be spaced.

Relative means that the copies will be spaced apart by the distance specified in the Distance option.

Overall means that the distance specified will be divided by the number of copies specified.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Copy Parallel

You will first be prompted to pick an object you wish to copy and then to locate a point to define the distance of the copy from the original.

CPP D=1 R=4

You will be prompted to pick objects you wish to copy. Four parallel copies will be made of each picked object with one unit of distance between each copy.

C P 40 10 0

You will be prompted to pick objects you wish to copy. Each picked object will be copied ten times to the right with four units between each copy (40 units divided by 10 copies).

Copy Radial or CPR

2D

The Copy Radial command duplicates objects on your drawing radially about a point. The angle of rotation each copy can be determined by locating points or typing in an angle value. You have the option of having the copies rotated or in their original orientation. Like the other Copy commands, multiple copies can be made either at an incremental angle value or an overall angle value.

After entering the command and options, you will be prompted to locate a pivot point. This is the point about which objects will be copied radially and it can be any point on your drawing. You will then be prompted to select the object to be copied. A temporary reference line will appear between the pivot point and the point picked on the object. When you select objects by box, the reference line will be between the pivot point and the origin of the object in the box. If you did not enter a specific angle value, you will be prompted to locate a point which will define an angle between the object selected and the first copy.

Each Undo (<%> key) will remove the last object or box of objects copied. The Undo buffer will remember the last 128 objects copied with this use of the command.

The <E> (end) key will terminate the selection of the current item that is being copied and allow a new one to be selected.

This command can also be entered as "COPY_Radial" or "CP Radial".

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Angle	= angle or Points (or Pts)	1	Points	%CPR.A	=0

C Commands

Repeat	= 0 thru 127	2	1	%CPR.R	=1
Select	= select	3	%PF.S	%CPR.S	=Pick
Mode	= Relative or Overall	4	Relative	%CPR.M	Relative
Object	= object	5	%PF.O	%CPR.O	=Any
ROtate coordinates	= switch	6	Yes none	%CPR.RO	=Yes

Definition of Options

Angle =

This option sets the angle between the object selected, the pivot point, and the first radial copy. A valid Angle value is any number between -360 and 360, or the words "Points" or "Pts. " If multiple copies are made, this angle is the angle between each copy or the overall angle across all copies as set by the Mode option. See **Common Options** for more information on entering angles.

Points or **Pts** means that you will be required to locate an additional point to define the angle after you have picked the object to be copied.

Repeat =

This option specifies the number of copies that will be made. A valid Repeat value is a whole number from 0 through 127. A Repeat value of zero is the same as 1.

Mode =

This option determines how repeated copies are to be spaced.

Relative means that the copies will be spaced apart by the angle specified in the Angle option.

Overall means that the angle specified will be divided by the number of copies specified.

Object =

See **Common Options** for more information.

ROtate =

This option determines if objects should be rotated about their individual origins as they are being copied radially or if they should remain in their original orientation. The Rotate value must be a switch value. Switch values are: True or False, Yes or No, On or Off, All or None.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Copy Radial

You will first be prompted to locate a pivot point about which objects are to be copied radially. You will then be prompted to pick an object. Because no angle value was entered, you will be

required to locate an additional point to define the copy angle. You can pick multiple items for radial copying at this angle about this pivot point. If you would like to copy the same object repeatedly, here is a great use for the <-> key (last picked object). Press the <-> key once for each additional copy you would like.

CP R R=4 A=45

You will first be prompted to locate a pivot point about which objects are to be copied radially. You will then be prompted to pick an object. The object will be radially copied four times counterclockwise with an angle of 45 degrees between each copy.

cpr -90,,in

After locating the pivot point you will be prompted to define a box by locating its diagonally opposite corners. All objects completely inside the box will be copied once ninety degrees clockwise.

COUnt or CT

The Count command searches your drawing for copies of a given object, counts them, and displays the total. The object is selected for counting by picking it or entering its tag. See the TAG and LIST commands for associated information.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Tag	= string	1	none	%CT.T	= ``
Classes	= classes	2	All	%CT.C.n	= All
coordinates			none		

Definition of Options**Tag =**

Specifies the "Tag" which is to be searched for in the drawing. All objects having the Tag specified will be counted. The *string* is any alphanumeric string up to 10 characters long and is case sensitive. If the string contains spaces or special characters it must be quoted ('like this' or \like this\). If this option is not used, you will be prompted to pick an object on your drawing that you want counted.

Classes =

Sets the classes that will be searched if a Tag is specified. Valid Classes values are:

classes is a list of the classes that you wish to have searched. There are 256 classes on each drawing, numbered 0 through 255. List the classes separating each number with a space or

C Commands

comma. Ranges of classes can be entered by separating the starting class and ending class with an ellipsis (three periods, like this: 10...20). Individual classes and ranges of classes are turned off by preceding them with an up arrow (shift-6 on the keyboard, like this: ^). The up arrow (^) is the TurboCAD calculator symbol for a logical "not" function. See **Drawing Structure** and **Common Options** for more information.

All means all classes will be searched.

Display means only the classes that are currently displayed will be searched.

Edit means only the classes that are currently editable will be searched.

Used means only the classes that contain data will be searched.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Count

The total count will be displayed for the object on your drawing that you pick.

COU TAG=\BIG CHAIR\

The total count of all groups tagged "BIG CHAIR" on all classes of your drawing will be displayed on the screen.

ct,door,40...100

The total count of all groups tagged "door" on classes forty through one hundred of your drawing will be displayed on the screen.

CRosshatch or CR

2D

The Crosshatch command fills enclosed areas with predefined line patterns. Once created, a crosshatch pattern is actually a Line primitive and may be edited as such. Because of this, Crosshatch patterns are suitable for pen plotting, where as "Bitmapped" patterns are not (see BOUNDARY command). Because line primitives have a maximum number of segments, several primitives may be necessary to create the pattern. If this is the case, they will be automatically grouped so they may be edited as a unit.

Enclosed areas are created by locating points or picking existing primitives or group members on your drawing. Whole groups cannot be used as the crosshatch enclosure. Existing objects are selected by picking them with the key. On Macintosh computers you can hold down the <Shift> key while picking with the mouse button (<Shift>-click). A temporary line will represent the enclosure as it is being created and a line will rubberband to the crosshairs so you

can see the extent of the enclosure thus far. If many objects are picked they will be connected with straight segments in the order they were picked.

Enclosed areas can overlap and be nested (one inside another). Pressing the <K> (close) key will close the current enclosure without crosshatching it immediately. Several nested or overlapping enclosures may be created this way before they are crosshatched. Nested enclosures will alternate between being crosshatched and not crosshatched starting with the outermost enclosure. A maximum of 256 vectors can make up an enclosure.

Each Undo (<%> key) will remove the last point located or object picked to form the enclosed area. The Undo buffer will remember the last 256 points located and/or objects picked with this use of the command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Pattern	= name	1	last	%CR.P	=`Lines`
Angle	= angle	2	last	%CR.A	=0
Scale	= number	3	last	%CR.S	=1
Origin	= Boundary or Point (or Pt)	4	Boundary	%CR.O	=Boundary
Class	= 0 thru 255	5	current	%WK.C	=0
Layer	= 0 thru 255	6	current	%WK.L	=0
PEn	= 0 thru 255	7	current	%WK.P	=0
STyle	= style	8	current	%WK.S	=Solid
PRiority	= 0 thru 255	9	current	%WK.PR	=125
WOrkset	= workset	10	current	%WK.W	=none
Frame			none	%CR.FR	
Select			none	%CR.SE	
Mimic			none	%CR.M	
coordinates			none		

Definition of Options

Pattern

Selects a predefined TurboCAD crosshatch pattern. Pattern names must be spelled out in full. The available patterns are:

ASTERISKS	EARTH	SPANISHTILE
BATTLEMENT	FIREBRICK	SQUARES

C Commands

BRICK	GRATE	SQUAREGRID
CEILINGGRID	HERRINGBONE	STARS
CHECKERPLATE	HONEYCOMB	STEEL
CHICKENWIRE	LINES	TRIANGLEGRID
CONCRETE	OCTAGONS	ZIGZAG
CROSSES	SOLIDFILL	

See "Crosshatch Patterns" at the end of this command for graphic examples of these crosshatch patterns.

Angle =

This option sets the rotation angle to be applied to crosshatch pattern. A valid Angle value is any number between -360 and 360. See the **Common Options** section for more information.

Scale =

This option sets the scale of the pattern. A scale of "1" means the scale at which the pattern was originally defined. The Scale value must be a number greater than zero. Since your choice of Work Units and Drawing Scale will affect the spacing of crosshatch patterns, it is a good idea to crosshatch a small area first in order to arrive at an acceptable scale value.

Origin =

This option sets the starting point of the pattern. This option is most commonly used to control the starting point of patterns so that patterns created at different times will align.

Boundary means that the first point that is located to define the enclosed area to be crosshatched will also be used as the origin of the pattern.

Point (or **Pt**) means that you will be required to locate a point to be used as the origin before defining the enclosure.

Class =

See **Common Options** for more information.

Layer =

See **Common Options** for more information.

Pen =

See **Common Options** for more information.

STyle =

See **Common Options** for more information.

PRiority =

See **Common Options** for more information.

WOrkset =

See **Common Options** for more information.

Frame

This keyword-only option, when used, will cause a line primitive to be created at the boundary you defined for crosshatching. This line will be independent from the lines in the pattern. In order for this option not to be confused with a pattern name, you must always enter it after a valid pattern name (i.e. "cr lines frame" or "cr,, frame").

Select

This keyword-only option, when used, will provide you with a list of available crosshatch patterns on the screen from which to choose. This option is valid only on UNIX systems. In order for this option not to be confused with a pattern name, you must always enter it after a valid pattern name.

Mimic

This keyword-only option, when used, will cause TurboCAD to prompt you to pick a line on the drawing before you define the boundary. The Class, Layer, Pen, Style and Priority of the line selected will be used in the crosshatch pattern. It will not mimic an existing pattern. In order for this option not to be confused with a pattern name, you must always enter it after a valid pattern name (i.e. "cr lines mimic" or "cr,, mimic"). See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples**Crosshatch Brick**

You will be prompted to locate points and/or pick objects on your drawing to define an enclosed area that will be filled with the "Brick" crosshatch pattern. The pattern will be full scale at an angle of zero. The Class, Layer, Pen, and Style options will default to the values shown on the Status Palette at the bottom of your screen. The Origin option will default to Boundary.

cr grate 90 2 p

You will first be prompted to locate a point for the pattern origin and then to locate points and/or pick object on your drawing to define an enclosed area that will be filled with the "Grate" crosshatch pattern. The pattern will be rotated ninety degrees counterclockwise and twice its normal scale. The Class, Layer, Pen, and Style options will default to the values shown on the Status Palette at the bottom of your screen.

Cross,Crosses Pe=45 ST=do

You will be prompted to locate points and/or pick object on your drawing to define an enclosed area that will be filled with the "Crosses" crosshatch pattern. The pattern will be full scale at an angle of zero. The pattern will be drawn in pen 45 and Dotted line style. The Class, Layer and Priority options will default to the values shown on the Status Palette at the bottom of your screen. The Origin option will default to Boundary.

C Commands

Cr,, o=p mi [a,0,0]

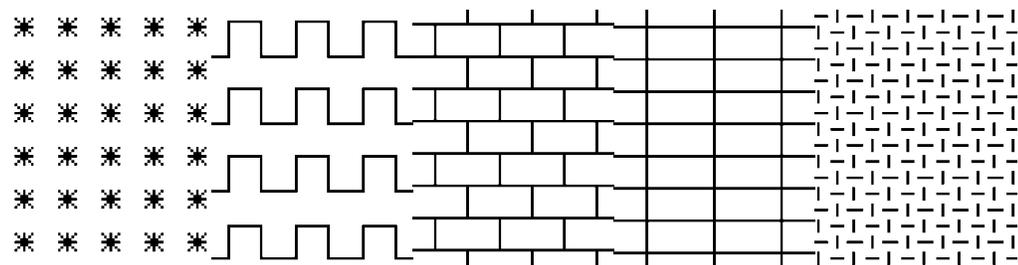
You will be prompted to pick an object on your drawing. The Class, Layer, Pen, Style, and Priority values of the object picked will be used for the pattern. The pattern will default to the last used pattern. The absolute coordinate "0,0" (your drawing origin) will be used as the origin for the pattern.

Cr,, o=p wo=`my patterns` [a,0,0]

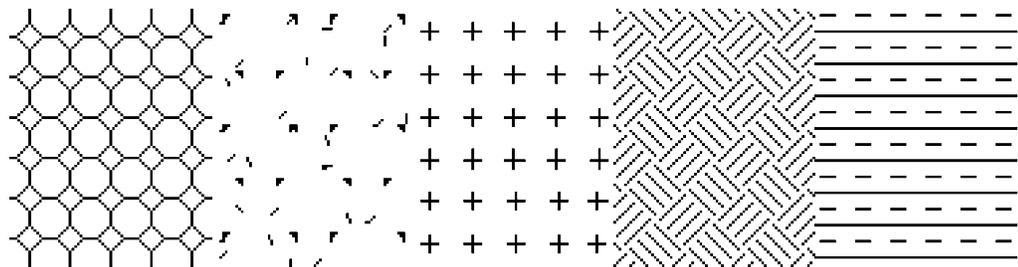
This is the same as the above example except that the Class, Layer, Pen, Style, and Priority values used will be those of the Workset (my patterns) which you had previously defined.

Crosshatch Patterns

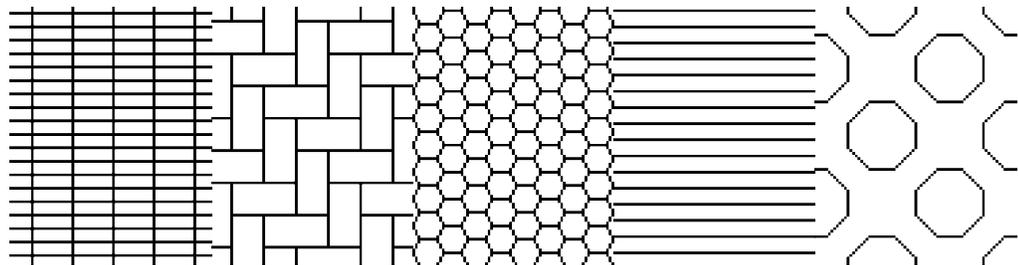
Shown below are graphic examples of the provided TurboCAD crosshatch patterns.



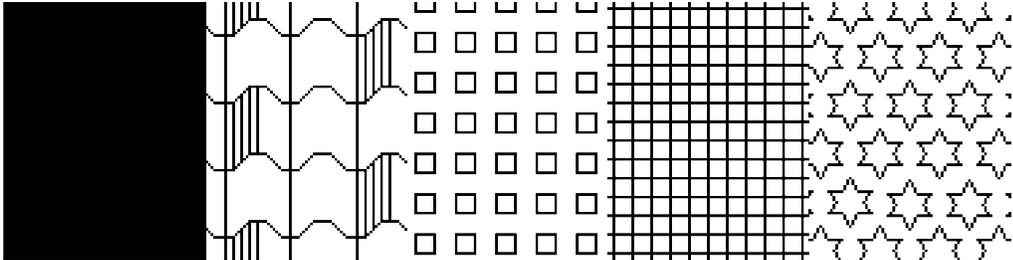
Asterisks Battlements Bricks Ceiling Grid Checkerblatee



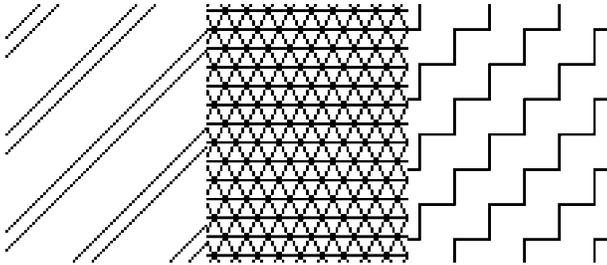
ChickenWire Concrete Crosses Earth Firebrick



Grate Herringbone Honeycomb Lines Octagons



Solidfill Spanish Tile Squares Square Grid Stars



Steel Triangle Grid Zig-Zag

Delete or DL

2D & 3D

The Delete command removes graphic objects from your drawing. Only complete objects (primitives and groups) can be deleted with this command. See one of the many other Delete commands for deleting group members, line segments, vertices, etc.

Objects can be selected for deletion by picking them with the crosshairs or defining a box. You can pick anywhere on an object to delete it. If deleting by box, you can also delete objects not included in the box by shift-clicking on them. Specific types of objects can be specified for deletion (i.e. "filtered"), if desired.

Each Undo (<%> key) will restore (unerase) the last object or box of objects selected. The Undo buffer will remember the last 256 objects deleted with this use of the command.

Options

Keyword	Value	Order	Default	CSW	Startup
Select	= select	1	%PF.S	%DL.S	=Pick
Object	= object	2	%PF.O	%DL.O	=Any
coordinates			none		

Definition of Options

Select =

See **Common Options** for more information.

Object =

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

D

The command options will default to the settings of the Select and Object options of the Preferences command. You will be prompted to pick the objects you wish to have deleted.

DE IN

All objects that are completely inside a rectangular area are deleted. You define the rectangular area by locating two points that are its diagonally opposite corners.

dl s=o o=t [||;]

All text that is not part of a group will be deleted from your drawing. The command is being executed with Select=Outside and Object=Text. Three coordinates have been entered on the

command line inside of the square brackets. The coordinates are separated with a vertical bar (|). The first two coordinates have all of their options defaulted which enters a point at the last point located on your drawing. This creates a box that essentially has no size, making all text on the drawing qualify for deletion. The last coordinate is a carriage return (;) which will terminate the command.

dl a

All objects on your drawing will be deleted. If it is a drawing with a large number of objects, you will be given a message that the "Undo" buffer is full and if you wish to continue. If you answer "Yes" then the Undo function will be inoperative.

Delete Edge or DL E**3D**

The Delete Edge command removes edges (segments) from 3-dimensional Wireframes. Edges are like 3-dimensional line segments. Edges are to a Wireframe primitive what segments are to a Line primitive. When you delete an edge, the two nodes (3-dimensional vertices) that defined the endpoints of the edge will join to become the same node. Deleting an edge is the 3D equivalent to deleting a 2D segment.

Each Undo (<%> key) will restore (unerase) the last edge deleted. The Undo buffer will remember the last 256 edges deleted with this use of the command.

This command can also be entered as "DELETE_Edge" or "DL Edge".

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Global	= global	1	%PF.G	%DLE.G	=Ask
coordinates			none		

Definition of Options

Global =

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples**Delete Edge**

You will be prompted to pick the edges you wish to have deleted. If a picked group has copies, the appropriate global change will be made based on the current setting of the Global option of the Preferences command.

D E All

You will be prompted to pick the edges you wish to have deleted. If a picked group has copies, they will all be automatically changed as well.

Delete Gap or DLG

The Delete Gap command removes gaps (invisible segments) from Lines, Doublelines and Leaderlines. Gaps are created in these primitive types with the GAP command. The CLIP command can also produce gaps. For purposes of this command, assume that a Gap is merely a segment in a multi-segment line which is invisible. Two vertices define the start and end of the Gap. Deleting a Gap is different than deleting a segment with the DELETE SEGMENT command. Deleting a Gap does not remove the Gap segment from the line. It just makes it visible.

To delete a gap you will be required to pick the nearest visible segment. Deleting a gap simply changes the gap from an invisible segment to a visible one.

Each Undo (<%> key) will restore (undelete) the last gap deleted. The Undo buffer will remember the last 256 gaps deleted with this use of the command.

This command can also be entered as "DELETE_Gap" or "DL Gap".

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Global	= global	1	%PF.G	%DLG.G	=Ask
coordinates			none		

Definition of Options

Global =

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Delete_g

You will be prompted to pick a segment near the gap you wish to have deleted. If a picked group has copies, the appropriate global change will be made based on the current setting of the Global option of the Preferences command.

DLG ON

You will be prompted to pick a segment near the gap you wish to have deleted. Only the picked group will be affected.

DLG all

You will be prompted to pick a segment near the gap you wish to have deleted. If you picked a gap in a line which is part of a copied group, all copies will have the gap deleted.

Delete GAP3 or DL3**3D**

The Delete Gap3 command removes gaps (invisible segments) from 3-dimensional Wireframes. Gaps are created in wireframes with the GAP EDGE command. For purposes of this command, assume that a Gap is merely an edge in a wireframe which is invisible. An edge in a wireframe is the equivalent of a segment in a 2D line. Two nodes define the start and end of the Gap. Deleting a Gap is different than deleting an edge with the DELETE EDGE command. Deleting a Gap does not remove the gapped edge from the wireframe. It just makes it visible.

To delete a gap you will be required to pick the nearest visible segment. Deleting a gap simply changes the gap from an invisible segment to a visible one.

Each Undo (<%> key) will restore (undelete) the last gap deleted. The Undo buffer will remember the last 256 gaps deleted with this use of the command.

This command can also be entered as "DELETE_GAP3" or "DL GAP3".

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Global	= global	1	%PF.G	%DL3.G	=Ask
coordinates			none		

Definition of Options

Global =

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

D Commands

Examples

Delete Gap3

You will be prompted to pick a wireframe near the gap you wish to have deleted. If a picked group has copies, the appropriate global change will be made based on the current setting of the Global option of the Preferences command.

dl3,o

You will be prompted to pick a wireframe near the gap you wish to have deleted. Only the picked group will be affected.

DL3 all

You will be prompted to pick an edge on a wireframe near the gap you wish to have deleted. If you picked a gap in a wireframe which is part of a copied group, all copies will have the gap deleted .

Delete Member or DLM

2D & 3D

The Delete Member command removes members from Groups. This command will only delete graphic primitives that are part of a group, not complete sub-groups (called group tiers). Group members are deleted by picking them with the crosshairs. Group members can be deleted globally from copied groups depending on the settings of the Global option.

Each Undo (<%> key) will restore (undelete) the last member deleted. The Undo buffer will remember the last 256 members deleted with this use of the command.

This command can also be entered as "DELETE_Member" or "DL Member".

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Global	= global	1	%PF.G	%DLM.G	=Ask
coordinates			none		

Definition of Options

Global =

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Delete Mem

You will be prompted to pick a member of a group you wish to have deleted. If a picked group has copies, the appropriate global change will be made based on the current setting of the Global option of the Preferences command.

de m g=some

You will be prompted to pick a member of a group you wish to have deleted. If the picked group has copies on your drawing, they will be highlighted and you will be prompted to pick the copies you wish to have changed.

Delete Node or DLN

3D

The Delete Node command removes nodes (3-dimensional vertices - endpoints of edges) from Wireframes. Nodes are deleted by picking them with the crosshairs, much in the same way that 2D "vertices" are deleted via the Delete Vertex command. When a node is deleted, the two edges which met at the node become one single edge. Nodes can be deleted globally from copied groups depending on the settings of the Global option.

Each Undo (<%> key) will restore (undelete) the last node deleted. The Undo buffer will remember the last 256 nodes deleted with this use of the command.

This command can also be entered as "DELETE_Node" or "DL Node".

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Global	= global	1	%PF.G	%DLN.G	=Ask
coordinates			none		

Definition of Options

Global =

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Del Node

You will be prompted to pick a node you wish to have deleted. If a picked group has copies, the appropriate global change will be made based on the current setting of the Global option of the Preferences command.

de no so

You will be prompted to pick a node you wish to have deleted. If the picked group has copies on your drawing, they will be highlighted and you will be prompted to pick the copies you wish to have changed.

Delete Segment or DLS

3D

The Delete Segment command removes visible segments from Lines and Doublelines. Segments are deleted by picking them with the crosshairs. If the object picked is not a Line or Doubleline, you will be asked if you would like it converted into a Line so the edit can be accomplished. When a Segment is deleted, its original endpoints meet to form a single vertex. Segments can be deleted globally from copied groups depending on the settings of the Global option.

Each Undo (<%) key) will restore (undelete) the last segment deleted. The Undo buffer will remember the last 256 segments deleted with this use of the command.

This command can also be entered as "DELETE_Segment" or "DL Segment".

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Global	= global	1	%PF.G	%DLS.G	=Ask
coordinates			none		

Definition of Options

Global =

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples**DL S**

You will be prompted to pick a segment you wish to have deleted. If a picked group has copies, the appropriate global change will be made based on the current setting of the Global option of the Preferences command.

d s a

You will be prompted to pick a segment you wish to have deleted. If a picked group has copies, you will be asked the global question.

Delete Vertex or DLV**2D**

The Delete Vertex command removes vertices from Lines and Doublelines. A vertex is either endpoint of a segment of a Line or Doubleline. Two adjacent segments of a multi-segment Line or Doubleline share a vertex. Vertices are deleted by picking them with the crosshairs. You cannot delete either vertex of a single-segment line. When a vertex is deleted, the two segments which shared the vertex become a single segment. If the object picked is not a Line or Doubleline, you will be asked if you would like it converted into a Line so the edit can be accomplished. Segments can be deleted globally from copied groups depending on the settings of the Global option.

Each Undo (<%> key) will restore (undelete) the last vertex deleted. The Undo buffer will remember the last 256 vertices deleted with this use of the command.

This command can also be entered as "DELETE_Vertex" or "DL Vertex".

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Global	= global	1	%PF.G	%DLV.G	=Ask
coordinates			none		

Definition of Options**Global =**

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

DEL vert

You will be prompted to pick a vertex you wish to have deleted. If a picked group has copies, the appropriate global change will be made based on the current setting of the Global option of the Preferences command.

dlv glob=one

You will be prompted to pick a vertex you wish to have deleted. If you picked a vertex within a group which has copies, only the picked group will be edited. Copies of the group will be unaffected. The picked group will become a unique group.

The Dimension command creates two kinds of dimensions, namely Linear and Radial. Several types are available for each kind, namely Incremental, Baseline, Left, Right, Centered, String and Datum. This command has a multitude of other options which determine the exact appearance of the dimension. Most or all of these options default to the values last used, so they rarely have to be set. We have attempted to assign reasonable factory default values to these options in order to give the user a good place to start.

Since you are typically dimensioning existing objects on your drawing, it is essential that you be familiar with the Locating Points chapter of this manual, especially with the concept of "snapping" to existing points or vertices via Graphic Mode Keys and the concept of using the key to snap to existing segments.

Linear dimensions generally require that you locate four points with the crosshairs to get the dimension started and subsequently each additional point will give you an additional dimension. The first four points correspond to a starting point, a point which determines the angle of the dimension, a point which determines the height of the dimension line, and the end point. Subsequently, you are locating only end points. Linear dimensions are either Smart or Non Smart, as controlled by the "Smart=" option. Smart means that the entire dimension is one complex primitive. Moving any of the leader lines in a Smart dimension with the Move Vertex command will automatically adjust the dimension value, giving the dimension a certain degree of "associativity". Non Smart means that the resulting dimension is a collection of individual lines and text primitives unassociated with one another. A Smart dimension can be converted into a Non-Smart dimension by converting it into a line via the Convert option of the Line command. Non-Smart dimensions cannot be converted into Smart dimensions.

Radial dimensions also generally require that you locate four points with the crosshairs to get the dimension started and subsequently an additional point for each additional dimension. The four points correspond to a starting point (or vertex of the radial dimension), a point which determines the first radial line, a point which determines the distance from the vertex of the

dimension line, and a point which determines the second radial line. Subsequently, you are locating only additional radial lines. The primitives that represent radial dimensions are automatically grouped and are always Non Smart. In order to edit the primitives that comprise a radial dimension, you must first "ungroup" it via the Ungroup command.

The parameters of the text within dimensions (i.e. Text Height, Slant, Font, etc.) are controlled by the currently set options of the Text command.

Each Undo (<%> key) will remove the last point located on the current dimension.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Kind	= Linear or Radial	1	last	%DM.K	=Linear
Type	= Incremental or Baseline or Left or Right or Centered or String or Datum	2	last	%DM.T	=Incremental
Leader	= leader	3	last	%DM.L	=Arrow
LENgth	= number	4	last	%DM.LEN	=.125"
Width	= number	5	last	%DM.W	=.125"
TExt	= Above or Centered or Belo or Horizontal	6	last	%DM.TE	=Centered
Spacing	= number	7	last	%DM.S	=.25"
Extension	= Points (or Pts) or number	8	Points	%DM.E	=Points
	= number	9	last	%DM.G	=.0625"
Overrun	= number	10	last	%DM.O	=.0625"
Dimension	=number	11	last	%DM.D	=.375"
Angle	= Points (or Pts) or number	12	Points	%DM.A	=Points

						D Commands
Units	= unit	13	last	%DM.U	=Inches	
Rounding	= number	14	last	%DM.R	=.0625"	
Format	= Fractional or Integer or One or Two or THree or Four or Five or Six or Float or None	15	last	%DM.F	=Fractional	
Radial	= Multiple or Included or Axis or Horizontal or Vertical	16	last	%DM.RA	=Multiple	
EXTStyle	= style	17	last	%DM.EXTS	=Solid	
SMart	= switch	18	On	%DM.SM	=On	
SHow	= switch	19	last	%DM.SH	=On	
Class	= 0 thru 255	20	%WK.C	%WK.C	=0	
Layer	= 0 thru 255	21	%WK.L	%WK.L	=0	
Pen	= 0 thru 255	22	%WK.P	%WK.P	=0	
STyle	= style	23	%WK.S	%WK.S	=Solid	
PRiority	= 0 thru 255	24	%WK.PR	%WK.PR	=125	
WOrkset	= workset	25	%WK.W	%WK.W	=none	
Mimic			none	%DM.M		
coordinates			none			

Definition of Options

Kind =

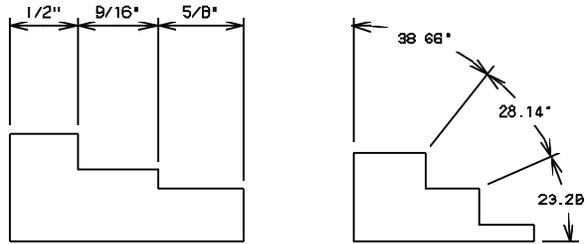
This option selects the kind of dimensioning you want. The two kinds are Linear and Radial. There are seven "Types" of dimensioning for each kind. See the examples following.

Linear means that the dimension will measure linear (straight) distances.

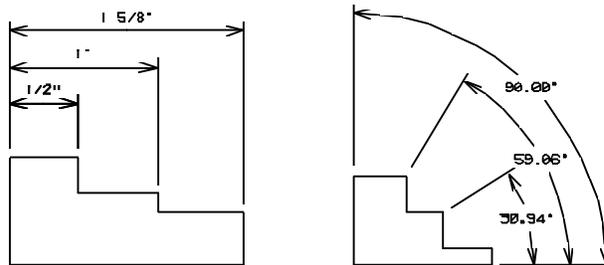
Radial means that the dimension will measure angles.

D Commands

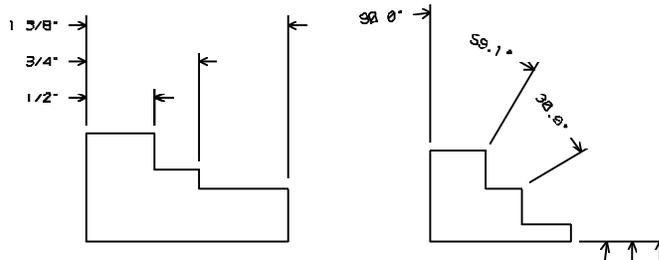
Type =



Incremental creates dimensions where one dimension follows another. After the dimension has been defined with the first three points, the calculated dimension is the distance (or angle) between the last two dimension points located.

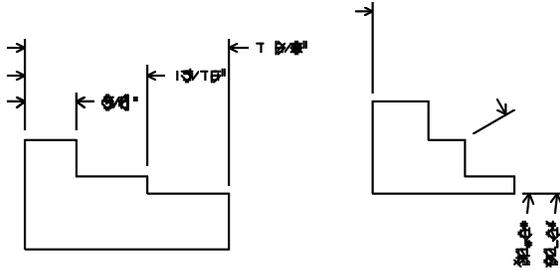


Baseline creates stacked dimensions originating at a common point. The calculated dimension is the distance (or angle) between the first and last dimension point located.

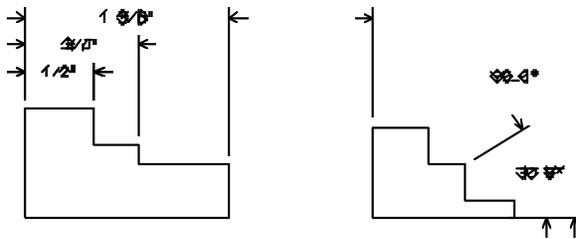


Left is the same as Baseline, except that the dimension line is on the outside of the extension line with the leaders pointing in. The text is located outside the dimension on the left side.

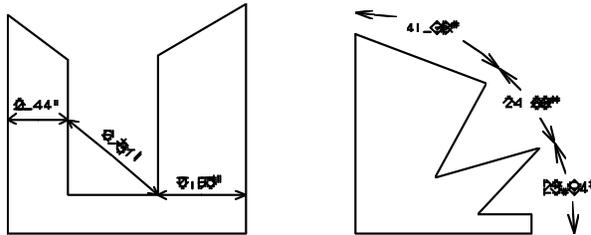
D Commands



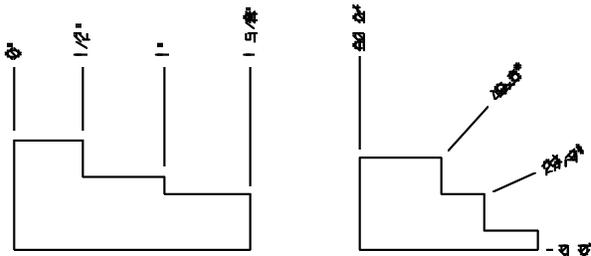
Right is the same as Left, except the text is located on the right.



Center is the same as Left, except the text is centered between the extension lines.



String creates dimensions where one dimension follows another with no extension lines drawn. The calculated dimension is the distance (or angle) between the last two dimension points located.



Datum draws extension lines with dimension text (no dimension lines) originating at a common point. The calculated dimension is the distance (or angle) between the first and last dimension point located.

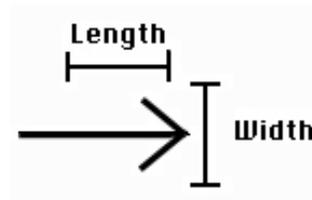
Leader =

Selects the leader that is to be placed on the ends of the dimension line. The leaders currently available are shown below.

→ Arrow	→ X	→ Down
→ Closed Arrow	→ ● Dot	→ ○ Loop
→ Open Arrow	→ ○ Up	→ □ Square
→ Fold Arrow	→ ○ Circle	→ / Slash
→ In Arrow	→ ■ Box	→ ~ Squiggle

LENGth =

This option determines the length of the Leader. Enter a number that is greater than zero. The length is defined in Plot units, not Work units.

**Width =**

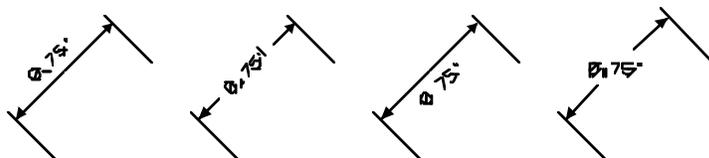
This option determines the width of the Leader. Enter a number that is greater than zero. The width is defined in Plot units, not Work units.

TExt =

For Linear dimensioning, this option sets the location of the dimension text relative to the dimension line. Valid Text values are:

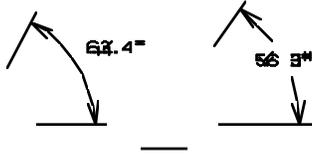
Above	Text is placed above the dimension line
Centered	Text is placed centered in a gap in the dimension line
Below	Text is placed below the dimension line
Horizontal	Text is always horizontal

In the case of Above and Below, the text will be offset from the dimension line by the value set in the Offset option of the Text command. For Centered text, the gap in the dimension line is determined by the amount of text to be accommodated. Dimension text is parallel to the dimension line except when Horizontal is chosen.



D Commands

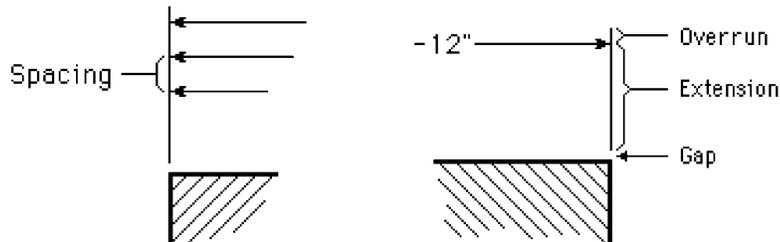
For Radial dimensioning, only a limited amount of flexibility is available.



(Note: For Incremental, Baseline, Center, and String types, the text the angle) is always horizontal and the position of the text can only be Above or Centered on the dimension line. Choosing Below or Horizontal in these types will result in Centered text. In Left and Right types, the text is always centered on the leader and at the angle of the leader and this option has no effect. In Datum type, the text is always centered on the extension and at the angle of the extension and this option has no effect.

Spacing =

This option sets the vertical spacing between stacked dimension lines. Enter a number that is greater than zero.



Extension =

This option sets the length of the extension line. The extension line length is measured from the point located to the dimension line (the overrun is not included). Valid Extension values are any number or the word Points (or Pts). Points means that you will be required to locate an extra point to define the length of the extension line. This point is located after the start point of the dimension and after the Angle point (if required).

Gap =

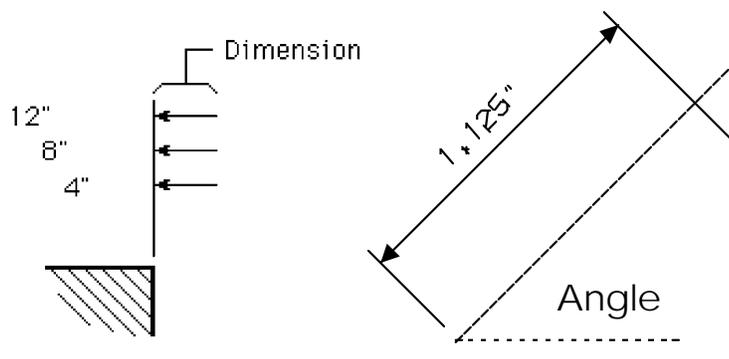
This option sets the gap between the points you locate and the extension line. This allows you to 'snap onto' an object and not have the extension line touch it. Enter a number that is greater than zero.

Overrun =

This option determines how far the extension line will extend beyond the dimension line. The Overrun can be any number greater than zero.

Dimension =

Sets the length of the dimension line in the case of outside dimension leader lines. This option is only used if the Type option is set to Left, Right or Center. The Dimension length must be a number greater than zero.

**Angle =**

This option sets the rotation angle for Linear dimensions. A valid Angle value is any number between -360 and 360 degrees, or -2π and 2π radians. See the Common Options section for more information.

The word Points (or Pts) can also be entered. You will be prompted to locate an additional point after the start point of the dimension.

Units =

This determines the type of units for Linear dimensions. For Radial dimensioning, the units are always degrees and this option has no effect. The valid unit types are shown here.

Mils or **MI** are thousandths of an inch. Dimensioning in mils is most commonly done with Format set to one of the decimal settings (like "Three").

Inches means the dimension will be calculated in inches. Inches can be presented in either a fractional or decimal format.

Feet or **Ft** dimensions in feet. If the Format option is set to Fractional, then dimensions are displayed in a feet/inches/fractions of an inch format (ie: 23' 4 3/16"). Any other Format value displays decimal feet (ie: 45.625 ft).

Miles means you are calculating your dimension distance in miles. Fractional format miles looks like '34 3/4 mi', and decimal format miles looks like '34.75 mi'.

MILLimeters or **MM** dimensions in millimeters. Although millimeters can be presented in a fractional format, it is more common to present them, and other metric units, in a decimal format.

Centimeters or **CM** dimensions in centimeters.

Meters measures distances in meters for dimensioning.

Kilometers will dimension distances in kilometers.

D Commands

Rounding =

This option affects the "rounding" of distances in Linear dimensioning and does not affect angles in Radial dimensioning. Calculated distances will be rounded to this value. The dimension text will display the actual distance rounded to the nearest increment of this rounding value. When working in English units like Feet or Inches it is common to set this value to something like 1" or 1/8". When working in metric units it is best to change this value to a number like .001 or .1.

Note that the Format option is capable of overriding the Rounding value if it is set to a larger value (ie: less decimal places). When working in a fractional environment, this Rounding value should always be set to a value equal to or larger than, and a multiple of, the Preferences Rounding value. For more information, see the Preferences and Rounding commands.

Format =

This option determines what format the dimension text is to be presented in. Some of the Format settings control the number of decimal places that are to be displayed. If the number of decimal places displayed by this option is less than the value set in the Rounding option, the dimension text will be rounded to the Format value.

Fractional means that the dimension text will present the calculated distance in a fractional format for Linear dimensions and in Degrees - Minutes - Seconds format for Radial dimensions. Some examples of fractional formats are: 3 1/2", 4 3/5 mm, 54' 4 7/8", 45°35'45". Linear fractional feet will always be expressed as feet, inches, and fractions of an inch.

Integer means that dimensions will be rounded to the nearest whole unit.

One means that dimensions will be rounded to the nearest one decimal place (1/10 of a unit) and presented in a decimal format (like 34.5 ft).

Two is the same as "One", except that it rounds to the nearest two decimal places.

Three is the same as "One", except that it rounds to the nearest three decimal places.

Four is the same as "One", except that it rounds to the nearest four decimal places.

Five is the same as "One", except that it rounds to the nearest five decimal places.

Six is the same as "One", except that it rounds to the nearest six decimal places.

Float is the same as "One", except that the dimension will be presented as accurately as it can be calculated. This will usually display up to seven decimal places.

None means that no text will be displayed. Dimension lines will be drawn with no dimension text.

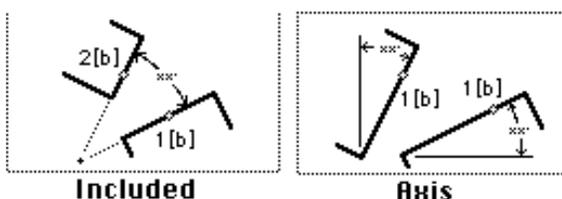
RArial =

This option gives you additional flexibility in Radial dimensioning and is ignored for Linear dimensioning. Valid entries for this option are:

Multiple is the most common setting for this option. That is, you create multiple, or "chained" angular dimensions much in the same way you create linear dimensions. The angle is always measured between the last two dimension points you located.

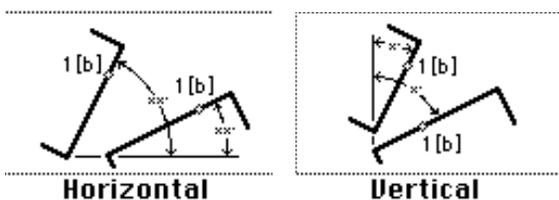
The other valid settings, Included, Axis, Horizontal, and Vertical, are all used for creating single (not chained) angular dimensions. They are typically used for dimensioning a single angle between existing objects on a drawing. The user first defines the "center" of the dimension, then the first dimension point, then a point to define the position of the dimension line, and finally the second dimension point. The user may select an existing segment with the key, and this will determine automatically the dimension "center", the first dimension point, and the location of the dimension line. No extension line will be drawn in this case.

Included means that the angle measured will always be the angle between two dimension points or existing segments.



Axis means that only one existing segment or dimension point is located by the user and the angle measured is the angle with the nearest orthogonal axis (X or Y).

Horizontal means that only one existing segment or dimension point is located by the user and the angle measured is the angle with the horizontal, or X-axis.



Vertical means that only one existing segment or dimension point is located by the user and the angle measured is the angle with the vertical, or Y-axis.

EXTStyle =

Sets the line style for the extension lines. The extension line style can be any of the line styles shown below for the Style option. See **Common Options** for more information.

SMart =

If this option is set to "On", the dimension command will create dimensions that are a unique primitive type. If set to "OFF", the dimensions created will be simple text and line primitives. Radial dimensions are always Non-Smart but are grouped for convenience. Any other switch value can also be used as a value for this option. Switch values are: On, OFF, Yes, No, All, None, True, False.

Smart linear dimensions have some degree of associativity. If you use the Move Vertex command to move the end of an extension line, the dimension is automatically recalculated. The Change command lets you change the properties of smart dimensions, including the Type, Format, Units, etc.

D Commands

Non-Smart linear dimensions are text and line primitives and can be edited as such. Radial dimensions must be ungrouped prior to editing the individual primitives that make up the dimension. The wording of the dimension text can only be altered with the ModifyText command on Non-Smart dimensioning. You change a Smart dimension into a Non-Smart dimension via the "Convert" option of the Line command. See the Line command for more information on converting complex primitives into their component parts.

SHow =

If this option is set to "On", dimensions will display the unit type after dimension text (like this: 2.540 mm). If set to "Off", no unit type will be shown (like: 2.540). Any other switch value can also be used as a value for this option. Switch values are: On, Off, Yes, No, All, None, True, False.

CLass =

See **Common Options** for more information.

Layer =

See **Common Options** for more information.

Pen =

See **Common Options** for more information.

STyle =

See **Common Options** for more information.

PRiority =

See **Common Options** for more information.

WOrkset =

See **Common Options** for more information.

Mimic

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Dim

Dimensions will be drawn with the settings of all options set the same as the last time the command was used. Except: Class, Layer, Pen, and Style will default to the Status Palette values, Extension and Angle will default to "Points", and Smart will be On.

dm,,b t=a u=mm f=th sh=n p=3 la=6

Baseline type dimensions will be drawn in pen three on layer 6. The calculated distance will be displayed in millimeters rounded to the nearest three decimal places. No unit type (mm) will be shown. The dimension text will be above the dimension line. Class, and Style will default to the values shown on the Status palette. All other options will default to their last used values, except Extension, Angle and Smart.

D Commands

DM K=R For=Frac Type=Incr Rad=Mult

Incremental radial dimensions will be drawn with the text in Deg-Min-Sec format. Class, Layer, Pen and Priority will default to Status palette values. All other options will default to last used values, except Extension.

DIGitizer or DG

The Digitizer command assigns graphic cursor (crosshairs) control to a digitizing tablet. When a digitizer is active, you can control the movement of the screen pointer or crosshairs with either the mouse or the cursor or pen of the digitizing tablet. Furthermore, you can set up your digitizer so that selection from the icon menu or pull-down menus can be done from the digitizer cursor as well as from the mouse. This level of flexibility allows you to operate TurboCAD entirely from the digitizer.

Any area of the digitizing tablet surface can be mapped to an area of your TurboCAD drawing. Defining the area by locating one, two or three points allows you to control the location, X axis and Y axis scale, and rotation of the area on the digitizing tablet relative to the area on your drawing.

Any area of the digitizing tablet surface can be mapped to your Macintosh screen. Defining the area by locating two diagonally opposite points allows you to control the "size" of your screen area on the digitizer. This screen area proportionately contains all of your icons and menus.

Any Graphic Mode function (ie: key on your keyboard) can be assigned to any button on the digitizing tablet's cursor. Other options of this command are used to establish the communications settings for the digitizer (baud rate, data bits, stop bits, parity, and port).

TurboCAD supports tablets which are driven through the serial port as well as those which are controlled via the Apple Desktop Bus (ADB) port.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Protocol	= None or Adb or Binary6 or Binary8 or Char4 or Char5 or Char6 or Code1 or Code2 or Ibm	= None	1	last	%DG.P

Match	= Previous or One or Two or Three	2	Previous	%DG.M	= One
Keys	= string	3	last	%DG.Kvcezap%kojntxya`	
Screen	= None or Last or Points (or Pts)	4	Previous	%DG.S	= None
Baudrate	= 9600 or 2400 or 1200 or 300	5	last	%DG.B	= 9600
DAtabits	= 7 or 8	6	last	%DG.DA	= 8
Stopbits	= 1 or 2	7	last	%DG.S	= 2
PArity	= None or Odd or Even	8	last	%DG.PA	= None
STEmsize	= number	9	last	%DG.STE	= 1000
POrt	= Modem or Printer or Adb	10	last	%DG.PO	= Modem
coordinates			none		

Definition of Options

Protocol =

This option sets the communication Protocol for the digitizing device. The Protocol is the general "language" used by your digitizing tablet for communication with a computer. Consult your tablet's operating manual or technical specifications for the proper setting of this option. Valid values for this option are:

None means no protocol. Use this option to "deactivate" your digitizer.

Adb means that the digitizing tablet is connected via the Apple Desktop Bus (Adb) on your Macintosh. The keyboard and mouse on a Macintosh are connected via Adb, and the digitizing tablet can be "chained" with them. The tablet can also be connected to the extra Adb port, if available, on your Macintosh. Apple pre-defined the protocol to be used with Adb devices, so if your tablet has an Adb connector, this is the right protocol. Consult your tablet's operating manual to verify compatibility with Adb.

All of the rest are standard serial protocols, meaning that your digitizing tablet is connected via a serial port on your computer. If your digitizer is a serial device, consult its operating manual to determine which of these applies.

Match =

This option determines how an area on the digitizing tablet will be matched to your drawing workarea. Valid Area values are Previous, One, Two and Three.

Previous means the area defined in the last use of this command will again be used. If no previous area has been defined, the lower left corner of your TurboCAD drawing will be positioned at the lower left corner of the active area of the digitizing tablet.

One means you will be prompted to locate one point on your TurboCAD drawing with the mouse and one point on the digitizing tablet with the cursor. The point on your drawing will be mapped to the point on the tablet. No adjustment will be made for scale or rotation at the digitizer and you will be digitizing at a one-to-one ratio.

Two means you will be prompted to locate two points on your TurboCAD drawing and two points on the digitizing tablet. The two points on your drawing will be mapped to the two points on the digitizer, thereby adjusting both the scale and rotation of the digitizer. The two points are assumed to be along the positive X axis. Scaling adjustment is made equally to both axes, so no adjustment is made for distortion of your paper on the digitizer. For example, determine two horizontal points on the digitizer (or two such points on the sketch or drawing which you have on your digitizer) for which you know the exact scaled distance. Also determine two similar horizontal points on the TurboCAD drawing on your screen. Use rounding or otherwise accurately key in the two points on your drawing. Hit the two points on your digitizer as accurately as possible. Even if the two points on your digitizer are not perfectly horizontal, TurboCAD will adjust for this automatically.

Three means you will be prompted to locate three points on your TurboCAD drawing and three points on the digitizing tablet. The three points on your drawing will be mapped to the three points on the digitizer, thereby adjusting the X axis scale, the Y axis scale and the rotation of the digitizer. The first two points are assumed to be along the positive X axis and the third is used solely to determine the Y axis scale thereby giving you the ability to adjust for paper distortion. Use the same procedure as described above for entering the points.

Keys =

This option assigns the Graphic Mode (view/locate) functions that will be performed by the buttons on the tablet's puck. The value for this option must be a string. Each character of the string is a key on the keyboard that is the keyboard equivalent for a function. For example, the string `VMZ` would assign the Vertex function to the first button, the Midpoint function to the second button, and the Zoom function to the third button. It is recommended that the string be quoted (like this` or \like this) as it may contain characters that are normally considered delimiters. Note: See the instructions

D Commands

that accompany your tablet to determine the order of the buttons on your tablet's cursor. The default string supplied with the software is `vcezup%kojntxya`, which correspond to:

<u>Button</u>	<u>Key</u>
1	Space (anywhere)
2	v (vertex, snap to nearest vertex)
3	c (closest point)
4	e (end current command, execute again)
5	z (zoom in)
6	u (unzoom, zoom out)
7	p pan to point)
8	% (undo, reject)
9	k (close, for lines, doublelines, fillets, etc.)
10	o (origin, snap to origin)
11	j (junction, snap to junction)
12	n (normal, perpendicular)
13	t (tangent, parallel)
14	x (x-axis lock)
15	y (y-axis lock)
16	a (angle lock)

Note: If your cursor has less than 16 buttons, TurboCAD will disregard those key assignments after the last button. When entering a string, keep in mind that the first one above is a "space, " which corresponds to clicking the mouse without a view/locate key.

Baudrate =

Sets the baud rate (communication speed). Valid Baudrate values are 9600, 2400, 1200 and 300. Set this value to match the baud rate set on the digitizing tablet. This option is not necessary for Adb tablets.

Screen =

Defines which part of your tablet will be mapped to your screen so that you may use the tablet's cursor to select from the icon menu and pull-down menus. A tablet must already be defined and active in order to use this option. The area on the tablet need not be the same size as the screen, and everything on the screen will be proportionately reduced or enlarged when mapped to this area. When defining this screen area, it is a good idea to previously obtain and print a "screen dump" while in TurboCAD and adhere it to your tablet. Be careful not to scale down the size of the picture too much.

None means that no area of the digitizer is mapped to the screen. This serves as a way to "turn off" an area if already defined.

Last means that the area last defined will again become active, provided you are using the same tablet and have not re-defined it.

Points means that you will be prompted to locate two diagonally opposite points on your tablet which will define the rectangular area to be mapped to your screen. After you define the two points, that area will be actively mapped to your screen.

DAtabits =

Sets the number of data bits that will be used. Valid Databits values are 7 and 8. Set this value to match the data bits set on the digitizing tablet. This option is not necessary for Adb tablets.

Stopbits =

Sets the number of stop bits that will be used. Valid Stopbits values are 1 and 2. Set this value to match the stop bits set on the digitizing tablet. This option is not necessary for Adb tablets.

Parity =

Sets the parity that will be used. Valid Parity values are None, Odd and Even. Set this value to match the parity set on the digitizing tablet. This option is not necessary for Adb tablets.

STEpsize =

Sets the "accuracy" as defined for your digitizer or tablet in steps/inch. Valid values are numbers greater than zero. Most digitizers/tablets fall in the range of 500 to 2000 steps per inch. Consult your tablet's manual for this information.

POrt =

Selects the port that the digitizer is plugged into. The Port must be set to Modem or Printer for serial tablets and Adb for Adb tablets.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Digitizer Adb two Port=Adb

Provided an Adb tablet is connected to your Adb port and powered up, you will be prompted to locate two horizontal points on your TurboCAD drawing, then two points on your tablet. Stepsize and Keys will default to the last used values. Your tablet will then be active and cursor control will be available both from the mouse and the tablet's cursor.

dig screen=points

Provided an Adb tablet is already active, you will be prompted to locate two diagonally opposite points on the tablet to be mapped to your screen. Subsequently, the tablet's cursor will act as a mouse pointer in that area.

D Commands

dig no

Returns graphic cursor control to the mouse and de-activates the currently active digitizer/tablet.

DG,binary6,T,`PZUVOM` PO=PR

You will first be prompted to locate two points on your TurboCAD drawing and then two points on the digitizing tablet. The scale and rotation of the digitizer will be adjusted to compensate for differences in the distance and orientation of the pairs of points located. The Protocol will be set to binary6. The Baudrate, Databits, Stopbits, Parity, and Stepsize will default to the last used values. The functions Pan, Zoom, Unzoom, Vertex, Origin and Midpoint will be assigned sequentially to the buttons of the tablet's puck. The digitizer will be driven through (and must be connected to) the printer port

Display or DP

2D & 3D

The Display command sets the Classes and Layers that are to be displayed on your screen. Specific pens can also be temporarily displayed.

Classes and Layers that are not visible are not editable (they cannot be changed). Layers that are visible are editable. However, Classes can be visible but not editable. See the Class command for controlling the editability of classes.

Class display has priority over Layer display. This means if a class is not displayed, no objects on that class will be displayed, even if the Layers of those objects are being displayed.

You cannot turn off the display of the current work layer or class. The class and layer you are working on must always be visible. Classes and Layers that are not displayed are still part of your drawing and will be saved with the drawing. See the Class, Layer, and Work commands for functions that work in conjunction with this command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Classes	= classes or All or Display or Used or Edit	1	last	%DP.C.n	=All
Layers	= layers or All or Display	2	last	%DP.L.n	=All

	or Used				
Pen	= 0 thru 255	3	none	%DP.P	=0
Regenerate			none	%DP.R	

Definition of Options

Classes =

Sets the classes that are to be displayed. A valid Classes value is a list of classes or the words "All," "Display," "Used," or "Edit."

classes is a list of the classes that you wish to have turned on and off from the classes that are currently being displayed. There are 256 classes on each drawing, numbered 0 through 255. List the classes you wish to have turned on, separating each number with a space or comma. Ranges on classes can be entered by separating the starting class and ending class with an ellipsis (three periods, like this: 10...20). Individual classes and ranges of classes are turned off by preceding them with an up arrow (shift-6 on the keyboard, like this: ^).

Here is an example how you would turn on classes 23, 43, 100 through 200 for display, and turn off classes 150 through 175 for display: "DP Classes = 23 43 100...200 ^150...175".

All means that all classes will be turned on for display.

Display means the classes currently being displayed will be displayed. (Now that sounds kind of silly, doesn't it?)

Used means only the classes that currently contain data will be displayed.

Edit means only the classes that are currently editable will be displayed. Classes are made editable using the Edit option of the Class command.

Layers =

Sets the layers that are to be displayed. A valid Layers value is a list of classes or the words "All," "Display," or "Used." Changing the display of Layers automatically forces a Regenerate. Changing the display of Classes does not.

layers is a list of the layers that you wish to have turned on and off from the layers that are currently being displayed. There are 256 layers on each drawing, numbered 0 through 255. List the layers you wish to have turned on, separating each number with a space or comma. Ranges on layers can be entered by separating the starting layer and ending layer with an ellipsis (three periods, like this: 10...20). Individual layers and ranges of layers are turned off by preceding them with an up arrow (shift-6 on the keyboard, like this: ^).

All means that all layers will be turned on for display.

Display means the layers currently being displayed will be displayed. This really doesn't change anything, except it forces a redisplay of your drawing.

Used means only the layers that currently contain data will be displayed.

D Commands

Pen =

Selects one pen for temporary display. A valid Pen value is a number from 0 through 255. You are not able to work in this temporary display mode. Press any key to restore the display of all pens. This option is used to verify which objects are drawn in a given pen. It is especially useful for verifying pen assignments before plotting when working on a monochrome monitor.

Regenerate

The current "display list" is discarded and a new one is created from the working file on disk. The display list is the file that controls the display of objects on your screen. Extensive editing can cause "holes" in the display list making redispays and object pick times longer than necessary. This option creates a new optimized display list to eliminate having to search through unnecessary data. This option is also useful when you suspect some type of display error.

Examples

Display Class=All

All classes will be made visible. Note that this does not necessarily make them editable.

DP L=200...250, ^225, 100

Layers 100 and layers 200 thru 250 will be added to the list of currently displayed layers. Layer 225 will be removed from the list of displayed layers.

DP R

The "display list" of your drawing will be regenerated and optimized. It is useful to do this once in a while after extensive editing has been done to a drawing or if you notice something unusual with your drawing on the screen.

DP C=^50 R

Class 50 will be made invisible. A regeneration of the display list will be performed.

DISTance or DS

2D

The Distance command calculates distances on your drawing. You can find the length and angle of a line segment, an entire linear object or a vector defined by locating two points. A cumulative total will be displayed in the News Area or a screen dialog for each of these calculations. Distances will always be rounded to the increment currently set for the "Rounding=" option and in the format currently specified by the "Format=" option of the Preferences command.

After entering the command and the desired option, you will be prompted to locate points or pick an object depending on your choice of options.

A maximum of 50 calculations can be made with each use of the command. The results of each calculation are remembered in these Current Status Words.

D Commands

<u>CSW</u>	<u>Value</u>
%DS.Distance	Length of vector
%DS.Angle	Angle of vector
%DS.Xdistance	X axis length of vector
%DS.Ydistance	Y axis length of vector
%DS.Tdistance	Total length

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Type	= Points (or Pts)	1	Points	%DS.T	=Points
or String					
or SEgments					
or Objects					
Silent			none	%DS.S	
coordinates			none		

Definition of Options

Type =

This option sets the type of distance calculation you wish to perform. Valid Type values are Points, String, Segments and Objects.

Points means the individual segment length, angle, length along the X axis, length along the Y axis and the cumulative length will be displayed for each vector defined by locating two points. Two points are required for each calculation.

String works similar to Points except that two points are not required for each calculation as the last point from the previous calculation is automatically used as the first point for the next calculation. The first calculation would require two points, but each successive calculation would require only one point.

SEgments calculates the same information as Points except that the calculation is made on an existing line segment on your drawing. You will be prompted to pick an existing line segment.

Objects means the individual and cumulative length will be displayed for each object picked. The length of an object is the total length of its segments. The length of circles and arcs can also be calculated. The results of each calculation are remembered in %DS.Distance and %DS.Tdistance.

Silent

If this keyword-only option is used, the results of the calculation will not be displayed. However, the Current Status Word values will still be set. This option is most commonly used when distance calculations are performed in a Macro.

D Commands

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Distance

You will be prompted to locate two points for each calculation. The results of the calculation will be displayed on the screen and remembered in the CSWs.

DS,SEG

You will be prompted to pick an existing segment of an object for each calculation. The results of the calculation will be displayed on the screen and remembered in the CSWs.

DDoubleline or DB

2D

The Doubleline command draws two parallel lines. The lines can be any specified distance apart and offset from the points you locate by any amount. There are options for a centerline and automatic closure of the doubleline ends. This command can also convert an existing Line primitive into a Doubleline .

The user should be cognizant of the differences, and similarities between, Doublelines and Walls, which are parallel lines created with the Wall command. A Doubleline is a primitive. That is, the parallel lines (and centerline) are a single primitive whose structure determines what is displayed on the screen, much as a Fillet, Rectangle, Circle, etc. are primitives. As such, the structure of the Doubleline can be changed via the Change Object command. Its vertices and segments can be moved "as one" via the Move Vertex and Move Segment commands. This makes it possible to edit a Doubleline extensively and always ensure that the structure of all of its vertices and segments remains intact. A Wall may appear the same as a Doubleline, but it is just a line primitive with no additional structure. As so, its segments and vertices are only editable as parts of a line. Walls, however have some additional advantages, such as intersection "cleanup" and automatic gapping of objects with which the Wall intersects. These, among other "trade offs" are discussed in detail in the Wall command.

A Doubleline may be converted to the equivalent of a Wall (or line) via the "Convert" option of the Line command.

Pressing the <Esc> (escape) or <clear> key will put a gap in a Doubleline by making the next segment invisible. The <K> (close) key will close the current doubleline by returning to the starting point (origin) of the line. If the first and last point of a doubleline are in the same location, the corner formed will be cleaned up.

Each Undo (<%> key) will remove the last point located. The Undo buffer will remember the last 84 points located with this use of the command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Width	= number	1	last	%DB.W	=.5"
Offset	= number	2	last	%DB.O	=0
Centerline	= style or None	3	None	%DB.C	=None
CAp	= None or Start or End or Both or Gaps or All or Lines	4	None	%DB.CA	=None
Class	= 0 thru 255	5	%WK.C	%WK.C	=0
Layer	= 0 thru 255	6	%WK.L	%WK.L	=0
Pen	= 0 thru 255	7	%WK.P	%WK.P	=0
STyle	= style	8	%WK.S	%WK.S	=Solid
PRiority	= 0 thru 255	9	%WK.PR	%WK.PR	=125
WOrkset	= workset	10	%WK.W	%WK.W	=none
Convert			none	%DB.CO	
Mimic			none	%DB.M	
coordinates			none		

Definition of Options**Width =**

Sets the distance, in Work Units, between the parallel lines making up the doubleline. The Width value can be any number including zero.

Offset =

Sets the perpendicular distance from the points you define for the Doubleline to the left side of the doubleline. The offset value can be positive (left) , negative (right) or zero (none). This enables you to define points outside the Doubleline and have the Doubleline drawn a specific distance away from them. You may use this value to snap to existing points on your drawing and have a Doubleline created a specific distance away from them. For example, if you are drawing a Doubleline with a width of 1/2 units, and you set the Offset option to 3 units, then

D Commands

locating two horizontal points on your drawing from left to right will produce a Doubleline whose left side is 2-3/4 units below the points you located.

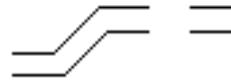
Centerline=

This option will draw a separate Line primitive between the points located at the same time the doubleline is being drawn. The centerline is drawn between the points located. If an Offset value is used while drawing a doubleline, then the centerline will not necessarily be in the center of the doubleline.

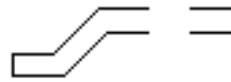
CAp =

This option will close the ends of the doubleline segments as described below. Valid Cap values are:

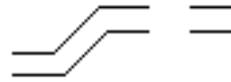
None means the ends of the doubleline will be open.



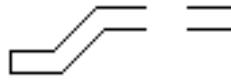
Start means only the first point of the doubleline will be closed.



End means only the last point of the doubleline will be closed.



Both means only the first and last points of the doubleline will be closed.



Gaps means both ends of any gaps will be closed.



All means both ends of any gaps will be closed.



Lines means the start and end of the visible lines of the doubleline will be closed. This is the first and last points of the entire doubleline and both ends of any gaps.



Class =

See **Common Options** for more information.

Layer =

See **Common Options** for more information.

Pen =

See **Common Options** for more information.

STyle =

See **Common Options** for more information.

PRiority =

See **Common Options** for more information.

WOrkset =

See **Common Options** for more information.

Convert

This option allows you to convert existing Lines on your drawing into Doublelines. You will be prompted to pick an existing Line on your drawing instead of locating points to create a Doubleline.

Mimic

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Double Width=1/8

Doublelines will be drawn with a distance of one-eighth unit between the parallel lines. The Offset, Close and Centerline options will default to None. The Class, Layer, Pen, and Style options will default to the values shown on the Status Palette.

DB %tx.he o=%tx.he/2 c

You will be prompted to pick an existing line on your drawing that will be converted into a doubleline with a distance equal to the current text height between the parallel lines. The Offset will be set to half that value meaning the doubleline will be drawn along its left face. Close and Centerline options will default to None. The Class, Layer, Pen, and Style options will default to the values shown on the Status Palette.

Ellipse or EL**2D**

The Ellipse command creates ellipses and elliptical arcs. Ellipticity is determined by setting a combination of major axis size, minor axis size, major to minor axis ratio or projection.

Each Undo (<%> key) will remove the last point located on the current ellipse. Each Undo past the current ellipse will remove one complete ellipse. The Undo buffer will remember the last 56 ellipses drawn with this use of the command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Construction	= Center or Ends or Box or COrner	1	last	%EL.C	=Box
Ratio	= 0 thru 1	2	last	%EL.R	=.5
Angle	= angle or Points (or Pts)	3	0	%EL.A	=0
Starting	= angle	4	0	%EL.S	=0
Included	= angle	5	360	%EL.I	=180
Major	= number	6	last	%EL.M	=2
MInor	= number	7	last	%EL.MI	=1
Projection	= angle	8	last	%EL.P	=30
CLass	= 0 thru 255	9	%WK.C	%WK.C	=0
Layer	= 0 thru 255	10	%WK.L	%WK.L	=0
Pen	= 0 thru 255	11	%WK.P	%WK.P	=0
STYle	= style	12	%WK.S	%WK.S	=Solid
PRiority	= 0 thru 255	13	%WK.PR	%WK.PR	=125
WORkset	= workset	14	%WK.W	%WK.W	=none
Mimic			none	%EL.MIM	
coordinates			none		

Definition of Options

Construction =

This option sets how the ellipse will be constructed. Valid Construction values are:

Center means the ellipse will be constructed by locating its center point and the endpoint of the major axis. If the size of the ellipse is defined by entering the Major or Minor option, only the center point will be required.

Ends means the ellipse will be constructed by locating the endpoint of the major or minor axis depending on other options used.

Box means the ellipse will be fit into a rectangular area that you define by locating the diagonally opposite corners. This construction type cannot be used in conjunction with the Major, Minor, Ratio or Projection options.

Corner means the ellipse will be fit into a rectangular area that you define by locating the center point and one corner. This construction type cannot be used in conjunction with the Major, Minor, Ratio or Projection options.

Ratio =

This option determines the ellipticity by specifying the ratio of the Minor axis length to the Major axis length. A ratio of 1/2 means the minor axis is half as long as the major axis. This option does not set the actual axes lengths, only the ratio. This option will override and reset the value of the Projection option. This option can be used in conjunction with either the Major or Minor option, but not both. This option cannot be used with the Box or Corner construction types.

Angle =

This option sets the angle of the Major axis of the ellipse. A valid Angle value is any number between -360 and 360, or -2¹ and 2¹ radians. See the **Common Options** section for more information.

The word **Points** (or **Pts**) can also be entered. You will be prompted to locate an additional point after the first point to define the angle.

Starting =

This option sets the starting angle of an elliptical arc. Valid Starting values are the same as for the Angle option, except that "Points" is invalid.

Included =

This option sets the included angle of an elliptical arc. Valid Included values are the same as for the Angle option, except that "Points" is invalid.

Major =

Sets length of the major axis of the ellipse. If the Major and Minor options are both used, then they override the Ratio and Projection options.

E Commands

MInor =

Sets length of the minor axis of the ellipse. If the Major and Minor options are both used, then they override the Ratio and Projection options.

Projection =

This option determines the ellipticity by specifying the projection angle of a circle. Imagine looking at a circle on edge and then rotating it about its diameter. A small rotation would give you a very "flat" ellipse which would approach a circle as the rotation angle approached 90 degrees. The projection angle is the number of degrees of this rotation. For example, a projection of 90 is a circle. A projection of 30 would result in a minor-to-major axis ratio of 1/2. Valid Projection values are the same as for the Angle option.

Class =

See **Common Options** for more information.

Layer =

See **Common Options** for more information.

Pen =

See **Common Options** for more information.

STyle =

See **Common Options** for more information.

PRiority =

See **Common Options** for more information.

WOrkset =

See **Common Options** for more information.

Mimic

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Ellip

Ellipses will be constructed by Box. You will be prompted to locate one corner of the box, then a point to define the angle, then the diagonally opposite corner of the box into which the ellipse will be fitted. Class, Layer, Pen, and Style options will default to Status Palette values.

El C Ang=45 Maj=3 Rat=.5

You will be prompted to locate a center point for each ellipse. Each ellipse will be drawn at an angle of 45 degrees and the length of the major axis will be three units. A Ratio of .5 means the minor axis length will be 1.5 units. Class, Layer, Pen, and Style options will default to Status Palette values.

ell end st=0 inc=180 sty=ph

You will be prompted to locate the major axis endpoints for each ellipse. An elliptical arc will be drawn for each two points located. The elliptical arc will be one half of an ellipse cut along the major axis. The style of the elliptical arc will be a Phantom line. Class, Layer, Pen and Priority options will default to Status palette values.

ERror or ER**2D & 3D**

The Error command explains error messages. Other options of this command let you turn the error warning bell on and off and set the number of allowable errors on the command line.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Number	= number	1	last	%ER.N	=99
Limit	= number	2	last	%ER.L	=1
Bell	= switch	3	last	%ER.B	=On

Definition of Options**Number =**

This option specifies which error you would like described. A valid Number value is any error number. TurboCAD currently has about 125 error messages. Associated with each error message is an error description. This option will display both the error message and description to give you a more detailed explanation as to what might have gone wrong.

Limit =

This option sets the number of error messages that will be displayed when processing the command line before the command processor gives up. Once the number of errors has been reached, no more information is read from the command line. Therefore, more errors may be present than have been encountered. The Limit value must be set to a number greater than zero. Regardless of the setting of this option, commands with any errors will not be executed.

Bell =

This option turns the error bell on and off. The Bell value must be a switch value. Switch values are: True or False, Yes or No, On or Off, All or None. A positive value means the bell will sound when an error occurs. A negative value means no bell will sound.

E Commands

Examples

ER

The last error message received and its description will be displayed.

error 45,,n

The error message and description for error number 45 will be displayed. The bell will be turned off. The bell will not sound when errors occur until it is turned back on.

Err Limit=3

Up to the first three errors found on the command line will be displayed and then processing will halt.

EXTEnd or ET

2D

The Extend command "trims" and "extends" existing lines or doublelines on your drawing while retaining their existing angle. The amount of trimming and extending can be specified in a number of ways. The simplest is to "drag" either end point of the line with the crosshairs. Other ways are to specify an imaginary "trim line", select an existing object to "trim" to, or specify an incremental or overall length. Furthermore, this command is capable of extending or trimming (joining) two existing, non parallel, lines so that their vertices (endpoints) meet at a single point. This is commonly known as "cleaning up a corner".

This command complements the Move Vertex command, which has less options and does not retain the angle of the line segments.

Each Undo (<%) key) will remove the last object picked to extend.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Function	= Drag or Points (or Pts) or Length or Join	1	Drag	%ET.F	=Drag
Length	= number	2	none	%ET.L	=1
Mode	= Relative or Overall	3	Relative	%ET.M	=Relative
Global	= global	4	%PF.G	%ET.G	=Ask
Convert			none	%ET.C	

coordinates

none

Definition of Options

Function =

This option sets way in which the amount of trimming or extending is to be defined.

Drag means that you will be prompted to pick an existing segment near one of its vertices and then you will dynamically "drag" it to its new position with the crosshairs.

Points means that you will be asked to define two points which are to be the endpoints of an imaginary "trim line" to which line segments thereafter picked will be trimmed to. An existing object (not necessarily a line) can be selected as the "trim line" by selecting it with the (boundary) key.

Length means that you will be asked to specify the length, in work units, that the line picked will be trimmed or extended to. The actual length is specified in the "Length=" option below and works in conjunction with the "Mode=" option.

Join means that you will be asked to pick two existing line segments. They will both be extended or trimmed so that their endpoints (vertices) meet.

Length =

This option sets the amount of trimming an extending in work units. It only has meaning if the "Function=" option above is set to Length. Any number, either positive, negative or zero is valid. This option works in conjunction with the "Mode=" option below.

Mode =

This option determines whether the length entered for the "Length=" option is to be relative or overall to the existing length of the line picked.

Relative means that the length entered will be added to (a positive length) or subtracted from (a negative length) the existing length of the line picked.

Overall means that the line picked will be extended or trimmed so that its resulting overall length will be the length value entered.

Global =

See **Common Options** for more information.

Convert

This keyword-only option, when used, causes any objects picked which are not lines or doublelines to be converted into lines first before the extending or trimming operation. When this option is not used, you will be asked if you want the object converted.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

E Commands

Examples

Extend

The "Function=" option will default to Drag. You will be prompted to pick the objects you wish to have extended. The nearest vertex of the line or doubleline picked will become attached to the crosshairs and you extend or trim it by clicking down with the mouse.

ET J

You will be prompted to pick two existing lines on your drawing. Their endpoints will be extended or trimmed so that they meet.

ET F=L L=3 M=R

The segment picked will have a distance of three work units added to its length.

ET Pts

You will be prompted to define two points which will determine the imaginary "trim line" to which subsequent segments are to be trimmed or extended to. An existing object can be selected with the key, and it need not be a line.

EXtrude or EX

3D

The Extrude command gives 3-dimensional depth to existing objects. If the object picked is not already a Wireframe, it first converts it into one. The extrusion is always made into and out of the screen (along the Z axis of the Viewplane Coordinate system). The user specifies how many equidistant "copies" of the object are to be made along the extrusion depth. All of the "copies" are connected via wireframes through the vertices.

Each Undo (<%> key) will remove the last object picked to extrude.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Z	= number	1	current	%EX.Z	=0
Depth	= number	2	last	%EX.D	=1
Repeat	= 0 thru 127	3	last	%EX.R	=2
coordinates			none		

Definition of Options

Z =

This option sets the Z axis depth that the extrusion should start at. This depth is measured along the Z-axis of the Viewplane Coordinate System. That is, along the axis normal to the screen.

Depth =

This option sets the total depth that the extrusion should have, starting at the Z value specified above. A positive Depth value is towards you (out of the screen) and a negative value is away from you (into the screen).

Repeat =

Sets the number of copies that will be made of the object. The number of repeat copies will be divided equally between the Z value and the Depth value.

Examples

Extrude 0 5 2

You will be prompted to pick the objects you wish to have extruded. The objects will be extruded from a Viewplane coordinate system depth of zero to five with two repeats. Two repeats means a copy of the object at zero depth and a copy of the object at a depth of five with wireframe segments connecting the vertices of the two copies.

Ex 3

You will be prompted to pick the objects you wish to have extruded. Because no Depth and Repeat values were entered, they will default to the last used values. Assuming the previous example was the last used Extrude command, the objects will be extruded from a Viewplane coordinate system depth of three to five with two repeats.

Fields are created in TurboCAD via the Text command (using the Field and Attribute options). They are pieces of text for which the wording has not been specified, like "place holders" for text. The most common use for fields is to include them in symbols or groups. When desired, the user can "fill" these with the particular text he wants. An example would be a resistor symbol with a text field to specify the resistance value.

The Field command fills existing text fields on your drawing. The field can be a "stand alone" object or part of a group. If several fields are part of a picked group, you will be given the opportunity to fill each field in the order it was made part of the group. Fields can be skipped over and left for later filling by entering a <Return>. Fields can contain only one line of text, not a complete paragraph. Once filled, if a field was originally given an Attribute number, it becomes an attribute, otherwise it is regular text.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Global	= global	1	%PF.G	%FD.G	=Ask
coordinates			none		

Definition of Options

Global =

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Field

You will be prompted to pick text fields on your drawing that you wish to fill.

FD A

You will be prompted to pick text fields on your drawing that you wish to fill. If there are copies of a group that you pick, all of the copies will be filled with the same value.

FILE or FF**2D & 3D**

The File command manipulates the text files for use by the Readline and Writeline commands and can also be used to check for the existence of any type of file. Up to 4 text files can be open for read/write functions at one time.

The Writeline and Readline commands will automatically open existing disk files or create new ones that need to be accessed, but will not close them until TurboCAD is terminated. If read/write functions need to be performed to more than four files, you must use the File command to close any unnecessary files, so another can be opened. See the Readline and Writeline commands for more information.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Function	= List or Open or Close or Rewind or Delete or Exist	1	List	%FF.F	=List
Filename	= filename	2	last	%FF.FI	=``

Definition of Options**Function =**

Determines the operation that is to be performed to the disk file. If a file function is successful, the Current Status Word %FF.Result is set to 1, otherwise it is set to 0.

List means the files that are currently open will be listed on your screen. The Filename option is ignored when this value is entered.

Open means the file specified in the Filename= option is opened and accessible for read and write operations. A maximum of four files can be open at one time.

Close means the specified file is closed and removed from the table of open files. Remember that if less than four files are open, the Readline and Writeline commands will automatically open any files they need.

Rewind means that the next line that will be read from will be the first line of the file. If this option is not used, each time the Readline command is executed it will read from the next line of the file called.

F Commands

Delete means the file is closed, removed from the table of open files and deleted from disk.

Caution: Use of this option can cause loss of data with no warning.

Exist checks to see if a file currently exists on disk. Remember that complete path names are important if the file in question is not in the current TurboCAD directory.

Filename =

Specifies the disk file that will be affected. If the filename contains spaces or special characters, it must be quoted (`like this` or \like this\).

Examples

File Close DoorSchedule

The file "DoorSchedule" will be closed and removed from the list of available files.

ff

A list of the files that are currently open for read/write will be displayed on your screen.

FF, FI=\Table A\, FUN=REW

The file "Table A" will be rewound so that the next time it is accessed by the Readline command, the first line will be read.

FILE EXIST `HD:FILES:MYFILE`

TurboCAD will check for the existence of the file "MYFILE" in the Hard Disk named HD inside the Folder named FILES. If it exists, the current status word %FF.Result will be set to 1. Otherwise, it will be set to 0.

Fillet or FL

2D

The Fillet command creates multi-segment lines with rounded corners as you create them. These are called Fillets in TurboCAD You will be prompted to locate corner points, or vertices, of the line. This command can also convert existing multi-segment Line primitives into Fillets. This command is complemented by the Fillet Existing command, which creates arcs (or "fits" them) between existing objects.

Pressing the <Esc> or <clear> key will allow you to change the current fillet radius. This allows a single fillet primitive to contain several different fillet radii. The <K> key will close the current fillet by returning to its starting point (origin). If the first and last point of a fillet are in the same location then the resulting corner will also be filleted.

Each Undo (<%> key) will remove the last point located. Fillets can contain a maximum of 84 segments.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Size	= number	1	last	%FL.S	= 1
Class	= 0 thru 255	2	%WK.C	%WK.C	= 0
Layer	= 0 thru 255	3	%WK.L	%WK.L	= 0
Pen	= 0 thru 255	4	%WK.P	%WK.P	= 0
STyle	= style	5	%WK.S	%WK.S	= Solid
PRiority	= 0 thru 255	6	%WK.PR	%WK.PR	= 125
WOrkset	= workset	7	%WK.W	%WK.W	=none
COnvert			none	%FL.CO	
Mimic			none	%FL.M	
coordinates			none		

Definition of Options

Size =

This option sets the size of the radius of the corners of the fillet. The Size value can be any non-negative number, and it is expressed in work units.

Class =

See **Common Options** for more information.

Layer =

See **Common Options** for more information.

Pen =

See **Common Options** for more information.

STyle =

See **Common Options** for more information.

PRiority =

See **Common Options** for more information.

WOrkset =

See **Common Options** for more information.

COnvert

This keyword-only option allows you to convert existing Lines on your drawing into Fillets. You will be prompted to pick an existing Line on your drawing instead of locating points to create the Fillet.

F Commands

Mimic

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Fillet Size =1/2

Filletts will be drawn with radiused corners of 1/2 work unit. Class, Layer, Pen, and Style options will default to the Status Palette values.

Fl %fl.s/2 p=5

Filletts will be drawn with radiused corners of one half the previously used size. The Fillet will be drawn in pen 5. Class, Layer, and Style options will default to the Status Palette values.

Fill Mi Co

You will first be prompted to pick an existing Fillet on your drawing, then to pick existing Lines that you would like converted into a Fillet. The Class, Layer, Pen, Style and Priority options will assume the values of the Fillet picked. The Size value will default to the last used value.

Fillet Existing or FLE

2D

The Fillet Existing command will "fit" an arc between two non-parallel lines, between two arcs or circles, or between a line and an arc or circle. This construction is commonly referred to as "Filleting". You will be prompted to select the two objects (lines or arcs/circles) between which the filleting operation is to be done. The endpoints of the resulting arc will be tangent to the objects selected, thus providing a "smooth" circular arc between them.

The radius of the resulting arc can either be specified by entering a number or by defining a point which will determine the radius. At the user's option, the lines or arcs/circles selected for filleting can be automatically "trimmed" or "extended" to result in a smooth circular transition between the two objects.

While in the process of selecting the two objects for filleting, each Undo (<%> key) will remove the last point located (or object selected). After the arc is created, the first two Undos (<%> keys) will "untrim" or "re-extend" each of the two objects, and the third will eliminate the arc and put you at the start of the command. In this way, you can "trim" one of the objects and not the other.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Size	= number or Points (or Pts) or None	1	Points	%FLE.S	= 1
Extend	= switch	2	Yes	%FLE.E	= Yes
Class	= 0 thru 255	3	%WK.C	%WK.C	= 0
Layer	= 0 thru 255	4	%WK.L	%WK.L	= 0
Pen	= 0 thru 255	5	%WK.P	%WK.P	= 0
STyle	= style	6	%WK.S	%WK.S	= Solid
PRiority	= 0 thru 255	6	%WK.PR	%WK.PR	= 125
WOrkset	= workset	7	%WK.W	%WK.W	=none
Mimic coordinates			none none	%FL.M	

Definition of Options**Size =**

This option sets the radius of the arc to be created by the filleting operation. The Size value can be any non-negative number, and it is expressed in work units.

Points (or **Pts**) means that the point located to select the first object will also be the endpoint of the arc created. TurboCAD will automatically compute the radius necessary to fit the arc with its point of tangency at that point.

None means that no arc will be created. The objects selected will be trimmed (or not trimmed) depending on the setting of the "Extend=" option.

Extend =

If this option is set to "On", the two objects selected for the filleting operation will be automatically "trimmed" or "extended" to the point of tangency with the arc created. If it is set to "Off", they will not be "trimmed" or "extended" and will remain the same. If not used, this option will always default to "On". Any other switch value can also be used as a value for this option. Switch values are: On, OFF, Yes, No, All, None, True, False.

Class =

See **Common Options** for more information.

Layer =

See **Common Options** for more information.

F Commands

Pen =

See **Common Options** for more information.

STyle =

See **Common Options** for more information.

PRiority =

See **Common Options** for more information.

WOrkset =

See **Common Options** for more information.

Mimic

When this option is used, you will be prompted to select an existing arc or circle on your drawing to determine the Size of the arc to be created. See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Fillet Existing Size =1/2

You will be prompted to select two objects on your drawing. If either of these objects is not a line or arc/circle, you will be asked if the object should be first converted into a line. An arc with a radius of 1/2 work unit will be "fit" between the two objects and will be tangent to the objects at its endpoints. The two objects will be trimmed or extended automatically. If the construction is not possible, an error message will be issued. Class, Layer, Pen, and Style options will default to the Status palette values.

FLE

You will be prompted to select two objects on your drawing as above. The point used to select the first object will be the point of tangency for the arc at that object. TurboCAD will calculate the radius of the arc to "fit" between the two objects and it will be tangent to the objects at its endpoints. The two objects will be trimmed or extended automatically. If the construction is not possible, an error message will be issued. Class, Layer, Pen, and Style options will default to the Status Palette values.

Fle Mimic

You will first be prompted to pick an existing Circle/Arc on your drawing, then to select two objects for filleting. The arc's Radius, Class, Layer, Pen, Style and Priority options will be the same as the values of the Circle/Arc picked.

The Font command attaches fonts to your drawing for use with the Text command. Only fonts that are so attached can be used in a drawing. The Text command will automatically attach a font which is not currently attached. The value of this command is that it can also remove and replace fonts. That is, once a drawing is created, you can globally replace one font for another without having to edit the text on the drawing.

Up to eight (8) fonts may be attached to each drawing. If a font is assigned to a position that is already in use, the new font will replace the existing one throughout the drawing. However, the new font will not be displayed until the drawing is reloaded or a "Display Regenerate" command is executed. TurboCAD automatically assigns the vector "default" font to position 0 upon startup.

Warning: Drawings containing bitmap (Macintosh) fonts cannot be transferred to a UNIX computer. You must first make all of the drawing stroked fonts.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Number	= 0 thru 7	1	0	%FT.N	=0
NAme	= name	2	%FT.NA.0	%FT.NA.n	= `default`

Definition of Options

Number =

Specifies to which of eight possible positions a font is to be assigned. Valid Number values are 0 through 7.

NAme=

Specifies the font name that is to be assigned to a position. The stroked fonts provided with TurboCAD are: Architect, Block, Capital, Computer, default, Poster, Roman, Roman1, Standard, Western.

Macintosh systems can also enter any of the bitmap font names that are in their system. Be sure to quote names that contain spaces or special characters ('like this' or \like this\).

Examples

Font Name=Capital Num=3

Assigns "Capital" font to position number 3. If a font was already assigned to position number three, it will be replaced with "Capital".

Ft, \Avant Garde\

Assigns "Avant Garde" font to position number zero. If a font was already assigned to position number zero, it will be replaced with "Avant Garde".

GAp or GA**2D**

The Gap command creates a gap (an invisible segment) in Lines and Doublelines. A Gap is just like any other segment of a multi-segment Line or Doubleline, except that it is invisible and thereby is not plotted or printed. Its vertices (or endpoints) are the vertices that the gap shares with the two visible segments it is between.

When a Gap is created in a Line or Doubleline, all of its segments are still part of the object, including the Gap segment. One of the useful features of Gaps is that they can be "deleted", or made visible again, via the Delete command. An example is when you insert a door symbol into a Doubleline. If you later remove the door, you can delete the Gap and retain your original Doubleline.

Other primitive types can also be gapped if they are first converted into a line. If you attempt to gap a primitive that is not a Line or Doubleline, you will be asked if you would like it converted into a Line primitive so it can be gapped.

Each Undo (<%> key) will remove the last gap created.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Size	= number or Points (or Pts)	1	Points	%GA.S	=Points
Pick	= switch	2	True	%GA.P	=True
Global	= global	3	%PF.G	%GA.G	=Ask
Convert			none	%GA.C	
coordinates		none	none		

Definition of Options**Size =**

Determines the length of the gap. If a Size value is entered, you will be prompted to locate one point on a line. The gap will be equally spaced around this point, and must fit within the segment picked. If this option is not used, or the Points value is entered, you will be required to locate two points on a line segment to define the size of the gap. You can make a gap that starts on one segment and ends on the next segment.

Pick =

When set to "True" (the default value), you have to select the object to be gapped first, then define the gap as above. When set to "False", the point you locate to define the gap also

G Commands

performs the selection. In congested drawings, it is often useful to "see" the object selected before gapping.

Global =

See **Common Options** for more information.

COnvert

This keyword-only option, when used, will force the conversion of the object selected into a line without asking you first.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Gap

You will be prompted to select a line or doubleline, then locate two points on the line or doubleline. A gap will be made between the two points.

Ga %db.w No

You will be prompted to locate a point on a line that is to be the center of the gap, whose length will be equal to the current Doubleline Width.

GAp Arc or GAA

2D

The Gap Arc command creates a gap in Arcs and Circles. Gaps are created by locating two points on the circle or arc. Gapping a circle converts it into an arc and gapping an arc breaks it into two arcs. Unlike gaps in Lines and Doublelines, gaps in circles and arcs cannot be made visible again via the Delete command. You can move the endpoint of an arc, however, via the Move Arc command.

Each Undo (<%> key) will remove the last gap created.

This command can also be entered as "GAP_Arc" or "GA Arc".

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
coordinates			none		none

Definition of Options

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

GA A

You will be prompted to locate two points on an arc or circle. A gap will be made between the two points.

GAp Edge or GAE

3D

The Gap Edge command creates a gap (an invisible segment) in Wireframes. Other primitive types can also be gapped with this command if they are first converted into a Wireframe. This command is the 3D complement to the Gap command.

This command, which gaps 3-dimensional objects, is very similar to the Gap command for 2D objects.

Each Undo (<%> key) will remove the last gap created.

This command can also be entered as "GAP_Edge" or "GA Edge".

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Size	= number or Points (or Pts)	1	Points	%GAE.S	=Points
Pick	= switch	2	True	%GAE.P	=True
Global	= global	3	%PF.G	%GAE.G	=Ask
coordinates			none	none	

Definition of Options

Size =

Determines the length of the gap. If this option is not used, or the Points value is entered, you will be required to locate two points on a wireframe edge to define the size of the gap. Used this way, you can make a gap that starts on one edge and extends around the corner to end on the next edge of a wireframe. If a Size value is given you will be prompted to locate one point on a wireframe. This point will be the center of the gap. Used this way, the entire gap length must fit in the wireframe edge selected.

Global =

This option determines how global changes to copied groups will be handled. Valid Global values are ASK, All, Some and Cancel. See the **Common Options** section for more information.

G Commands

Pick =

When set to "True" (the default value), you have to select the wireframe to be gapped first, then define the gap as above. When set to "False", the point you locate to define the gap also performs the selection. In congested drawings, it is often useful to "see" the object selected before gapping.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Gap Edge

You will be prompted to select an edge on a Wireframe, then locate two points to define the limits of the gap. A gap will be made in the wireframe between the two points.

GAE %db.w NO

You will be prompted to locate a point on a wireframe that is to be the center of the gap. The length of the gap will be equal to the current Doubleline Width. Only a single point will be required. This point will determine which wireframe edge is to be gapped as well as the location of the gap center.

GEt or GE

2D & 3D

The Get command retrieves symbols stored on disk so they may be placed on your drawing. Symbols can be rotated, scaled, mirrored and placed on a special class as they are retrieved.

Symbols are stored on disk using the Put command.

When a symbol is retrieved, can have it automatically attached to the crosshairs for placement on the drawing (depending on the "Placement=" option). Each time the 'esc' key is pressed the symbol attached to the crosshairs will rotate 90 degrees counterclockwise.

The 'E' key will end the current session of copies and get the same symbol from disk. This is most useful when symbols have Text Fields to be filled.

Each Undo (<%> key) will remove the last symbol placed on your drawing.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Symbol	= name	1	last	%GE.S	=none
Angle	= angle	2	0	%GE.A	=0
SCale	= number	3	1	%GE.SC	=1

						G Commands
Class	= 0 thru 255	4	none	%GE.C	=0	
ASsume	= 0 thru 255 or All or None	5	none	%GE.AS	=0	
Field	= switch	6	Yes	%GE.F	=Yes	
Repeat	= 0 thru 511	7	500	%GE.R	=500	
Placement	= Drag or Points (or Pts)	8	Drag	%GE.P	=Drag	
FOnt	= font	9	%TX.F	%GE.FO	=`default`	
Xscale	= number		1	%GE.X	=1	
Yscale	= number		1	%GE.Y	=1	
Mirror	= None or Xaxis or Yaxis or Both		None	%GE.M	=None	
	coordinates		none	none		

Definition of Options

Symbol=

Specifies the name of the symbol stored on disk that is to be retrieved. If the complete path to the file name is entered (For example, Get Symbol=`HD:Symbols:Door`), only the specified folder (directory) will be searched for the symbol. If only the file name is entered, and not the path, then the folders (directories) specified as the Current directory and the one specified for Symbols in the Path command will be searched.

Angle =

This option sets the angle which symbols will be rotated when retrieved. A valid Angle value is any number between -360 and 360. See the **Common Options** section for more information on angle values.

Scale =

This option sets the value which symbols will be scaled in both the X and Y axes when retrieved. The Scale value must be a number greater than zero. This option will override and reset the Xscale and Yscale options.

Class =

This option specifies the Class that symbols will be placed in when retrieved. If this option is not used, the symbol will retain the Class in which it was originally created. The Class value must be a number from zero through 255. See the Class command for more information on Classes.

G Commands

ASsume =

This option specifies the layer of the stored symbol that is to assume the current work layer. If this option is not used, the symbol will retain the Layers in which it was originally created. The Layer value must be a number from zero through 255 or the words "All" or "None". See the Work and Layer commands for more information.

Tip: The idea behind this option is to let you select a Layer that is your "unassigned layer" (I suggest Layer 0). This way symbols can be created on this unassigned layer when they need to be placed in different layers when retrieved. By creating symbols that contain both this unassigned layer along with assigned layers, you can have only part of a symbol assume the current work layer when retrieved.

Field =

This option determines if you would like to have an opportunity to fill the text fields of the retrieved symbol when you place it on your drawing. If a symbol contains multiple text fields, they will be presented to the operator in the order in which they were made a part of the group. The Field value must be a switch value. Switch values are: True or False, Yes or No, On or Off, All or None.

Repeat =

This option specifies the number of copies that will be made. The Get command is automatically terminated when the specified number of copies is reached.

PLacement =

Determines if the retrieved symbol is to be made visible for dragging. Valid Placement values are Drag and Points.

Drag means the symbol will be attached to the crosshairs at its origin for dragging to the desired location for placement.

Points means the symbol will not be visible for dragging. When a point is located the symbol will appear. The origin of the symbol will be placed at the point located. This value is most useful when the symbol is so large as to make dragging slow and awkward. Or to more quickly placed symbols when the location is known beforehand, like in a Macro.

FOnt =

This option specifies the Font that all text of the of the stored symbol will assume when retrieved. If this option is not used, the text will assume the current font. The Font value must be one of the provided TurboCAD stroked fonts or a bitmap font in your Macintosh system file. The provided stroked fonts are: Architect, Block, Capital, Computer, default, Poster, Roman, Roman1, Standard, Western.

Although it is possible to create a group on your drawing that contains multiple fonts, stored symbols can only contain one font. Therefore, if a group with multiple fonts is saved as a symbol to disk, they will all be the same font when the symbol is retrieved.

Xscale =

This option sets the value which symbols will be scaled in the X axis only. The Xscale value must be a number greater than zero. This option will work only if the "Scale=" option was not used.

Yscale =

This option sets the value which symbols will be scaled in the Y axis only. The Yscale value must be a number greater than zero. This option will work only if the "Scale=" option was not used.

Mirror =

This option sets the axis which symbols will be mirrored (flipped) about when retrieved. Valid Mirror values are None, Xaxis, Yaxis and Both.

None means the symbol will not be mirrored.

Xaxis means the symbol will be mirrored about the drawing X axis (flipped vertically).

Yaxis means the symbol will be mirrored about the drawing Y axis (flipped horizontally).

Both means the symbol will be mirrored about both axes (flipped vertically and horizontally).

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples**Get Door**

The folders (directories) set as the Current directory and the directory for Symbols in the Path command will be searched for a file named "Door." If found, the symbol will appear in the center of your screen attached to the crosshairs. You will be prompted to drag it to the desired location. Multiple copies of the symbol may be placed on your drawing. The symbol will retain the Class, Layers and Priority with which it was created. The symbol will not be scaled, rotated or mirrored. Any text that is part of the symbol will become the current font. If the symbols contains Text Fields, you will be prompted to fill them.

ge m=x a=-45 c=77 fie=n pl=p r=2

The folders (directories) set as the Current directory and the directory for Symbols in the Path command will be searched for the same file as was last retrieved. The symbol will not appear on your screen until a point has been located to place it. The Get command will be terminated after two copies have been placed. The symbol will retain the Layers and Priority with which it was created, but will be placed on Class 77. The symbol will not be scaled or mirrored, but will be rotated 45 degrees clockwise. Any text that is part of the symbol will become the current font. You will not be given an opportunity to fill any Text Fields that may be a part of the symbol.

GE X=4 Y=2 FIE=FA FO=STANDARD S=`SYM:VALVE` [R,4;]

The folders (directory) "SYM" will be searched for the file "VALVE." The symbol will be placed on your drawing four work units to the right of the last point located. The Get command will

then be terminated. The symbol will retain the Class, Layers and Priority with which it was created. The symbol will not be mirrored or rotated, but will be scaled 4 times horizontally and 2 times vertically. Any text that is part of the symbol will become `Standard` font. You will not be given an opportunity to fill any Text Fields that may be a part of the symbol.

Goto or GO

Macro

The Goto command redirects the execution of a Macro to the line beginning with a specified label.

Macros normally execute sequentially; that is, line by line. In addition to sequential execution, Macros can branch conditionally and unconditionally. The Goto command can skip to any line in a Macro that has a label.

Goto starts searching for the label beginning with the line in the Macro that follows the Goto statement and continues searching until it reaches the last line of the Macro. It then goes to the first line of the Macro and searches back down to the line that contains the original Goto statement. If the specified label is not found, an error message is issued and execution of the Macro is terminated.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Label	= label	1	none	%GO.L	= ``

Definition of Options

Label =

A label is an alphanumeric string ending with a colon (:). The label can contain only numbers, letters and underscores, and must begin with one of the letters A through Z. Spaces, commas, tabs and special characters (like \$%^) are not allowed. The label must be the first item on the line. Labels are not read as an executable part of the Macro, much the same way that comments are handled. Labels can be any length, but only the first 32 characters are significant. For example, the Macro interpreter will read the labels

`The_Beginning_of_Repeat_Section_1:` and `The_Beginning_of_

Repeat_Section_2:` as being valid but identical. Case is unimportant in a label. So, the labels "LOOP:", "Loop:" and "loop:" are equivalent.

Examples

Consider the following Macro:

You may not be familiar with some of the Macro commands used here, and this may not be the most efficient organization of a Macro, but the idea is to show the Goto command in action.

```
Prompt `Enter 'C' for Circle and 'S' for Square` ans
If ( Find ( Cc Subn (ans 1 1) ) ) Goto Circle:
If ( Find ( Ss Subn (ans 1 1) ) ) Goto Square:
Stop
Circle:          Circle Size=1
Stop
Square:         Rectangle Size=1
Stop
```

The operator is first prompted to enter a `C` or an `S` depending on the graphic primitive desired. The operator's response is remembered in the variable `ans`. Line two of the Macro checks the operator response to see if it is an upper or lower case C. If so, the Goto statement will branch to the line labeled "Circle:" and the Circle command will be executed. After exiting the Circle command, the Stop command on line six will terminate the Macro.

If no match is found on line two, execution of the Macro continues on line three and a similar check is performed. If neither check results in a match, the Stop command on line four will be executed, thereby terminating the Macro.

The Grid command displays a reference grid on your drawing. At this time, only one type of reference grid is available. Grids are viewed only on the screen, are not printed or plotted, and are independent of the Rounding value set by the Rounding command.

A TurboCAD grid is just like grid paper, or a set of orthogonal lines. It is composed of two grids superimposed on each other, the Small grid and the Large grid, each capable of being assigned a different color. This gives you the option of seeing grids just like gridded paper, on which, for example, grid lines are spaced 1/8" apart (the Small grid) and a bolder grid line (the Large grid) appears every whole inch.

The small (fine) grid lines and large (coarse) grid lines can be set to any spacing values. The X and Y axis values can be set independently. Besides setting the grid spacing, this command can be used to turn the display of the reference grids on and off. See Locating Points for the quick method of toggling grids on and off via the <G> keyboard key.

G Commands

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Small	= number or Half or Double or Rounding (or Rd)	1	last	%GD.S	=.25" (.1mm)
Large	= number or Half or Double or Rounding (or Rd)	2	last	%GD.L	=1" (1mm)
Type	= Full or Rulers or Isometric	3	last	%GD.T	=Full
Display	= switch	4	Yes	%GD.D	=Yes
Xsmall	= number r Half or Double or Rounding (or Rd)		last	%GD.X	=.25" (.1mm)
XLarge	= number or Half or Double or Rounding (or Rd)		last	%GD.XL	=1" (1mm)
Ysmall	= number or Half or Double or Rounding (or Rd)		last	%GD.Y	=.25" (.1mm)
YLarge	= number or Half or Double or Rounding (or Rd)		last	%GD.XL	=1" (1mm)

Definition of Options

Small =

Sets the distance between the small (minor) orthogonal grid lines for both axes. The Small value can be any number that is zero or greater, or the words Half, Double or Rounding. This options will override and reset the Xsmall and Ysmall options. This option sets the spacing for isometric type grids.

Half means the small grid spacing will be set to one half of the current value.

Double means the small grid spacing will be set to twice the current value.

Rounding means the small grid spacing will be set to twice the current Rounding value, as set by the Rounding command.

Large =

Sets the distance between the large (major) orthogonal grid lines for both axes. The Large value can be any number that is zero or greater, or the words Half, Double or Rounding, as described above. This options will override and reset the XLarge and YLarge options. This option is ignored for isometric type grids.

Type =

Sets the type of grids to be displayed. This feature is not currently operational - look for it in a future revision. Only Full type grids are supported.

Valid grid Type values are Full, Rulers and Isometric. Note that changing the grid type does not automatically change the Rounding values. If you are changing from orthogonal to isometric grids, or vice versa, be sure to set Rounding to the proper values.

Full means that a dot or line orthogonal grid will be displayed covering the entire screen.

Rulers means that large and small tick marks will be displayed along one horizontal and vertical edge of your screen.

Isometric means that a dot grid with lines at 30, -30 and 90 degrees will be displayed covering the entire screen. The Small option sets the spacing of isometric grids as measured along the 30 degree grid line. The Large, Xsmall, XLarge, Ysmall and YLarge options are ignored when isometric type grids are used.

Display =

This option makes the grids visible or invisible. The Display value must be a switch value. Switch values are: True or False, Yes or No, On or Off, All or None. The grid type and spacing values are remembered for when the grids are displayed again. Internally this is accomplished by setting the spacing values to a negative number. The numbers are automatically made positive when the grid display is activated. See Locating Points for the quick method of toggling grids on and off via the <G> keyboard key.

G Commands

Xsmall =

Sets the distance between the small (minor) orthogonal grid lines along the X axis. The Xsmall value can be any number that is zero or greater, or the words Half, Double or Rounding, as described earlier.

XLarge =

Sets the distance between the large (major) orthogonal grid lines along the X axis. The Xlarge value can be any number that is zero or greater, or the words Half, Double or Rounding, as described earlier.

Ysmall =

Sets the distance between the small (minor) orthogonal grid lines along the Y axis. The Ysmall value can be any number that is zero or greater, or the words Half, Double or Rounding, as described earlier.

YLarge =

Sets the distance between the large (major) orthogonal grid lines along the Y axis. The Ylarge value can be any number that is zero or greater, or the words Half, Double or Rounding, as described earlier.

Examples

Grid Small=.1 Large=.5 Type=Full

Grids will be displayed with the small reference grid lines a distance of 1/10 unit in both the X and Y axes. The large reference grid lines will be a distance of 1/2 unit in both axes.

GD RD 2

Grids will be displayed with the small reference grid lines a distance equal to twice the current Rounding value of both the X and Y axes. The large reference grid lines will be a distance of 2 units in both axes.

g d=n

Display of the grids will be turned off. The current grid type and spacing is remembered for when the grids are made visible again.

GROup or GP

2D & 3D

The Group command creates "groups" of objects by joining them together into one entity. Objects can be selected for grouping by picking them or by defining a box which contains them. Any combination of primitives and groups can be grouped together to form new groups. Groups can be tagged (named) for later counting with the Count and List commands.

Groups can be nested (groups inside a new higher level group). Each nested group is called a group Tier. Newly created groups can form a new tier or they can be grouped together on the

current tier. The way that new tiers are formed is by "locking" a group by tagging it (giving it a name). A tagged group cannot be added to. If other objects are grouped with a tagged group then a new (higher) group tier is automatically created.

Creating tiered groups allows you to Ungroup one tier at a time and retain the groups that made up a larger group. Remember that untagged groups, when grouped, become part of the same group tier and therefore lose their original group definition. This means that when a group tier is Ungrouped, all of the members of that group tier that are not tagged groups revert to primitives.

Groups can be tagged using the Tag option of this command or the Tag command itself. Group tiers are broken using the Ungroup command.

Groups can be given Attributes by having one or more pieces of text (which are defined as Attribute type text in the Text command) as part of the group. Groups, and their attributes, can be listed using the List command.

Groups which have been copied via the Copy command can have global changes made to all or some of the copies just by specifying the change on any one of the copies.

Each Undo (<%) key) will deselect the last picked object or box of objects to be grouped.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Tag	= name	1	none	%GP.T	=none
Select	= select	2	%PF.S	%GP.S	=Pick
Object	= object	3	%PF.O	%GP.O	=Any
Class	= 0 thru 255	4	%WK.C	%GP.C	=0
Priority	= 0 thru 255	5	%WK.PR	%GP.P	=125
coordinates			none		

Definition of Options

Tag =

Locks the group tier being created by naming it. The Tag value can be any alphanumeric string up to 10 characters long. Tags are case sensitive. If the string contains spaces or special characters it must be quoted (`like this` or \like this\). The tag is used for extracting lists of named objects on your drawing via the List command or counting via the Count command.

Select =

See **Common Options** for more information.

Object =

See **Common Options** for more information.

G Commands

Class =

See **Common Options** for more information.

Priority =

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Group Name=Chair

You will be prompted to pick the objects on your drawing that you wish to be joined together into a group called "Chair". In this set of examples let's create a group containing multiple tiers. Assume that you have picked two primitives to create the group "Chair", a line and an arc. The group will be created on the current Class and Priority.

GP DESK CL=40 PR=20

You will be prompted to pick the objects on your drawing that you wish to be joined together into a group called "DESK". Assume that you pick a rectangle to create the group. The group will be created on Class 40 and Priority 20.

gro,,in,,50

This group will be created by defining a rectangular area by locating its diagonally opposite corners. All objects completely inside the box will be made part of the group. Assume that you define a box that includes the previously created "Chair" and "DESK" groups along with several other primitives. The new group will contain two tagged groups on tier 1 and several untagged primitives on tier 2. The new group will not be tagged.

This means that more group members can conceivably be added to tier 2 of this group at a later date. The group will be created on Class 50 and the current Priority.

Help or HE

2D & 3D

The Help command displays on your screen the entire list of TurboCAD commands or the options of a specific TurboCAD command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Command	= name	1	none	%HE.C	=none

Definition of Options

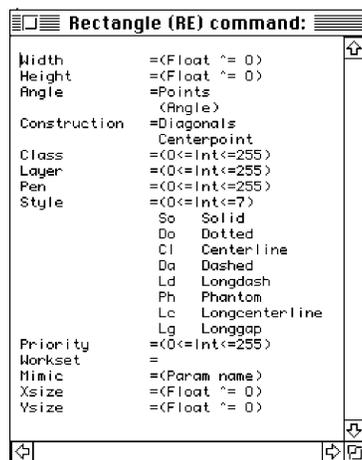
Command =

This option specifies which TurboCAD command you would like to get help on. A valid Command value is any 2D, 3D or Macro command name, abbreviation or code. If no command name is entered, you will see a list of all of the TurboCAD commands and their codes. See the Command List at the beginning of this manual for a complete listing of the TurboCAD commands.

Examples

Help Rect

The documentation for the Rectangle command will be displayed on your screen. Here is an example of what the help message would look like on a Macintosh computer. The first column contains the list of all of the options, in positional order, and the second column contains all of the valid entries for each option.



h,h

The documentation for the Help command (that's this one) will be displayed on your screen.

help

A list of all of the TurboCAD commands and their codes will be displayed on your screen.

IF or IF

Macro

The If command conditionally controls the execution of a Macro. A conditional statement is one that executes another statement only if a given condition is true. The Condition value is a relational expression as described below. It must be enclosed within parentheses. The Statement value is any TurboCAD graphics or Macro command. If the condition is true, the statement will be executed. If not, execution will continue on the next line of the Macro.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Condition	= expression	1	none	%IF.C	=False
Statement	= string	2	none	%IF.S	=None

Definition of Options

Condition =

This option defines the condition that must be met to execute the Statement. The Condition value is an expression enclosed in parentheses that must be written as:

(string1 compare string2)

where string1 and string2 are numeric or character expressions. The compare value must be one of the following relational operators:

<u>compare</u>	<u>Meaning</u>
<	Less than
<=	Less than or equal to
>	Greater than
>=	Greater than or equal to
==	Equal to
^=	Not equal to (inequality)

All of the relational operators have equal precedence, which is lower than the precedence for mathematical operators. This means that in a condition comparing one numeric expression with another, the expressions will be evaluated before their relationship is determined.

This type of condition is called a relational expression. If items compared in a relational expression are not of the same type - numbers compared to numbers and characters compared to characters - the condition will be false. For example, the number 123 is not the same as the character string `123`, and the relational expression (123 == `123`) will be false.

Relational expressions may be grouped together to form more complex conditions by comparing one expression to another. These more complex conditions are called logical expressions, and are written as:

(condition operator condition)

or (operator condition)

where condition is as described above and operator is a binary logical operator in the first case and a unary logical operator in the second case. Binary simply means that the operator acts on two conditions, whereas a unary operator acts on a single condition. These operators are explained below.

<u>Binary operator</u>	<u>Meaning</u>	<u>Description</u>
&&	And	Both conditions must be true for the whole condition to be true.
	Or	Either condition can be true for the whole condition to be true.
<u>Unary operator</u>	<u>Meaning</u>	<u>Description</u>
^	Not	If the condition was true, it's now false, and vice versa.

The unary operator has a higher precedence than the binary operators, but all logical operators have lower precedence than the relational operators.

Logical expressions may use parentheses to group conditions into specific relationships. For example, the expression:

(&Var1 == &Var2 && (&Var3 == 0 || &Var3 == 2))

will only be true if &Var1 is equal to &Var2 and if &Var3 is equal to either 0 or 2.

Statement =

This option specifies the string that is to be executed if the Condition is true. The Statement value can be any 2D or 3D TurboCAD graphics command, Macro command, Macro or calculation. The entire line from the end of the Condition to the end of the line will be read as the Statement.

Examples

Consider the following Macro:

```
Start: Prompt `Enter 1 for Circle and 2 for Square`,a,num
      If (a < 1 || a > 2) Goto Start:
      If (a == 2) Goto Square:
      Circle Size=1
```

H-K Commands

Stop

Square: Rectangle Size=1

Stop

You might not be familiar with some of the Macro commands used here, but the idea is to show the If statement in action. The Prompt command on line one prompts the operator to enter a 1 or a 2 depending on the graphic primitive desired. The operator's response is remembered in the variable "a". Line two checks the operator's response by comparing the number value remembered in the variable against the number 1 and 2. The If statement reads: If a is less than one or greater than two then continue execution at the line that begins with the label "Start:". If not, go to the next line in the Macro where the If statement says: If a is equal to two then continue execution at the line that begins with the label "Square:". If not, execute the next line in the Macro, which is the Circle command. The Stop command following the Circle or Square command will terminate the Macro.

Insert or IN

2D

The Insert command inserts groups into existing Lines and Doublelines on your drawing. In a single operation, the group is automatically rotated to the angle of the line and the line is gapped to accommodate the group. The inserted group can be picked from your drawing or can be a symbol retrieved from disk. After selecting the group, you define the point of insertion by locating a point on the Line or Doubleline. If you try to insert something into a primitive other than these, you will be asked if you would like the object converted into a Line so that it can be done.

Pressing the <Esc> or <clear> key after a group is inserted, but before you exit the command, will mirror (flip) the symbol first across and then along the line. This mirroring will be done about the group's origin.

If groups are being picked from your drawing for insertion, the <E> key will end the insertion of the current group and prompt you to pick a new group.

Each Undo (<%> key) will remove the last group inserted. The Undo buffer will remember the last 128 groups inserted with this use of the command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Symbol	= name	1	last	%IN.S	=none
SCale	= number	2	1	%IN.SC	=1
Distance	= number	3	none	%IN.D	=1

Gap	= number	4	none	%IN.G	=1
Field	= switch	5	none	%IN.F	=1
Xscale	= number	6	1	%IN.X	=1
Yscale	= number	7	1	%IN.Y	=1
coordinates			none		

Definition of Options

Symbol =

Specifies the name of the symbol stored on disk that is to be retrieved. If the complete path to the file name is entered (For example, Insert Symbol=`HD:Symbols:Door`), only the specified folder (directory) will be searched for the symbol. If only the file name is entered, and not the path, then the folders (directories) specified as the Current directory and the one specified for Symbols in the Path command will be searched. If no name is specified, you will be prompted to pick a group from your drawing for insertion.

Scale =

This option sets the value which symbols will be scaled in both the X and Y axes when inserted. The Scale value must be a number greater than zero. This option will override and reset the Xscale and Yscale options.

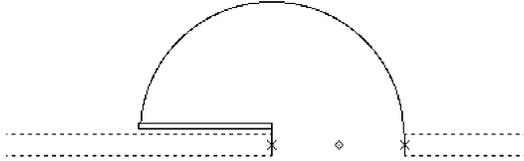
Distance =

This option sets the distance that the group will be inserted from the nearest vertex of the segment picked. The distance is measured from the vertex to the origin of the group being inserted. The Distance value must be a non-negative number. A value of zero means that the origin of the group will be placed at the nearest vertex of the segment picked. If this option is not entered, you will be prompted to drag the group to the desired location on a segment.

Gap =

Sets the size of the gap that will be made in the segment. The Gap value must be a non-negative number. If a gap size is entered, the gap is centered on the group origin. Entering a gap of zero means that no gap will be made in the segment. If this option is not entered, the size of the gap will be automatically calculated.

There is a way to control the size of the automatic gapping. When a symbol is selected for insertion, the program looks at the first member of the symbol (first one selected when creating the group) to see if it is a Point primitive. If it is, a gap will be made that is the horizontal distance (along the symbol's X axis) between the first two Point markers. The Point primitive must be the first object in the group and must contain only two markers. If the first object in the group is not a Point primitive the gap will be the overall width of the symbol (along the symbol's X axis). I suggest you use the "Point" type marker as it can usually be hidden under existing lines in the symbol. In the example below, we used the "X" marker for clarity. The diamond marker indicates the symbol origin.



Field =

This option determines if you would like to have an opportunity to fill the text fields of the retrieved symbol or picked group when you perform the insertion. If a symbol contains multiple text fields, they will be presented to the operator in the order in which they were made a part of the group. The Field value must be a switch value. Switch values are: True or False, Yes or No, On or Off, All or None.

Xscale =

This option sets the value to which symbols will be scaled in the X axis only. The Xscale value must be a number greater than zero. This option will not work if the "Scale=" option is used.

Yscale =

This option sets the value to which symbols will be scaled in the Y axis only. The Yscale value must be a number greater than zero. This option will not work if the "Scale=" option is used.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Insert F=No

You will first be prompted to pick a group from your drawing and then to drag it to the location on a segment you would like the group inserted into. The point picked on the segment is the location that the group will be inserted. The gap will be automatically calculated.

IN DOOR 1/2 3'0"

The folders (directories) set for the current directory and those set for Symbols in the Path command will be searched for a file named "DOOR." If found, the symbol will appear in the center of your screen attached to the crosshairs. You will be prompted to drag it to the segment you would like the group inserted into. The symbol will be inserted such that its origin is three feet from the nearest vertex of the segment picked. The symbol will be scaled to one half and a gap will be made in the line to accommodate the symbol at this new scale. You will be prompted to fill any unfilled text fields in the symbol.

Insert Node or INN

3D

The Insert Node command inserts nodes into 3-dimensional Wireframe primitives. A node in a 3-dimensional Wireframe is similar to a vertex in a 2-dimensional Line.

After picking an edge, locate a point for the new node. The original edge will be replaced with two new edges, each being drawn from the endpoints of the original edge to the new node location.

Since adding a node to a wireframe also adds a segment, and wireframes are limited to 64 edges, you may not be able to perform this operation to a wireframe that is at its limit. You can remedy this problem by using the Break command to break the wireframe into two or more Wireframe primitives each containing fewer edges.

Each undo (<%> key) will remove the last node inserted. The Undo buffer will remember the last 64 nodes inserted with this use of the command.

This command can also be entered as "INSERT_Node" or "IN Node".

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Global	= global	1	%PF.G	%INN.G	=Ask
coordinates		none			

Definition of Options

Global =

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Inn

You will first be prompted to pick an edge of a wireframe and then to locate a point for the new node. If there are multiple copies of the group you are altering, you will be asked if you want to make the same change to the other copies.

i n o

Same as the above example except that if there are multiple copies of the group you are altering, the node will only be inserted on the copy you picked.

Insert Vertex or INV

2D

The Insert Vertex command inserts (adds) vertices into Line, Doubleline, Fillet, Spline and Leaderline primitives. If this edit is performed on a primitive other than these, you may be asked if you would like the object converted into a line so the edit can be performed.

After picking a segment, locate a point for the new vertex. The original segment will be replaced with two new segments, each being drawn from the endpoints of the original segment to the new vertex location.

Since adding a vertex to a line also adds a segment, and lines are limited to 128 segments, you may not be able to perform this operation to a line that is at its limit. You can remedy this problem by using the Break command to break the line into two or more Line primitives each containing fewer segments.

Each undo (<%) key) will remove the last vertex inserted. The Undo buffer will remember the last 128 vertices inserted with this use of the command.

This command can also be entered as "INSERT_Vertex" or "IN Vertex".

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Global	= global	1	%PF.G	%INV.G	=Ask
coordinates			none		

Definition of Options

Global =

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

INv

You will first be prompted to pick a segment of a line and then to locate a point for the new vertex. If there are multiple copies of the group you are altering, you will be asked if you want to make the same change to the other copies.

i V o

Same as the above example except that if there are multiple copies of the group you are altering, the vertex will only be inserted on the copy you picked.

The Layer command sets the layer that you wish to work on. It also sets the layers that are to be displayed. The current work layer is the layer displayed on the Status palette and the one that new objects will be drawn on by default. You can only draw on one layer at a time. Any combination of layers can be turned on or off for display. Layers cannot be edited when they are not displayed.

The work layer can also be assigned via the Work command, and the layers to be displayed can also be assigned via the Display command. In those commands, you can also make assignments for Class, Pen, Style, and Priority at the same time. See these commands for more information.

There are 256 layers on each drawing, numbered 0 through 255. Every object on your drawing has a layer assignment, just as it has a Class, Pen, Style and Priority assignment. Layers differ from classes in that class is a characteristic of a group, where layer is a characteristic of a primitive. A primitive that has not yet been made part of a group is, for all intents and purposes, a group of only one item. Therefore, it has both a layer and class assigned to it. When several primitives are grouped together, they retain their individual layers, but all assume the same class.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Work	= 0 thru 255	1	last	%LA.W	=0
Display	= layers or All or None or Used	2	last	%LA.D.n	=All

Definition of Options

Work =

Sets the current work layer that is displayed on the Status palette at the bottom of your screen. The work layer is the default layer that new objects will be drawn on if the Layer option is not used in a command that creates graphic primitives. You must enter a whole number from 0 through 255.

Display =

Sets the layers that are displayed. A layer must be displayed in order for the objects on that layer to be editable. However, if an object which is displayed (i.e on a layer which is displayed) is on a class which is displayed but not editable, then the object is visible but not editable. This

is the only condition in which you can see something but not edit it. See the Class command to understand this difference between classes and layers.

Layers is a list of the layers that you wish to have turned on and off from the layers that are currently being displayed. There are 256 layers on each drawing, numbered 0 through 255. List the layers you wish to have turned on, separating each number with a space or comma. Ranges of layers can be entered by separating the starting layer and ending layer with an ellipsis (three periods, like this: 10...20). Individual layers and ranges of layers are turned off by preceding them with an up arrow (shift-6 on the keyboard, like this: ^).

Here is an example how you would turn on layers 23, 43, 100 through 200 for display, and turn off layers 150 through 175 for display: "DP Layers = 23 43 100...200 ^150...175".

All means that all layers will be turned on for display.

None means that all layers will be turned off for display, except the current layer. The layer you are working on must always be displayed.

Used means that just the layers that contain data will be displayed.

Examples

LAYER

Executing the Layer command this way will do nothing. All of the command options will default to last used, which will not change the Work, Display or Edit settings.

la 4 a

Your work layer will be set to four and all layers will be displayed.

Layer Display=None

All layers, except the work layer, will be made invisible. Only the visible work layer will be editable.

lay,,125, 200...220, 230...235, ^6, ^10...20

Layers 125, 200 through 220, and 230 through 235 will be made visible. Layers 6 and 10 through 20 will be made invisible.

LEaderline or LL

2D

The Leaderline command creates multi-segment lines with “leaders” at the start, end or both ends of the line. Leaders are predefined “terminators” provided with TurboCAD. They consist mainly of different arrowheads, boxes, dots and loops.

Each Undo (<%) key) will remove the last point located. The Undo buffer will remember the last 124 points located with this use of the command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Placement	= Start or End or Both or None	1	last	%LL.P	=none
Leader	= Arrow or Box or Closedarrow or Circle or Dot or DOwn or Loop or Openarrow or Slash or SQuiggle or SQUAre or Up or X	2	last	%LL.L	=Arrow
LENgth	= number	3	last	%LL.LEN	=.125"
Width	= number	4	last	%LL.W	=.0625"
Smart	= switch	5	Yes	%LL.S	=Yes
Class	= 0 thru 255	6	%WK.C	%WK.C	=0
LAYer	= 0 thru 255	7	%WK.L	%WK.L	=0

					L Commands
PEn	= 0 thru 255	8	%WK.P	%WK.P	=0
STyle	= style	9	%WK.S	%WK.S	=Solid
PRiority	= 0 thru 255	10	%WK.PR	%WK.PR	=125
WOrkset	= workset	14	%WK.W	%WK.W	=none
Mimic			none	%LL.M	
coordinates			none		

Definition of Options

Placement =

This option determines the placement of the leader on the line. Valid Placement values are Start, End, Both and None.

Start means the leader will be placed at the start of the multi-segment line.

End means the leader will be placed at the end of the multi-segment line.

Both means leaders will be placed at both the start and end of the line.

None means no leaders will be placed on the line.

Leader =

Selects the type of leader to be used. The valid leader types are:

→ Arrow	→ X	→ Down
→ Closed Arrow	→ ● Dot	→ Loop
→ Open Arrow	→ ○ Up	→ Square
→ Fold Arrow	→ ○ Circle	→ Slash
→ In Arrow	→ ■ Box	→ Squiggle

LENgth =

Sets the length of the leader in plot units. Plot units are units measured on the plotted paper. The size of leaders are not affected by the work units or drawing scale.

Width =

This option is similar to the Length option, except it sets the length of the leader.

Smart =

This option determines if the leaderline drawn will be a Leaderline primitive (smart) or just line segments (dumb). Smart leaderlines, like Smart dimensions, can be edited as single entities. Unless you somehow want to change the appearance of the leader itself, you are better off leaving leaderlines as Smart. The Smart value must be a switch value. Switch values are: True or False, Yes or No, On or OFF, All or None.

CLass =

See **Common Options** for more information.

L Commands

Layer =

See **Common Options** for more information.

Pen =

See **Common Options** for more information.

STyle =

See **Common Options** for more information.

PRiority =

See **Common Options** for more information.

WOrkset =

See **Common Options** for more information.

Mimic

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Leader End Slash

You will be prompted to locate points to draw a multi-segment line. A smart leaderline will be created with the “slash” leader placed at the end of the line. The Width and Length of the leader will default to the last used values. The Class, Layer, Pen, and Style values will default to the values shown on the Status palette.

ll p=b l=arrow len=1/10 w=1/2

You will be prompted to locate points to draw a multi-segment line. A smart leaderline will be created with the “arrow” leader placed at both ends of the line. The width of the leader will be one half unit and the length will be one tenth unit. The Class, Layer, Pen, and Style values will default to the values shown on the Status palette.

Line or LI

2D

The Line command creates new lines, continues existing lines on your drawing, or converts existing primitives on your drawing into lines. A line primitive can contain, at your option, multiple segments (i.e a "polyline") or individual segments.

TurboCAD multi-segment lines can contain invisible segments, called gaps. Pressing the <Esc> or <clear> key will put a gap in the line you are drawing by making the next segment invisible. The first segment cannot be a Gap segment. See the Gap command for information regarding putting Gaps into existing lines.

Each undo (<%> key) will remove the last point located. The Undo buffer will remember the last 124 points located with this use of the command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Function	= Multiple or Single or Individual or Continue or CONVert	1	Multiple	%LI.F	=Multiple
Class	= 0 thru 255	2	current	%WK.C	=0
Layer	= 0 thru 255	3	current	%WK.L	=0
Pen	= 0 thru 255	4	current	%WK.P	=0
Style	= style	5	current	%WK.S	=Solid
PRiority	= 0 thru 255	6	current	%WK.PR	=125
WOrkset	= workset	7	%WK.W	%WK.W	=none
Mimic			none	%LI.M	
coordinates			none	none	

Definition of Options

Function =

Sets the type of line that will be drawn. Valid Type values are Multiple, Single, Individual and Continue.

Multiple means the line drawn will contain multiple segments (up to 123). Use the <E> key to terminate one multi-segment line and start a new one.

Single means the line will be drawn just like a multi-segment line, but each segment is an independent primitive.

Individual means that two points are required to define each line segment. The segments are not touching end to end as in Single and Multiple type lines. Each line segment is an independent primitive.

Continue is the same as Multiple except that you will first be prompted to pick an existing line on your drawing that you wish to add more segments onto. A segment will rubberband from the end of the line picked and you can continue to draw, adding additional segments to the line.

CONVert means that you will be asked to select an existing object on your drawing to be converted into a line primitive. You must pick an item which is not already a line. Examples of

L Commands

this are rectangles, circles, ellipses, etc., which you want converted into multi-segment lines.
This function is also used to covert fillet and dimension primitives into their component parts.
Filletts are converted into lines and arcs, and smart dimensions are converted into lines,
leaderlines, and text.

Class =

See **Common Options** for more information.

Layer =

See **Common Options** for more information.

Pen =

See **Common Options** for more information.

Style =

See **Common Options** for more information.

PRiority =

See **Common Options** for more information.

WOrkset =

See **Common Options** for more information.

Mimic

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Line Style=Dotted

You will be prompted to locate points between which a multiple segment line will be drawn.
The style of the line will be dotted. The Class, Layer, Pen and Priority will default to the values shown on the Status palette.

LI conv P=4

You will be prompted to pick an existing object on your drawing which will be converted into a multi-segment line. The pen of the line will be set to four. The Class, Layer, and Style will default to the values shown on the Status palette.

l,c

You will be prompted to pick an existing line on your drawing, and then you will be prompted to locate points to add more segments to the end of the line.

LISt or LS**2D & 3D**

The List command lists tagged groups on your drawing. The list can be displayed on your screen or sent to disk as an ASCII text file. The list will always show the group tags and coordinates of the group's origin. It can optionally show any attributes, subgroups, point coordinates and line coordinates. Specific Classes of your drawing can be set to be searched, allowing you to "screen" your list by classes.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Tag	= tag	1	none	%LS.T	= none
TYPe	= Subgroup and/or Attribute and/or Point and/or Line	2	none	%LS.TY	= none
Classes	= classes	3	All	%LS.C.n	= All
Filename	= name	4	none	%LS.F	= none
Delimiter	= character	5	none	%LS.D	= none
coordinates			none		

Definition of Options**Tag =**

This option sets the top tier group tag that will be looked for and listed. A valid Tag value is any tag of an existing group on your drawing. If no Tag is entered all tagged groups will be listed. Remember that tags are case sensitive and limited to ten characters. If the tag contains spaces or special characters it must be quoted ('like this' or \like this).

TYPe =

This option determines the type of list that will be generated. Valid Type values are Subgroup, Attribute, Point and Line. The top tier group tags will always be listed along with the coordinates of the group origin. In addition to these, any or all of the values shown above can also be listed. Use the logical "and" symbol (&&) to string together multiple values. For example, the command " List Type=Att&&Sub" would list tags, attributes and subgroups for all tagged groups on your drawing.

Subgroup means the tags of groups which are "nested" within larger groups will also be shown. The second column of the list indicates which groups are subgroups of which. Zero

L Commands

indicates a top tier group; the number 1 is the tier immediately below the top tier, and so on. Top tier group tags are always listed, even if the Type option is not used.

Attribute means the attributes of each group, if any, will be listed.

Point means if a Point primitive is a member of a listed group then the X, Y and Z axis coordinates of each vertex will be listed.

Line means if a Line primitive is a member of a listed group then the X, Y and Z axis coordinates of each vertex will be listed.

Classes =

Sets the classes that will be searched for the specified tagged groups. Valid Classes values are:

classes is a list of the classes that you wish to have searched. There are 256 classes on each drawing, numbered 0 through 255. List the classes separating each number with a space or comma. Ranges on classes can be entered by separating the starting class and ending class with an ellipsis (three periods, like this: 10...20). Individual classes and ranges of classes are turned off by preceding them with an up arrow (keyboard shift-6: ^).

All means that all classes will be searched.

Edit means that just the classes that are currently editable will be searched.

Display means that just the classes that are currently displayed will be searched.

Used means that just the classes that contain data will be searched.

Filename =

This option specifies the name of the disk file that the list will be written to. If the file name exists, you will be asked if it is to be overwritten. Otherwise, a new file will be created. If this option is not used, the list will be displayed on your screen. If the filename contains spaces or special characters it must be quoted (^ like this` or \like this\).

Delimiter =

This option specifies the character that is to separate fields in a list. Any single character can be entered as the delimiter. The default is a columnized output.

coordinates

Points can be entered as coordinates on the command line for processing when the command is executed. See the **Coordinate Entry** section for more information.

Output Format

The Type option allows you output a list in different formats. An example of two lists is shown below. Here is an explanation of the fields of a list.

Field Explanation

1 A sequential item number. Only the top tier group is counted as an item. This number depends on the order that data is found in the drawing and may change after editing.

- 2 The group tier. A zero indicates the top tier group.
- 3 The Attribute number. A zero indicates the group itself.
- 4 The Class of the group or the layer of an attribute.
- 5 The group Tag (its name) or the attribute text.
- 6 The X axis location of the origin of the group.
- 7 The Y axis location of the origin of the group.
- 8 The Z axis location of the origin of the group.

Example of a list:

1	0	0	1	Chair	3.25000	6.50000	0.00000
2	0	0	45	Desk	4.50000	7.75000	0.00000
3	0	0	1	Chair	12.37500	23.00000	0.00000
4	0	0	10	File	14.00000	4.62500	0.00000
5	0	0	200	Office	25.00000	53.75000	0.00000

Example of a Type=Attributes list:

1	0	0	1	Chair	3.25000	6.50000	0.00000
1		1		Blue			
1		2		AcctDept			
2	2	0	45	Desk	4.50000	7.75000	0.00000
2	1	0	33	Phone	5.00000	8.00000	0.00000
2	1	1		555-6789			
2	2	0	30	Plant	8.50000	9.00000	0.00000
3	0	0	1	Chair	12.37500	23.00000	0.00000
3		1		Red			
3		2		AcctDept			
4	1	0	10	File	14.00000	4.62500	0.00000
5	0	0	200	Office	25.00000	53.75000	0.00000

Examples

List

A list of all the top tier of all tagged groups on all classes of your drawing will be displayed on your screen. Attributes will not be listed.

Ls Chair Att

A list of all top tier groups tagged "Chair" on all classes of your drawing will be displayed on your screen. Attributes will also be listed.

L Commands

ls,, a&&s&&p&&l,cl=200...220 f=Objects

A list of all tagged groups and their subgroups, attributes, point coordinates and line coordinates on classes 200 through 220 will be sent to a disk file named "Objects".

LOAd or LD

2D & 3D

The Load command opens a TurboCAD drawing stored on disk and displays it on your screen. Only one TurboCAD drawing can be loaded at a time. Therefore, attempting to load a drawing when one is already open will force the closure of the current drawing. If changes have been made to the current drawing since the last save, you will be given the opportunity to save your drawing.

If a limited number of Layers or classes were displayed the last time you saved the drawing, then only those will be displayed when you load it. All of the other information, however, is still there just as when you were last working on it. On a Macintosh computer, double clicking on a drawing icon first executes the TurboCAD application and then loads your drawing.

The "Open" command is a built-in alias for the Load command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Drawing	= name	1	none	%LD.D	=none
Force			none	%LD.F	

Definition of Options

Drawing =

Specifies the name of an existing TurboCAD drawing file on disk that is to be loaded into your workarea. If the complete path to the file is entered (For example, Load Drawing='HD:Drawings:Schematic'), only the specified folder (directory) will be searched for the drawing. If only the file name is entered, and not the path, then the folders (directories) specified as the Current directory and the one specified for Drawings in the Path command will be searched. File names containing spaces or special characters should be quoted ('like this' or \like this\) so they are not seen as separate options to the command.

Force

This option forces the clearing of the existing drawing by bypassing the safety question. This option is most commonly used in Macros where operator intervention is not desired.

Warning: Use of this option can cause loss of data with no warning.

Examples

Load \Floor Plan\

The file "Floor Plan" will be searched for in the folders (directories) specified for Drawings in the Path command. If found, the current drawing will be closed and the drawing "Floor Plan" will be loaded into your workarea. If the current drawing has not been saved, you will be given the opportunity to do so.

ld `HardDisk:Drawings:Detail A` f

The file "Detail A" will be searched for in the folder "Drawings" on volume "HardDisk". If found, the current drawing will be closed and the drawing "Detail A" will be loaded into your workarea. Because the Force option was used, you will not be given an opportunity to save the current workarea. Using the Force option is extremely dangerous, so I don't recommend it.

L0okup or LU

2D & 3D

The Lookup command looks up and displays all of the properties of an object on your drawing. Object properties are things like Class, Style, Size, Width, Font, etc.

Properties come in two flavors: Group properties and Primitive (or Member) properties. Group properties are Class, Priority, Tag, X & Y Location (origin), X & Y Scale and Rotation angle. These properties are displayed in the first paragraph of information displayed when you do a Lookup on an object. The Primitive properties vary with the type of primitive picked. Lines, for example, have different properties than text. The Primitive properties are shown in the second paragraph of information.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Object	= object	1	%PF.0	%LU.0	=Any
Silent			none	%LU.S	
Nohighlight			none	%LU.N	
coordinates			none		

Definition of Options

Object =

This option determines the type of objects that can be affected by this command. Valid Object values are Any, ARcs, Boundaries, Circles, Copies, DIMensions, Doublelines, Ellipses, Fillets, Groups, Lines, POints, POLygons, Primitives, Rectangles, Splines, SURfaces, Text and Wireframes. See the **Common Options** section for more information.

L Commands

Silent

Suppresses the display of the properties to your screen. This option is useful in a Macro when you want the Lookup Current Status Words to contain the properties of an object, but not display them on the screen.

Nohighlight

Suppresses the highlighting of the object that is picked. This option is useful in a Macro when you want the Macro to run without any feedback to the operator.

coordinates

Points can be entered as coordinates on the command line for processing when the command is executed. See the **Coordinate Entry** section for more information.

Current Status Words

Here is a list of the Current Status Words that are set when an object is picked. Not all CSWs are set by each object and therefore may contain a value from a previously picked object. Only the CSWs set when you pick an object are displayed by the Lookup command.

<u>Group Properties</u>	<u>CSW</u>
Gobject	%LU.G
Tag	%LU.
Class	%LU.C
Priority	%LU.P
Rotate	%LU.R
GXscale	%LU.GX
GYscale	%LU.GY
GXOrigin	%LU.GXO
GYOrigin	%LU.GYO
Gzorigin	%LU.GZ
GXMaximum	%LU.GXM
GYMaximum	%LU.GYM
GXMinimum	%LU.GXM
GYMinimum	%LU.GYMI
GXSize	%LU.GXSI
GYSize	%LU.GYSI
Members	%LU.M
COpies	%LU.CO
Tlrs	%LU.TI

Member Properties	CSW
Mobject	%LU.MO
Mtag	%LU.MT
MTler	%LU.MTI
Layer	%LU.L
Pen	%LU.PE
Style	%LU.ST
Cap	%LU.CA
Dimension	%LU.D
Extension	%LU.E
EXTStyle	%LU.EXTS
Fill	%LU.F
GAp	%LU.GA
Justification	%LU.J
Leader	%LU.LE
Llength	%LU.LL
Lwidth	%LU.LW
Height	%LU.H
Width	%LU.W
Theight	%LU.TH
Twidth	%LU.TW
Horizontal	%LU.HO
Vertical	%LU.V
Font	%LU.FO
FORmat	%LU.FOR
Major	%LU.MA
Minor	%LU.MI
MARker	%LU.MAR
Offset	%LU.OF
Overrun	%LU.OV
Placement	%LU.PL
ROUnding	%LU.ROU
Show	%LU.SH

L Commands

SIZE	%LU.SIZ
Angle	%LU.A
STArting	%LU.STA
Included	%LU.I
Slant	%LU.SL
Attribute	%LU.AT
FIeld	%LU.FIE
Spacing	%LU.SP
Type	%LU.TY
Units	%LU.U
COUnt	%LU.COU
Segment	%LU.SE
VERTEx	%LU.VERTE
Xorigin	%LU.X
Yorigin	%LU.Y
Zorigin	%LU.Z

Examples

Lo

You will be prompted to pick an existing object on your drawing. Its properties will be displayed on your screen and remembered in the appropriate Current Status Words.

LU S S

You will be prompted to pick an existing object on your drawing. Only splines will be accepted. The spline's properties will not be displayed on your screen. However, its properties will still be remembered in the appropriate Current Status Words.

MAcro or MA**2D & 3D**

The Macro command executes existing macro files residing on disk. For more information on what macros are and how to use them, see the **Macro Language** section. This command only executes existing macros, it does not write them.

The slash character (/) is a built-in alias for the Macro command when it appears at the start of the command line. For example, the command "/Door", followed by a "Return" would execute the macro called "Door".

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Filename	= name	1	none	%MA.F	=none
Arguments	= string	2	none	%MA.A	=none

Definition of Options**Filename =**

Specifies the name of an existing TurboCAD Macro file on disk that is to be executed. If the complete path to the file is entered (For example, Macro Filename=HD:Macros:Stairs`), only the specified folder (directory) will be searched for the file. If only the file name is entered, and not the path, then the folders (directories) specified as the Current directory and the one specified for Macros in the Path command will be searched. File names containing spaces or special characters should be quoted (`like this` or \like this\) so they are not seen as separate options to the command.

Arguments =

Arguments are used in programming languages to "pass" values from one program to another or to include values which will be used in the program. Macros are like programs written in the TurboCAD command language. Up to nine (9) arguments can be passed to the macro being executed. The arguments are separated with spaces. This option will read the contents of the command line until the end of line if no quotes (`like this` or \like this\) are used. The arguments are placed in the variables %Arg1 through %Arg8. The variable %Arg0 will contain the number of arguments that were found. If there are more than eight arguments on the line, the contents of the line from the start of the ninth argument through the end of the line will be placed in %Arg9, and %Arg0 will be set to 9.

Examples**Macro Door**

The file "Door" will be searched for in the folders (directories) specified for macros in the Path command. If found, it will be executed as a macro.

M Commands

ma `HardDisk:Macros:Text` %TX.Height %TX.Font

The file "Text" will be searched for in the folder "Macros" on volume "HardDisk". If found, it will be executed as a macro. The current text height and font values will be placed in the variables %Arg1 and %Arg2, respectively. The variable %Arg0 will be set to 2, because two arguments were set.

Merge or MG

2D & 3D

The Merge command retrieves a TurboCAD drawing from disk and merges it with your current drawing. At your option, you can have the origin of the drawing being merged match the origin of the drawing you are merging on to. Since you can relocate the origin of the drawing currently active (via the Origin command), this means you can define the location of the drawing merged. You can also have the merged drawing match the scale of the current drawing. If the operation of merging would result in a larger drawing than the current drawing, TurboCAD will automatically enlarge the size of the drawing. The merged drawing can be placed in a single class if desired. This is handy for creating a "reference" drawing.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Drawing	= name	1	none	%MG.D	=none
Class	= All or 0 thru 255	2	All	%MG.C	=All
Origin			none	%MG.O	
Scale			none	%MG.S	

Definition of Options

Drawing =

Specifies the name of an existing TurboCAD drawing file on disk that is to be merged into your workarea. If the complete path to the file is entered (For example, Merge Drawing=`HD:Drawings:Schematic`), only the specified folder (directory) will be searched for the file. If only the drawing name is entered, and not the path, then the folders (directories) specified as the Current directory and that specified for Drawings in the Path command will be searched. Drawing names containing spaces or special characters should be quoted (`like this` or \like this\) so they are not seen as separate options to the command.

Class =

Sets the class of your current workarea that the drawing will be merged into. The Class value must be a number from 0 through 255 or the word "All". Entering a number means that the

M Commands

entire drawing being merged will be placed in that class of your drawing. All means each class of the merged drawing will be placed on the equivalent class of the current workarea. If this option is not used, it defaults to "All".

Note: As mentioned above, merging a drawing into a specific class can be handy for referencing another drawing against the one you are working on. You can designate a "reference" class by turning it off for edit. When saving your current drawing, this reference class can be stripped from the workarea by using the Classes option of the Save command.

Origin

This keyword-only option, when used, will cause the origin of the drawing being merged to match the origin of the current drawing. If not used, the lower left hand corners of both drawings will be matched.

Scale

This keyword-only option, when used, will cause the drawing being merged to assume the scale of the current drawing.

Examples

Merge \Floor Plan\

The file "Floor Plan" will be searched for in the current directory (folder) and the directory specified for Drawings in the Path command. If found, it will be merged, class for class, into the current workarea.

mg `HardDisk:Drawings:Detail A` 12

The file "Detail A" will be searched for in the folder "Drawings" on volume "HardDisk". It will be merged into the current workarea in class twelve.

MESSage or MS

Macro

The Message command displays a message in the News Area. The message can be displayed for a fixed period of time or until the operator acknowledges it. On Macintosh systems, the News Area is a dialog box.

Messages are most useful in a macro when you are trying to keep the operator informed of what's going on.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Statement	= string	1	last	%MS.S	=none
Time	= 0 thru 360	2	last	%MS.T	=0

Definition of Options

Statement =

This option defines the message that the operator will see. A valid Statement value is any alphanumeric string up to 82 characters. Be sure to quote the string if it contains spaces or special characters (like this` or \like this\` so it will be read correctly. If you want variable substitution to be performed, then the name of the variable or current status word must be between "back slashes" (\like this\). Any currently defined variables or any valid current status word will show the actual value in this case.

Time =

Sets the time, in seconds, that the message will remain displayed on the screen. A valid Time value is any whole number from 0 through 360. Zero seconds means the message will remain displayed until the operator acknowledges it by pressing the mouse button or any keyboard key.

Examples

Message \Now calculating the distance...\ 5

The message "Now calculating the distance..." will be displayed in the News Area for five seconds.

ms s=`Press any key to continue.`

The message "Press any key to continue." will be displayed in the News Area until the operator acknowledges it by pressing a key.

ms \a, b, c, %tx.h, %tx.font\ t=0

The current values of the Global variables a, b, and c along with the current text height and current font will be displayed until the operator acknowledges it by pressing a key.

Mirror or MR

2D

The Mirror command flips objects on your drawing so to produce mirror images of them. Objects are mirrored about a mirroring axis of any angle that you can define by locating two points. The objects to be mirrored can be selected by picking them or by defining a box which contains them. The pivot point for mirroring can be a common point for all objects selected or can be a point on each individual object. At your option, this command can copy the objects selected as well as mirror them, leaving the original objects selected intact.

This command will mirror only ungrouped primitives or whole groups. Members of a group cannot be individually mirrored. Text that is part of a mirrored group will remain right

reading. Once a group is mirrored, it is a unique group; it is no longer a copy of other similar groups.

Each undo (<%> key) will return the last mirrored object or box of objects to its original orientation. The Undo buffer will remember the last 128 objects mirrored with each use of the command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Axis	= Points (or Pts) =Points or Xaxis or Yaxis or Both	1	Points	%MR.A	
Select	= select	2	%PF.S	%MR.S	=Pick
Pivot	= Axis or Point (or Pt) or Plck or Object	3	Axis	%MR.P	=Axis
Object	= object	4	%PF.O	%MR.O	=Any
Copy			none	%MR.C	
coordinates			none		

Definition of Options

Axis =

Sets the angle of the axis about which objects will be mirrored. Valid Axis values are:

Points (or **Pts**) means you will be prompted to locate two points to define the angle of the mirroring axis.

Xaxis means objects will be mirrored about the drawing X axis (flipped vertically).

Yaxis means objects will be mirrored about the drawing Y axis (flipped horizontally).

Both means objects will be mirrored about both axes (flipped both horizontally and vertically).

Select =

See **Common Options** for more information.

Pivot =

Sets the point that objects will be flipped about. Valid Pivot values are:

M Commands

Axis means that the pivot point is the on the Axis itself, provided that it was defined by locating two points.

Point (or **Pt**) means you will be prompted to locate a common pivot point for all objects selected to be mirrored.

Pick means objects will be mirrored about the point that they are picked. If objects are being selected by box, the pivot point is the center of the box.

Object means all selected objects will be mirrored about their individual origins.

Object =

See **Common Options** for more information.

Copy

This keyword-only option, when used, creates mirrored copies of the selected objects and leaves the originals intact. You get the idea.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Mirror

You will first be prompted to locate two points to define the mirroring axis and then to pick the objects you would like mirrored. The pivot point will be on the mirroring axis you defined.

MI Y A O

All objects on your drawing will be flipped horizontally about their own origin.

mr copy pivot=point axis=x select=touch

All objects that are touching a box that you define by locating its diagonally opposite corners will be vertically copied as mirror images about a common point.

mir, bot,in,pt

All objects that are completely inside a box that you define by locating its diagonally opposite corners will be mirrored about both axes pivoting about the center of the box.

MODifytext or MT

The Modifytext command allows you to change the wording of text on your drawing. Pick the text you wish to change and it will appear in a scrollable dialog box to be edited.

Once the dialog box appears the text can be edited by using standard Macintosh editing tools, such as Cut, Copy, and Paste either using the Edit pull-down menu or keyboard equivalents. Additional text, from an existing text file on disk, can be appended to the text in the dialog via

the "Read File" button. The text in the dialog box can be saved as a text file for future use via the "Save as" button. Hitting the "OK" button places the edited text back onto your drawing.

Another use for the Modifytext command (by using the "Filename=" option) is to create and edit text files, such as macros. These files are independent of the drawing you are working on.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Filename	= name	1	none	%MT.F	=none
Global	= global	2	%PF.G	%MT.G	=Ask
coordinates		none	none		

Definition of Options

Filename =

This option will read from an ASCII text file on disk. The contents of the file will be placed in the Modifytext buffer for modification. If you enter the file name of a file which does not exist, TurboCAD will automatically create a new empty file with that name. Once modified, it will be written back to the file on disk. If this option is not used, you will be prompted to pick a paragraph of graphic text on your drawing for modification.

If the complete path to the filename is entered (For example, Modifytext Filename= `HD:Macros:Door`), only the specified folder (directory) will be searched for the filename. If only the filename is entered, and not the path, then the folders (directories) specified as the Current directory and the one specified for macros in the Path command will be searched.

Global =

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Modify

You will be prompted to pick the graphic text on your drawing that you wish to change. It will appear in a scrollable dialog box where it can be edited and placed back onto the drawing or into a disk text file.

MT \Door Schedule\

Text will be read from the disk file "Door Schedule" and placed in the Modifytext buffer where it can be edited and placed back onto the disk file or onto the drawing. If no such file is found in the Current directory or the directory specified for Macros in the Path command, a new empty file named "Door Schedule" will be created.

Move or MV

2D & 3D

The Move command relocates objects on your drawing. The objects to be moved can be selected by picking them or defining a box which contains them. The location for the move is determined by dragging the object, entering coordinates, or locating two points to define the distance and angle.

Each undo (<%) key) will remove the last object or box of objects moved. The Undo buffer will remember the last 128 objects moved with this use of the command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Xdistance	= number	1	none	%MV.X	=0
Ydistance	= number	2	none	%MV.Y	=0
Zdistance	= number	3	none	%MV.Z	=0
Select	= select	4	%PF.S	%MV.S	=Pick
Object	= object	5	%PF.O	%MV.O	=Any
Vector			none	%MV.V	
Distance	= number	6	none	%MV.D	=0
Angle	= angle	7	none	%MV.A	=0
coordinates			none		

Definition of Options

Xdistance =

Sets the distance that objects will be moved along the X axis (horizontally). A positive X axis value is to the right, while a negative value is to the left. Entering a value for this option will cause an error to be issued if you are using the Vector option or the Distance and Angle options below.

Ydistance =

Sets the distance that objects will be moved along the Y axis (vertically). A positive Y axis value is up, while a negative value is down. Entering a value for this option will cause an error to be issued if you are using the Vector option or the Distance and Angle options below.

Zdistance =

Sets the distance that objects will be moved along the Z axis of the Viewplane. A positive Z axis value is toward you, while a negative value is away from you. Entering a value for this option will cause an error to be issued if you are using the Vector or the Distance and Angle options

below. This value will be ignored if you select objects which are 2D objects and will affect only 3D wireframes.

Select =

See **Common Options** for more information.

Object =

See **Common Options** for more information.

Vector

This keyword-only option allows the distance and angle of the move to be specified by locating two points with the crosshairs. An error will be issued if this option is used at the same time as the X, Y, or Z distance options or the Distance and Angle options below.

Distance =

This option, along with the Angle option below, allows you to specify the distance vector numerically. Any positive or negative distance value is valid. An error will be issued if this option is used at the same time as the X, Y, or Z distance options or the Vector option.

Angle =

This option works with the Distance option to specify the distance vector numerically. An error will be issued if this option is used at the same time as the X, Y, or Z distance options or the Vector option. See **Common Options** for more information on entering angles.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples**Move**

Objects will be moved one at a time by picking them, dragging them to the desired location, and locating a point to place them.

m vec

You will first be prompted to locate two points to define the distance and angle that objects will be moved. You may then pick the objects to be moved this distance.

MV Y=2.5 S=ALL OB=R

All Rectangle primitives on your drawing will be moved up two and one half units.

mo se=in d=2 a=30

You will first be prompted to define a box by locating its diagonally opposite corners. All objects completely inside the box will be moved a distance of 2 work units at an angle of 30 degrees. Several boxes can be defined to move objects that may be in an irregularly shaped area.

M Commands

M,3,2

Every object you pick with the crosshairs will be moved a distance of 3 work units in the X and a distance of 2 work units in the Y.

Move Arc or MVA

2D

The Move Arc command allows you to change the included angle of an arc by moving either arc endpoint. You will be prompted to pick the desired endpoint of an arc and drag it to a new location. Moving the arc endpoint past the point of a complete circle will result in an arc with an included angle equal to the overlap.

This command will change only the included angle of an arc, not its radius (size). The included angle of an arc, its radius, etc. can also be changed numerically via the Change and Change Object commands. See these commands for more information.

This command can also be entered as "MOVE_Arc" or "MV Arc".

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Global	= global	1	%PF.G	%MVA.G	=Ask
coordinates			none		

Definition of Options

Global =

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

mva

You will be prompted to pick an arc near the endpoint you wish to move. A line will rubberband from the center of the arc. The intersection of the arc and this line will be the location of the new arc endpoint.

Mov Ar

Same as the last example, only typed differently.

Move Edge or MVE

3D

The Move Edge command relocates edges of 3-dimensional wireframe primitives similarly to how the Move Segment command moves line segments. Edges can be moved in any or all of the three axes either in the Viewplane or World coordinate systems.

Each undo (<%)> key) will remove the last edge moved. The Undo buffer will remember the last 128 edges moved with this use of the command.

This command can also be entered as "MOVE_Edge" or "MV Edge".

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Xdistance	= number	1	none	%MVE.X	=0
Ydistance	= number	2	none	%MVE.Y	=0
Zdistance	= number	3	none	%MVE.Z	=0
System	= Viewplane or World	4	Viewplane	%MVE.S	=Viewplane
Global	= global	5	%PF.G	%MVE.G	=Ask
Vector coordinates			none none	%MVE.V	

Definition of Options

Xdistance

Sets the X axis distance that the selected edges will be moved. The Xdistance value can be any positive or negative number, and is mutually exclusive of the Vector option. The System option sets the type of coordinate system to be used, which determines the X axis orientation.

Ydistance

Sets the Y axis distance like the Xdistance option above.

Zdistance

Sets the Z axis distance like the Xdistance option above.

System =

Sets the type of coordinate system that will be used. Valid System values are Viewplane and World. See the **Common Options** for more information.

Global =

See **Common Options** for more information.

M Commands

Vector

This keyword-only option requires you to pre-define the distance that edges are moved by locating two points on your drawing. This option is mutually exclusive of the Xdistance, Ydistance and Zdistance options.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Move Edge

Edges will be moved one at a time by picking and dragging them.

mve d

You will be prompted to locate two points to define the distance that edges will be moved. You may then pick the edges to be moved this distance.

MV E Y=5+1/2 S=W

Pick the edges you wish to have moved a positive five and one half units along the Y axis of the World coordinate system.

Move Member or MVM

2D

The Move Member command relocates primitives that are members of a group.

Each undo (<%> key) will return the last moved member to its original location. The Undo buffer will remember the last 128 members moved with this use of the command.

This command can also be entered as "MOVE_Member" or "MV Member".

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Xdistance	= number	1	none	%MVM.X	=0
Ydistance	= number	2	none	%MVM.Y	=0
Object	= object	3	%PF.O	%MVM.O	=Any
Global	= global	4	%PF.G	%MVM.G	=Ask
Vector			none	%MVM.V	
Distance	= number		none	%MVM.D	=0
Angle	= angle		none	%MVM.A	=0
coordinates			none		

Definition of Options

Xdistance =

Sets the distance that members will be moved along the X axis (horizontally). A positive X axis value is to the right, while a negative value is to the left. Entering a value for this option will cause an error to be issued if you are using the Vector option or the Distance and Angle options below.

Ydistance

Sets the Y axis distance like the Xdistance option above.

Object =

See **Common Options** for more information.

Global =

See **Common Options** for more information.

Vector

This keyword-only option allows the distance and angle of the move to be specified by locating two points with the crosshairs. An error will be issued if this option is used at the same time as the X or Y distance options or the Distance and Angle options below.

Distance =

This option, along with the Angle option below, allows you to specify the distance vector numerically. Any positive or negative distance value is valid. An error will be issued if this option is used at the same time as the X or Y distance options or the Vector option.

Angle =

This option works with the Distance option to specify the distance vector numerically. An error will be issued if this option is used at the same time as the X or Y distance options or the Vector option. See **Common Options** for more information on entering angles.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Move Member

Group members will be moved one at a time by picking them, dragging them to the desired location and locating a point to place them. If this change is being made to a copied group, you will be asked if the change should be made to the other copies.

m m v

You will first be prompted to locate two points to define the distance and angle that members will be moved. You may then pick the members to be moved this distance. If this change is being made to a copied group, you will be asked if the change should be made to the other copies.

M Commands

MVM X=2 Y=-1 Glob=A

The group members that you pick will be moved one unit down and two units to the right. Copies of this group will all automatically be changed the same way.

Move Node or MVN

3D

The Move Node command relocates nodes of 3-dimensional wireframe primitives similarly to how the Move Vertex command moves 2-dimensional vertices. Nodes can be moved in any or all of the three axis either by the Viewplane or World coordinate systems.

Each undo (<%> key) will remove the last node moved. The Undo buffer will remember the last 128 nodes moved with this use of the command.

This command can also be entered as "MOVE_Node" or "MV Node".

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Xdistance	= number	1	none	%MVN.X	=0
Ydistance	= number	2	none	%MVN.Y	=0
Zdistance	= number	3	none	%MVN.Z	=0
System	= Viewplane or World	4	Viewplane	%MVN.S	=Viewplane
Global	= global	5	%PF.G	%MVN.G	=Ask
Vector coordinates			none none	%MVN.V	

Definition of Options

Xdistance

Sets the X axis distance that the selected nodes will be moved. The Xdistance value can be any positive or negative number. This option is mutually exclusive of the Vector option. The System option sets the type of coordinate system to be used, which determines the X axis orientation.

Ydistance

Sets the Y axis distance like the Xdistance option above.

Zdistance

Sets the Z axis distance like the Xdistance option above.

System =

Sets the type of coordinate system that will be used. Valid System values are Viewplane and World. See the **Common Options** for more information.

Global =

See **Common Options** for more information.

Vector

This keyword-only option requires you to pre-define the distance that nodes are moved by locating two points on your drawing. This option is mutually exclusive of the Xdistance, Ydistance and Zdistance options.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples**Move Node**

Nodes will be moved one at a time by picking and dragging them.

mvn,v

You will be prompted to locate two points to define the distance that nodes will be moved. You may then pick the nodes to be moved.

MVN X=1/2 S=W

Pick the nodes you wish to have moved a positive one half unit along the X axis of the World coordinate system.

Move Parallel or MVP**2D**

The Move Parallel command moves and reshapes objects such that all segments are parallel to, and equidistant from, the original object. All primitives except Text, Smart Dimensions, and Pattern Fills (Boundaries) can be copied in parallel. The resulting primitive will be of the same type as the original. Objects are selected by picking them. Primitives picked which are part of a group will be moved in parallel and will continue to be part of the group. The move distance can be specified by locating a point or entering a distance.

Each undo (<%> key) will return the last moved object to its original location. The Undo buffer will remember the last 128 objects moved with this use of the command.

This command can also be entered as "MOVE_Parallel" or "MV Parallel".

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Distance	= number or Points (or Pts)	1	Points	%MVP.D	=1
Global coordinates	= global	2	%PF.G none	%MVP.G	=Ask

Definition of Options

Distance =

Sets the distance that the object will be moved. The Distance value can be any positive or negative number, or the word "Points." A positive distance will move the object to the right of the original direction it was drawn.

Points means you will be prompted to locate a point after picking the object to define the distance.

Global =

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

MV Parallel

You will first be prompted to pick the object to be moved parallel and then to locate a point where it will be moved to. If the object you picked was part of a group, you will be asked the appropriate global change question.

m p d=-db.w/2

The objects that you pick will be moved parallel a distance equal to one half of the current Doubleline width. The move will be made to the left of the direction the object was originally drawn. For example, a line drawn from left to right would be moved up. If the object you picked was part of a group, you will be asked the appropriate global change question.

Move Segment or MVS

2D

The Move Segment command relocates segments of Line, Doubleline, Spline and Fillet primitives. If another primitive type is picked, you will be asked if it should be converted into a line so the edit can be performed.

Each undo (<%> key) will return the last moved segment to its original location. The Undo buffer will remember the last 128 segments moved with this use of the command.

This command can also be entered as "MOVE_Segment" or "MV Segment".

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Xdistance	= number	1	none	%MVS.X	=0
Ydistance	= number	2	none	%MVS.Y	=0
Global	= global	3	%PF.G	%MVS.G	=Ask
Vector			none	%MVS.V	
Convert			none	%MVS.C	
Distance	= number	4	none	%MVS.D	=0
Angle	= angle	5	none	%MVS.A	=0
coordinates			none		

Definition of Options

Xdistance =

Sets the distance that segments will be moved along the X axis (horizontally). A positive X axis value is to the right, while a negative value is to the left. Entering a value for this option will cause an error to be issued if you are using the Vector option or the Distance and Angle options below.

Ydistance

Sets the Y axis distance like the Xdistance option above.

Global =

See **Common Options** for more information.

Vector

This keyword-only option requires the distance and angle of the move to be specified by locating two points with the crosshairs. An error will be issued if this option is used at the same time as the X or Y distance options or the Distance and Angle options below.

Convert

This keyword-only option, when used, will force the conversion of the item picked into a line and bypass the safety question. Since segments are parts of line primitives, this conversion is necessary if the item picked is not a line primitive.

Distance =

This option, along with the Angle option below, allows you to specify the distance vector numerically. Any positive or negative distance value is valid. An error will be issued if this option is used at the same time as the X or Y distance options or the Vector option.

Angle =

This option works with the Distance option to specify the distance vector numerically. An error will be issued if this option is used at the same time as the X or Y distance options or the Vector option. See **Common Options** for more information on entering angles.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Move_Seg

Segments will be moved one at a time by picking them, dragging them to the desired location and locating a point to place them. If this change is being made to a copied group, you will be asked if the change should be made to the other copies.

mVs V

You will first be prompted to locate two points to define the distance that segments will be moved. You may then pick the segments to be moved this distance. If this change is being made to a copied group, you will be asked if the change should be made to the other copies.

Mv s,X=.5 G=0

Pick the segments you wish to have moved a positive one half unit to the right. If this change is being made to a copied group, only the group picked will be changed.

Mv s,D=3 A=20

Pick the segments you wish to have moved a distance of 3 work units at 20 degrees. If this change is being made to a copied group, only the group picked will be changed.

9

Move Vertex or MVV

2D

The Move Vertex command relocates vertices of line, doubleline, spline and fillet primitives. If another primitive type is picked (i.e. a rectangle), you will be asked if it should be converted into a line so the edit can be performed.

Each undo (<%> key) will return the last moved vertex to its original location. The Undo buffer will remember the last 128 vertices moved with this use of the command.

This command can also be entered as "MOVE_Vertex" or "MV Vertex".

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Xdistance	= number	1	none	%MVV.X	=0
Ydistance	= number	2	none	%MVV.Y	=0
Global	= global	3	%PF.G	%MVV.G	=Ask
Vector			none	%MVV.V	
Convert			none	%MVV.C	
Distance	= number	4	none	%MVV.D	=0
Angle	= angle	5	none	%MVV.A	=0
coordinates			none		

Definition of Options

Xdistance =

Sets the distance that vertices will be moved along the X axis (horizontally). A positive X axis value is to the right, while a negative value is to the left. Entering a value for this option will cause an error to be issued if you are using the Vector option or the Distance and Angle options below.

Ydistance

Sets the Y axis distance like the Xdistance option above.

Global =

See **Common Options** for more information.

Vector

This keyword-only option requires the distance and angle of the move to be specified by locating two points with the crosshairs. An error will be issued if this option is used at the same time as the X or Y distance options or the Distance and Angle options below.

Convert

This keyword-only option, when used, will force the conversion of the item picked into a line and bypass the safety question. Since vertices are parts of line primitives, this conversion is necessary if the item picked is not a line primitive.

Distance =

This option, along with the Angle option below, allows you to specify the distance vector numerically. Any positive or negative distance value is valid. An error will be issued if this option is used at the same time as the X or Y distance options or the Vector option.

Angle =

This option works with the Distance option to specify the distance vector numerically. An error will be issued if this option is used at the same time as the X or Y distance options or the Vector option. See **Common Options** for more information on entering angles.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Mov Vert

Vertices will be moved one at a time by picking them, dragging them to the desired location and locating a point to place them. If this change is being made to a copied group, you will be asked if the change should be made to the other copies.

M V V

You will first be prompted to locate two points to define the distance that vertices will be moved. You may then pick the vertices to be moved this distance. If this change is being made to a copied group, you will be asked if the change should be made to the other copies.

Mv v,X=.5 G=0

Pick the vertices you wish to have moved a positive one half unit to the right. If this change is being made to a copied group, only the group picked will be changed.

MvV,D=3 A=20

Pick the vertices you wish to have moved a distance of 3 work units at 20 degrees. If this change is being made to a copied group, only the group picked will be changed.

Move World or MVW

3D

The Move World command relocates all 3-dimensional data relative to the display window. This allows you to move data back into your display window that may have been moved outside of it because of rotating objects or changing views.

First locate a point that you want the data moved from, and then locate a point that you want the data moved to.

This command can also be entered as "MOVE_World" or "MV World".

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
coordinates			none		

Definition of Options

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

m w

You will be prompted to locate two points that define the distance that the world should be moved in your display window.

New or NW

2D & 3D

The New command removes the current drawing from your screen and creates a new blank drawing. If the current drawing had been altered since the last save to disk, you will be asked if you wish to save the changes before clearing.

The options of this command allow you to set up the new drawing. Any options not used will default to the values used in the drawing that was just cleared.

You cannot undo the New command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Width	= number	1	last	%NW.W	= 34
Height	= number	2	last	%NW.H	= 22
Scale	= number	3	last	%NW.S	= 1
Mscale	= number	4	last	%NW.M	= 1
WOrk	= units	5	last	%NW.WO	= Inches
Plot	= units	6	last	%NW.P	= Inches
Resolution	= number or Fractional or Decimal	7	last	%NW.R	= Fractional
Drawing	= name	8	none	%NW.D	= ``
SEttings	= Default or Stored	9	last	%NW.SE	= Default
Force				%NW.F	= none

Definition of Options

Width =

This option sets the width of the "sheet of paper" you are going to draw on. We call this the workarea. The width and height of the workarea is measured in plot units (also called sheet units). Drawing scale has no effect on the width value because it is defined in plot units.

Height =

This option sets the height of the "sheet of paper" you are going to draw on. Other than that, it is the same as width.

N Commands

Scale =

This option sets the scale that you wish to work in. Scale is the ratio you desire between plot units and work units, disregarding the fact that these units may be different. This ratio is commonly referred to as the English Scale. For example, if you set the work units to feet, the plot units to inches, and the scale to 1/8 (.125), then you are working at 1/8"=1'. A scale of "1" means that there is one work unit on your drawing for each unit on the sheet of paper when plotted. A scale of ".0625" means there are sixteen work units for each sheet unit. This option is mutually exclusive with the Mscale option below. If Mscale is used, then the scale is automatically calculated.

Note: This can cause some confusion for metric users because scales are expressed differently internationally than in the USA. For example, if work units is set to meters, plot units are set to millimeters and scale is set to "1", this means that objects on your drawing that are one meter large will be plotted one millimeter large on the paper. This would normally be expressed as a scale of 1:1000 (or 1/1000). You may have to apply an artificial scale value to obtain the scale desired.

Mscale =

Mscale is also the ratio you desire between plot units and work units but taking into account the units themselves. This ratio is commonly referred to as the Absolute (or Metric) scale, and is equivalent to the ratio between the size of an object in real life and its size on the drawing. In the example above, the Mscale would be 96, because working at 1/8"=1' is really the same as saying that you are going to draw things 96 times smaller than real life. You can calculate the Mscale by taking the expression for the English scale, converting both sides to the same units, and dividing the right side by the left side. The example above would yield the expression 1/8"=12" giving the result of 96. This option is mutually exclusive with the Scale option above. If scale is used, then the Mscale is automatically calculated.

Work =

This option sets the units that you will work in. Work units are the units that most distances and sizes are measured in. Valid Work values are:

MILS	or ML	MILLimeters	or MM
Inches	or IN	Centimeters	or CM
Feet	or FT	Meters	or M
Miles	or MI	Kilometers	or KM

Plot =

This option sets the units that you will plot in. Plot units are the units that your drawing sheet is measured in. The Width and Height values are in plot units. Valid Plot values are the same as for work units.

N Commands

Resolution =

Sets the smallest addressable unit of the workarea. Valid Resolution values are decimal, fractional or a real number.

Decimal means the program will compute the smallest addressable unit that is a multiple of ten and does not exceed 536,870,912 units in either axis of the workarea.

Fractional means the program will compute the smallest addressable unit that is a multiple of two and does not exceed 536,870,912 units in either axis of the workarea.

number is the number of addressable units that is desired per work unit. In this case you cannot exceed approximately 2 billion units in either axis of the workarea. Multiply the number by the scale and then the number of plot units on the longer axis to determine the total number of resolution units in the workarea. Make sure your resolution divides evenly into the work units you are using. Feet divided into thirteenths, for example, are not too handy.

Now is as good a time as any to explain the difference between drawing resolution and display resolution. The display resolution is limited to 16 bit accuracy while the drawing resolution is 32 bit accuracy. This means that coordinates can be entered into your drawing much more accurately than they will be displayed. Display resolution is limited to 16 bits to speed up redraw times.

A 32 bit drawing resolution can conceivably give us up to about 4 billion points of accuracy along one axis of the drawing (if you include both the positive and negative quadrants). You will notice by the restrictions mentioned above that we do not allow that level of accuracy in the new workarea when resolution is automatically calculated using the Decimal or Fractional values. The reason for this is that we are reserving a portion of these 4 billion units should you decide to increase the size of your workarea later (make your drawing bigger in width or height) or move your drawing origin. However, when defining your own resolution you have access to about 2 billion units. Keep in mind that if you use them all for your current sheet size you will have none left over should you later need to make your sheet larger. On the other hand, you will have created a drawing workarea with the greatest possible accuracy.

To give you an idea of the level of accuracy that is obtainable in a TurboCAD drawing, a drawing 34 inches wide by 22 inches high with work units in full scale inches has an addressable resolution of over one eight millionth of an inch (1/8388608).

Drawing =

Names your drawing. This is the name that will be used as the default file name when saving your drawing to disk. File names containing spaces or special characters should be quoted (like this` or \like this\) so they are not read as separate options to the command.

Settings =

This option can change the settings of all dialogs at the time a new drawing workarea is created. Valid Settings values are Default and Stored. If this option is not used, the dialogs (and all Current Status Words) are left as they are.

Default means all dialogs will be set to their factory default settings.

Stored means all dialogs will be set to their last stored settings. The Store option of the Preferences command determines if and when dialog settings are stored.

Force

This option forces the clearing of the existing drawing by bypassing the safety question. This option is most commonly used in macros where operator intervention is not desired.

Caution: Use of this option can cause loss of data with no warning.

Examples

New

Because no options were entered, they will all use their default values. If a drawing is currently open, the values for all options will default to those of the opened drawing. You will be given an opportunity to save the open drawing if it has not yet been saved. If TurboCAD has just been started, the options will default to the values shown under 'Startup' in the Options listing above.

N,42,30,1/8,,f,i,f

A new drawing workarea will be created that is 42 inches wide and 30 inches high. Work units will be fractional Feet at 1/8 scale ($1/8" = 1'0"$). Plot units are inches. TurboCAD will calculate the Mscale and set it to 96. The drawing workarea will be unnamed, except on Macintosh computers where it is titled "Untitled." If a drawing was open when the command was executed, you will be given an opportunity to save it.

nw d=`Piping Diagram` sc=nw.s*2

A new drawing workarea will be created that has the the same setup parameters as the open drawing except that it will be named "Piping Diagram" and its scale will be twice that of the open drawing. If TurboCAD was just started, setup parameters will be per the "Startup" values listed above, except the scale will be 2.

N,850,600,,100,m,mm,d

A new drawing workarea will be created that is 850 inches wide and 600 millimeters high. Work units will be meters at an absolute scale of 1:100. Plot units are millimeters. TurboCAD will calculate the scale and set it to 10 (meaning that 1 meter entered will result in a distance of 10 mm on the paper, which is the same as a ratio of 1:100). The drawing workarea will be unnamed, except on Macintosh computers where it is titled "Untitled". If a drawing was open when the command was executed, you will be given an opportunity to save it.

OPen or OP

2D & 3D

This command is a built-in alias for the "Load" command. See the Load command for information.

Origin or OR

2D

The Origin command relocates the origin of your drawing. The origin sets the absolute zero (i.e. 0,0) coordinate of your drawing in both the X and Y axes. TurboCAD makes the lower left hand corner of your drawing as the default origin. You can specify the new origin by locating a point, specifying offsets from a point you locate, or specifying offsets from the existing origin.

The rounding (crosshair jumping) and reference grid will automatically be redrawn to originate at the new origin.

Hint: It is best to define a new origin with Rounding turned on so the new origin coincides with existing data on your drawing.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
X	= number	1	0.0	%OR.X	= 0.0
Y	= number	2	0.0	%OR.Y	= 0.0
Point	= switch	3	Yes	%OR.P	= Yes
Reset			none	%OR.R	
coordinates			none		

Definition of Options

X =

Specifies the distance along the X axis, in plot units, that the new origin will be relative to the existing origin or from a point you locate. If you use the Point option below, it will be relative to the point you locate. If not, it will be relative to the existing origin.

Y =

Specifies the distance along the Y axis, in plot units, that the new origin will be relative to the existing origin or from a point you locate. If you use the Point option below, it will be relative to the point you locate. If not, it will be relative to the existing origin.

Point

This keyword-only option, when used, will cause TurboCAD to ask you to locate a point to define the new origin. If values were entered for X and/or Y above, then the new origin will be offset from the point you locate by those values. If this option is used without entering values for X and/or Y, then the point you locate will be the new origin.

Reset

Moves the origin of your drawing to the extreme lower left corner. This is the default origin location when a new drawing is created.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples**O**

You will be prompted to locate a point that you want to be the new origin. The coordinates of the point relative to the old origin are displayed in the Status Area. If several points are located, only the last point is used.

or,3,5,no

The new origin will be 3 units to the right of, and 5 units above, the existing origin.

or r

The origin of your drawing will be set at the lower left corner of the drawing.

OVerlay or OV**2D**

The Overlay command relocates objects on your drawing such that they are overlaid on a common point of your choice. You first locate the overlay point, then select objects that you wish to have moved to that point. In essence, you are moving all selected objects to same place on your drawing. If you opt to select objects by picking them, then the point that is picked on each object is the point that will be overlaid on the overlay point. If you select objects by defining a box, then the origin of each object is the point that will be overlaid on the overlay point. This command is similar to the Align command, except that you are defining a point for overlaying rather than a line for alignment.

O Commands

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Select	= select	1	%PF.S	%OV.S	=Pick
Object	= object	2	%PF.O	%OV.O	=Any
coordinates			none		

Definition of Options

Select =

See **Common Options** for more information.

Object =

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

OverLay

You will first be prompted to locate the overlay point and then to pick the objects you wish to have moved to that point. The point picked on the object is the point that will be overlaid on the point. Remember, you can use the Locate function keys to snap to vertices, midpoints origins, etc.

ov,all,circles

All circle primitives on your drawing will be moved so that they overlay on a point that you locate. The origin of each circle will be overlaid on the point.

Over In

All objects that are completely inside a box that you define by locating its diagonally opposite corners will be moved so that they overlay on a point that you locate. The origin of each object will be overlaid on the point.

Path or PA**2D & 3D**

The Path command specifies the directories (folders) that are to be searched when TurboCAD tries to retrieve a file from disk. Drawings are retrieved from disk via the Load, Open, and Merge commands. Symbols are retrieved via the Get, Insert, and Replace commands. Macros, which are just text files, are retrieved for execution via the Macro command (or the "/" alias) and are retrieved for editing via the Modifytext command. Other commands which retrieve text files are the Text and File commands. A different path can be set for each different type of file, allowing the user to organize disk files effectively. A Current path can also be set which is always searched.

Each file type can have several directories (folders) listed that will be searched in the specified order. When a request is made by a TurboCAD command to retrieve a file from disk, the current folder will be searched and then the folders specified for that file type in the order listed. The search is terminated upon finding the first occurrence of the file. Use the Delimiter character below to separate folder names when specifying multiple folders. Remember to quote folder names that contain spaces or special characters (like this` or \like this\). See the Examples below.

To separate directories in a path on Macintosh computers, you use a colon (:).

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Current	= folder	1	last	%PA.C	= none
Drawings	= folder	2	last	%PA.D	= none
Symbols	= folder	3	last	%PA.S	= none
Macros	= folder	4	last	%PA.M	= none
Patterns	= folder	5	last	%PA.P	= none
DElimiter	= character	6	` `	%PA.DE	= ` `

Definition of Options**Current =**

Specifies the Current directory The current directory is the one that files are saved to by default and is also the directory that is searched first when requesting disk files. Drawings, however, are saved to the directory from which they came regardless of changes to the Current directory, unless you specifically specify a different directory for saving.

Drawings =

Specifies the folders (directories) that will be searched when retrieving a drawing from disk using the Load, Open or Merge commands. This has no effect on where drawings are saved.

P Commands

Symbols =

Specifies the folders (directories) that will be searched when retrieving a symbol from disk using the Get, Insert or Replace commands. This has no effect on where symbols are saved.

Macros =

Specifies the folders (directories) that will be searched when executing a macro or using the Modifytext, Text, and File commands.

Patterns =

Specifies the folders (directories) that will be searched when retrieving a Crosshatch pattern from disk. This option performs no function on Macintosh systems because the crosshatch patterns are built into the program.

DElimiter =

Sets the delimiter that will be used to separate paths. The Delimiter value must be a single keyboard character. If the character is a space or special character it must be quoted ('like this' or \like this\).

Examples

PA `` `HardDisk:Drawings:` `HardDisk:Symbols:`

The Current folder remains the same. when a drawing or symbol is requested, the current folder will be searched for it. If it is not found, then the folder 'HardDisk:Drawings' will be searched for drawings and the folder 'HardDisk:Symbols' will be searched for symbols.

pa c=` m=`HardDisk:Textfiles:`

The Current folder remains the same. when the Macro, Text, Modifytext, or File commands make a request for a file, then the folder 'HardDisk:Textfiles' will be searched after the current directory.

pa,`HardDisk:NewProject`

The Current folder is changed to the folder 'HardDisk:NewProject'. All files saved will go to this folder. The only exception is if you load an existing drawing, in which case it will be saved into the folder from which it came.

path

This is an easy way to set the current directory on Macintosh computers. A dialog will appear in which you select any file from the folder you wish to be the Current directory. It will automatically become the Current directory. The file you select can be of any type, i.e. it is irrelevant. Only the folder in which it is important.

PAUse or PS**Macro**

The Pause command temporarily suspends the execution of commands in TurboCAD. This feature can be useful in macros to stop or slow down the execution of the macro so the user can have more time to see what is happening. For pauses which provide you with a message or allow user input, see the Message and Prompt commands.

This command can also be used interactively. Used this way, the keyboard appears to be “locked” for the pause period. Any entries made through the keyboard during the pause period will be “queued” and executed when the pause period ends.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Time	= seconds	1	last	%PS.T	=5

Definition of Options

Time =

Sets the length of the pause in seconds. A valid Pause value is any whole number from 0 through 360. A time of zero means the pause will last until a key is pressed.

Examples

Pause 15

TurboCAD will be in a state of suspended animation for fifteen seconds.

ps

TurboCAD will be suspended for the same period of time as the last time the Pause command was used.

Pen or PN**2D & 3D**

The Pen command sets the current pen that you wish to draw with. You can also temporarily display all of the items drawn with a single pen with this command. The current work Pen is the pen that new objects will be drawn with by default, and is the pen displayed in the Status Area.

There are 256 pens on each drawing, numbered 0 through 255. Every object on your drawing has a pen assignment, just like it has a Class, Layer, Style and Priority assignment. Pens are a

P Commands

property of each primitive (or group member). Line weights are assigned to pens using the Weight command and colors are assigned to pens using the Colors command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Work	= 0 thru 255	1	last	%PN.W	=0
Display	= 0 thru 255	2	last	%PN.D	=0

Definition of Options

Work =

Sets the current work pen that is displayed on the Status palette at the bottom of your screen. The work pen is the default pen that new objects will be drawn with if the Pen option is not used in a command that creates graphic primitives. You must enter a whole number from 0 through 255.

Display =

Sets the pen that will be temporarily displayed. Only the data on your drawing which was created in that pen will be displayed until you hit any key on your keyboard. You can continue working on your drawing only after all of the pens are displayed. You must enter a whole number from 0 through 255. This is handy for checking which objects are drawn with a given pen when working on a monochrome screen.

Examples

PEN

Executing the Pen command this way will do nothing. All of the command options will default to last used, which will not change the Work or Display settings.

pn 4

Your work pen will be set to four.

Pen Display=13

Only objects drawn in pen thirteen will be displayed temporarily. Hitting any key will redisplay all of the pens on your drawing.

PPoint or PT

2D

The Point command creates point markers on your drawing. Several styles of markers are available. Point markers are useful as construction or reference points that can be snapped to. Point markers will not plot using the TurboCAD Plot program, but they will print from

Macintosh computers. Multiple points can be placed in a single use of this command, but they will all be part of the same Point primitive, much like all of the segments in a multi-segment line are part of a single line primitive. To delete a single point marker within a multi-point primitive, you must use the Delete Member command.

Each undo (<%> key) will remove the last point located. The Undo buffer will remember the last 124 points located with this use of the command.

This command does not change the "marker" used to temporarily mark points located while working in other commands. This marker is controlled via the Marker option of the Preferences command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Marker	= Diamond or Dot or Period or PLus or Asterisk or O or X or Square or DOTSquare or DOTDiamond or XSquare	1	last	%PT.M	=Diamond
Function	= P0ints (or Pts) =Points or Distance or Percent or DIVide	2	last	%PT.F	
Distance	= number	3	last	%PT.D	=1
Percent	= number	4	last	%PT.P	=50
DIVide	= 2 thru 127	5	last	%PT.DIV	=4
Class	= 0 thru 255	6	%WK.C	%WK.C	=0
Layer	= 0 thru 255	7	%WK.L	%WK.L	=0
Pen	= 0 thru 255	8	%WK.P	%WK.P	=0
PRiority	= 0 thru 255	9	%WK.PR	%WK.PR	=125

P Commands

WOrkset	= workset	10	%WK.W	%WK.W	=none
Mimic			none	%PT.MI	
coordinates			none		

Definition of Options

Marker =

· Dot	□ Square
+ Period	◇ Diamond
+ Plus	◻ Dot in Square
* Asterisk	◇ Dot in Diamond
○ O	⊠ X in Square
× X	

Selects the type of point marker to be used. The available markers are shown here as they appear on the dialog of a Macintosh computer. The marker names, however, must be entered as shown on the previous page.

Function =

Determines how point markers are to be located on your drawing. Valid values are Points (or Pts), Distance, Percent, or Divide. The definitions of these values are as follows:

Points (or **Pts**) means simply that a point marker will be placed wherever you locate the point, either with the crosshairs or by coordinates. Remember that the location of the point with the crosshairs is affected by the use of a Locate key.

Distance means that a point will be located a specific distance (as specified by the "Distance=" value below) from the nearest vertex of the segment or arc selected. The segment or arc must be selected with the key. The distance value can be a positive or negative number, and it is interpreted in the current Work Units. A positive value means that the distance will be measured from the nearest vertex along the direction of the segment or arc and can go beyond the end of the object. A negative value means that the distance will be measured from the nearest vertex but 180 degrees from the direction of the segment or arc.

Percent means that TurboCAD will first calculate the length of the segment or arc selected and then place a point marker a percent of that distance (as specified by the "Percent=" value below) from the nearest vertex, along the object. The segment or arc must be selected with the key. The percentage value can be a positive or negative number, and can be greater than 100 percent. Positive and negative percentage values have the same meaning as positive or negative Distance values as defined above.

Divide means that TurboCAD will first calculate the length of the segment or arc selected and then place the number of point markers necessary to divide that length into a number of equal lengths as specified by the "Divide=" value below.

Distance =

Sets the distance, in Work Units, to be used when the "Function=" option above is set to Distance. Any positive or negative value is valid.

Percent =

Sets the percentage to be used when the "Function=" option above is set to Percent. Any positive or negative value is valid.

DIVide =

Sets the number of equal divisions to be calculated when the "Function=" option above is set to Divide. Any integer between 2 and 127 is valid.

Class =

See **Common Options** for more information.

Layer =

See **Common Options** for more information.

Pen =

See **Common Options** for more information.

PRiority =

See **Common Options** for more information.

WOrkset =

See **Common Options** for more information.

Mimic

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Point X

The "X" marker will be used to mark points that you locate. The markers will be in the Class, Layer, Pen and Priority shown on the Status palette.

pt,di,d,4.5

The "Diamond" marker will be used to mark points that you locate. A point marker will be placed a distance of 4.5 work units from the nearest vertex of any arc or segment located with the key. The markers will be in the Class, Layer, Pen and Priority shown on the Status Palette.

pt,,p,,40

The marker last used with this command will be used to mark points that you locate. TurboCAD will first calculate the length of any arc or segment located with the key, then

P Commands

place a point marker at a point 40 percent of that distance away from the nearest vertex. The markers will be in the Class, Layer, Pen and Priority shown on the Status palette.

pt,,DIV,,8

The marker last used with this command will be used to mark points that you locate. TurboCAD will first calculate the length of any arc or segment located with the key, then place point markers as necessary to divide the object into eight equal lengths. The markers will be in the Class, Layer, Pen and Priority shown on the Status palette.

point mimic

You will be prompted to select with the crosshairs an existing point marker on your drawing, then you will be prompted to locate new points. The marker used, as well as the Class, Layer, Pen and Priority will be the same as those of the point marker selected.

POLygon or PG

2D

The Polygon command creates equilateral polygons on your drawing. A polygon is created using the same construction methods as those used in creating a circle. The polygon is inscribed in the imaginary circle that you define.

Each undo (<%> key) will remove the last point located on the current polygon. Each undo past the current polygon will remove one complete polygon. The Undo buffer will remember the last 124 polygons created with this use of the command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Sides	= number	1	last	%PG.S	=6
Construction	= Radius or Diameter or Circumference	2	Radius	%PG.C	=Radius
SIZE	= number	3	none	%PG.SIZ	=none
Angle	= angle	4	none	%PG.A	=none
CLass	= 0 thru 255	5	%WK.C	%WK.C	=0
Layer	= 0 thru 255	6	%WK.L	%WK.L	=0
Pen	= 0 thru 255	7	%WK.P	%WK.P	=0
STyle	= style	8	%WK.S	%WK.S	=Solid
PRiority	= 0 thru 255	9	%WK.PR	%WK.PR	=125
WORkset	= workset	10	%WK.W	%WK.W	=none

Mimic	none	%PG.M
coordinates	none	none

Definition of Options

Sides =

Specifies the number of sides that the polygon will have. A valid Sides value is a whole number from 3 through 120. Three sides will yield a triangle, four a square, and so on.

Construction =

This option specifies how the polygon will be constructed. Valid values are: Radius, Diameter and Circumference.

Radius will construct a polygon by defining its radius. The radius is the linear distance between the center (origin) and a vertex on the polygon. If the Size or Angle options are not used, a second point is required to define the size and/or angle of each polygon.

Diameter will construct a polygon by defining its diameter. The diameter is the linear distance from one vertex of the polygon to the opposite side of the imaginary circle the polygon is inscribed in by passing through the center. If the Size or Angle options are not used, a second point is required to define the size and/or angle of each polygon. If a size is entered, the center point and a point on the polygon is located. Otherwise, the points are located on opposite sides of the imaginary circle.

Circumference will construct a polygon by defining its circumference. The circumference is the radial distance around the imaginary circle the polygon is inscribed in. If the Size option is not used, three points are required to define each polygon. If the Size option is used, only the center point is required. If the Angle option is not used, a second point is required to define the rotation angle of the polygon. This point is the same as the second point that defines the size if it's not entered.

SIZE =

This option sets the size of the polygon. The size is either the radius, diameter or circumference as set by the Construction option. Valid Size values are any positive number. If this option is not used, two or three points will be required to define the radius, diameter or circumference of the polygon.

Angle =

This option sets the rotation angle of the polygon. A valid Angle value is any number between -360 and 360, or -2^1 and 2^1 radians. See the Common Options section for more information on angle values.

Class =

See the **Common Options** section for more information.

Layer =

See the **Common Options** section for more information.

P Commands

Pen =

See the **Common Options** section for more information.

STyle =

See the **Common Options** section for more information.

PRiority =

See the **Common Options** section for more information.

WOrkset =

See the **Common Options** section for more information.

Mimic

See the **Common Options** section for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Polygon Sides=8

Polygons will be drawn with eight sides. You will be prompted to locate two points to define both the radius and angle of each polygon. The two points are the center point and a vertex of the polygon. The Class, Layer, Pen, and Style options will default to the values shown on the Status palette.

pg 3 c 4 45

Polygons will be drawn with three sides. The size of the polygon is determined by inscribing it inside an imaginary circle that has a circumference of four units. You will be prompted to locate one point, the center point, of each polygon. One vertex of the polygon will be at an angle of 45 degrees counterclockwise from horizontal. The Class, Layer, Pen, and Style options will default to the values shown on the Status palette.

PREferences or PF

2D & 3D

The Preferences command sets the value of some general options that are used by many other TurboCAD commands, macros and Locate keys. For a more detailed narrative on Preferences, refer to the Preferences paragraphs in the **Setting Up a Drawing** section in Basics.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Zoom	= .001 thru .999	1	last	%PF.Z	= 0.5

P Commands

Lock	= angle	2	last	%PF.L	= 15
Polar	= angle	3	last	%PF.P	= 0
Angle	= Degrees	4	last	%PF.A	= Degrees
	or Radians				
	or Gradians				
Clockwise	= switch	5	last	%PF.C	= No
Marker	= Diamond	6	last	%PF.M	= Diamond
	or Dot				
	or Period				
	or Plus				
	or Asterisk				
	or 0				
	or X				
	or Square				
	or DOTSquare				
	or DOTDiamond				
	or XSquare				
Select	= select	7	last	%PF.S	= Pick
Object	= object	8	last	%PF.O	= Any
Global	= global	9	last	%PF.G	= Ask
SEArcharea	= .001 thru .999	10	last	%PF.SEA	= 0.13
Prompts	= switch	11	last	%PF.P	= On
CUrsor	= Crosshairs	12	last	%PF.CU	= Crosshairs
	or Plus				
	or X				
	or Largeiso				
	or Smalliso				
	or None				
Highlight	= Fuzzy	13	last	%PF.H	= Fuzzy
	or Halftone				
	or Blink				
	or None				
	or Next				

P Commands

	or Field				
	or Dialog				
	or Highlight				
	or Small				
	or Large				
	or 0 thru 10				
SString	= string	14	last	%PF.ST	= ``
Rounding	= number	15	last	%PF.R	= 1/1024
MACro	= 1 thru 127	16	last	%PF.MAC	= 5
COmmand	= switch	17	last	%PF.CO	= Off
Format	= Fractional	18	last	%PF.F	= Fractional
	or Float				
	or Integer				
	or One				
	or Two				
	or THree				
	or Four				
	or Flve				
	or Six				
Tracking	= None	19	last	%PF.T	= None
	or Slow				
	or Medium				
	or Fast				
Keys	= switch	20	last	%PF.K	= Off
TABs	= number	21	last	%PF.TA	= 8
Verbose	= None	22	last	%PF.V	= None
	or Type				
	or Debug				
	or Both				
SMoothing	= 0.0 thru 1.0	23	last	%PF.SM	= 0.5
STOre	= None	24	last	%PF.STO	= None
	or Quit (or Exit)				
	or Immediately				

SAve	= number	25	last	%PF.SA	= 50
Base	= Last or Polar or Horizontal	26	last	%PF.B	= Last
SCroll	= switch	27	last	%PF.SC	= On
Vlewlocate	= switch	28	last	%PF.VI	= On

Definition of Options

Zoom =

Sets the percentage of zoom that happens when the <Z> (zoom) and <U> (unzoom) keys are pressed. The Zoom value must be a number greater than 0 and less than 1. A Zoom value of ".5" (the default) means that each zoom in will display one half of the previous window. Unzooming (zooming out) does the inverse, it displays twice the previous window.

Lock =

Sets the incremental angle value that will be locked to when using the <A> (angle lock) key. A valid Lock value is any number between -360 and 360. A Lock value of forty-five degrees, for example, means that points located with the <A> key would be rounded to the nearest 45 degree increment from the previous point. The "Zero" angle is determined by the value entered for the "Base=" option below. See the **Common Options** section for information on entering angle values.

Polar =

This option sets the angle that is to be considered the zero angle for polar coordinates. A valid Polar value is any number between -360 and 360. Setting Polar value to forty-five degrees, for example, means that entering a polar coordinate with an angle of zero would be 45 degrees counterclockwise from horizontal. See the **Common Options** section for information on entering angle values.

Angle =

Sets the angle units that all angle values will be expressed in. Valid Angle values are Degrees, Radians and Gradians.

Degrees is a unit of angular measure equal in magnitude to the central angle subtended by 1/360 of the circumference of a circle. Got that? Angles expressed in degrees can be entered decimally (34.578) or in a degrees/minutes/seconds format (23°14'42"). See the **Common Options** section for more information on entering angle values.

Radians is a unit of angular measure equal in magnitude to the central angle subtended at the center of a circle by an arc of length equal to the radius of the circle. Wow, that's worse than the description for degrees! Anyway, a radian is equal to $360/2^1$, or approximately 57°17'44.6". A complete circle is equivalent to 2^1 radians. Angles expressed in radians are usually entered decimally (34.578). If an angle is entered in a degrees/minutes/seconds format (23°14'42"), it is automatically converted into radians. See the **Common Options** section for information on entering angle values.

P Commands

Gradians is a unit of angular measure equal to 1/400th of a circle (0.9 degrees). A complete circle is equivalent to 400 radians. Angles expressed in gradians are usually entered decimally (100.00). See the **Common Options** section for information on entering angle values.

Clockwise =

Sets the direction, clockwise or counterclockwise, that is the direction for positive angle values. The Clockwise value must be a switch value. Switch values are: True or False, Yes or No, On or Off, All or None. A positive switch value means positive angle values will be clockwise and a negative switch value (the default) means positive angle values will be counterclockwise.

Marker =

Specifies which marker type will be used by commands that temporarily mark points on your drawing. The available markers are the same as in the Point command and are shown here.

Select =

The setting of this option is used as the default value for all other commands that have a Select option. This option determines how objects are to be selected. Valid Select values are Pick, Inside, Outside, Touching, Notinside, NotOutside, NotTouching and All. See the **Common Options** section for more information.

Object =

The setting of this option is used as the default value for all other commands that have a Object option. This option determines the type of objects that can be affected by a command. Valid Object values are Any, ARcs, Boundaries, CIRCles, Copies, DIMensions, Doublelines, Ellipses, Fillets, Groups, Lines, LEaderlines, POints, POLygons, Primitives, Rectangles, Splines, Text and Wireframes. See the **Common Options** section for more information.

Global =

The setting of this option is used as the default value for all other commands that have a Global option. This option determines how global changes to copied groups will be handled. Valid Global values are ASK, All, Some and Cancel. See the **Common Options** section for more information.

SEArcharea =

This option sets the distance from the crosshairs that will be searched for objects when picking them on your drawing with a "Locate" key (such as <V>, <C>, etc.). The Searcharea value is defined in actual inches on the screen and is therefore unaffected by zooming. The value must be greater than zero but less than one.

Prompts =

This option turns the display of operator prompts on or off. This will also suppress messages that would normally be displayed in the News Area. The Prompts value must be a switch value. Switch values are: True or False, Yes or No, On or Off, All or None.

This option is most useful in a Macro that you would like to run silently .

CUrsor =

Sets the cursor type that will be used when in the graphic mode (i.e. when the crosshairs are active). Enter one of the cursor names listed in the Options list above. The option performs the same function as the " key.

Highlight =

Controls the way TurboCAD will highlight an object when in a command which causes an object to be highlighted. Examples of such commands are the Group command, the Distance command, etc., which cause objects to be temporarily highlighted. Valid values are as follows:

Fuzzy means that objects will appear "fuzzy" when highlighted. This is the TurboCAD default.

Halftone means that objects will appear dotted at a halftone of their current color.

Blink means that objects will "blink" on the screen when highlighted. On Macintosh computers only one "blink" will occur very rapidly.

None means that objects will not be highlighted. The command, however, will work properly. This is useful in a macro.

Next means that objects will be highlighted in the color of the object's pen number plus one.

Field, Dialog, Highlight, Small, Large mean that objects will be highlighted in the color which is assigned to those items in the Colors command. For example, entering "Field" will cause objects to be highlighted in the same color as the drawing background, meaning that they will temporarily disappear.

0 thru 10 means that objects will be highlighted in the color corresponding to the color assigned to the pen number entered. Colors are assigned to pens via the Colors command.

SString =

Defines the TurboCAD command line prompt string. The prompt string appears at the beginning of the command line ahead of the commands that are typed in. The standard prompt is " > ". This may be changed to any prompt string you desire up to ten characters. If numbers or spaces are to be used, the string should be quoted ('like this' or \like this\) so it will not be interpreted as a calculation or another option. Macintosh computers do not support this feature.

Rounding =

Numeric values displayed to the operator will be rounded to this value. Distances, area and coordinate readouts are examples of numeric values that are affected by this option. Setting this option to a smaller value will obviously provide more accuracy, up to the built-in single-precision limit of six places. For a more detailed discussion of rounding, refer to the Preferences paragraphs in the **Setting Up a Drawing** section in Basics.

Note: The Format option is capable of overriding the Rounding value if it is set to a larger value (ie: less decimal places).

MACro =

Sets how deeply macros can be nested. Nesting is when one macro calls another. Setting a limit on the nesting of macros helps preserve memory and prevent runaway macros (infinite recursion). See the Macro Language section for more information.

COmmand =

This option turns the display of the current command on and off. The name of the current command is normally displayed on the right side of the Prompt line on your screen. A switch value (on, off) is required.

Format =

This option determines what format numeric distance or location values which TurboCAD displays are to be presented in. Examples of numeric values are distances calculated via the Distance selection of the Query menu, sizes (lengths, widths, etc.) of existing objects when using the Change selections from the Edit menu, numeric information about an object when using the Get Info selection from the Query menu, etc. Some of the Format settings control the number of decimal places that are to be displayed. If the number of decimal places displayed by this option is less than the value set in the Rounding option, the numeric text will be rounded to the Format value. For a more detailed discussion of Format values and the interaction of this parameter with the Rounding value, refer to the Preferences paragraphs in the **Setting Up a Drawing** section in Basics.

Fractional means that the numeric text will present the calculated number in a fractional format. Some examples of fractional formats are: 3 1/2", 4 3/5 mm, 54' 4 7/8". Fractional feet will always be expressed as feet, inches, and fractions of an inch.

Float is the same as "one", except that the number will be presented as accurately as it can be calculated. This will usually display up to seven significant digits.

Integer means that numbers will be rounded to the nearest whole unit.

One means that numbers will be rounded to the nearest one decimal place (1/10 of a unit) and presented in a decimal format (like 34.5 ft).

Two is the same as "one", except that it rounds to the nearest two decimal places.

THree is the same as "one", except that it rounds to the nearest three decimal places.

FOur is the same as "one", except that it rounds to the nearest four decimal places.

Five is the same as "one", except that it rounds to the nearest five decimal places.

Six is the same as "one", except that it rounds to the nearest six decimal places.

Tracking =

This option turns cursor tracking on and off. Cursor tracking is a feature that displays the crosshairs and dynamically moves them to the next point when coordinates are read from the point stack or a macro. The slickest use of tracking is in the playback of macros. The crosshair tracking makes the playback of a macro look just like an operator is interactively locating points on the drawing. It's great for demonstration and training purposes. Valid Tracking values are None, Slow, Medium and Fast.

P Commands

None means no crosshair tracking will be used. The macro or point stack will be read as quickly as possible.

Slow means the crosshairs will track slowly across the screen (like a movie played in slow motion).

Medium means the crosshairs will track across the screen at a speed between Slow and Fast. This speed was designed to most closely simulate the normal speed most people work with the mouse.

Fast means the crosshairs will track quickly (a la fast forward).

Keys =

This option works in conjunction with tracking to temporarily sound a click and display the locate keys used during the playback of a macro. It's great for demonstration and training purposes. A switch value (on, off) is required.

TAbs =

Sets the number of spaces that tabs represent in text. Each tab entered will be replaced by the number of spaces you enter for this value. You must enter a number from 0 through 15. Zero means no space for the tab.

Verbose =

This option will display, in the News Area, the command line being processed and/or the settings of the command's options after the command line has been processed but before it is executed. This feature is very useful in debugging macros because you can see how the command interpreter is processing each line of the macro. Valid Verbose values are None, Type, Debug and Both.

None means the verbose feature is turned off. This is the normal mode that commands and Macros are processed in.

Type means the command line that was entered will be displayed verbatim in the News Area.

Debug means the options of the command will be displayed in the News Area after variable substitution and calculations are performed, but before execution. Below is an example of the Debug display of "Cir rad 4". The first column is the option name, the second column is the value that the option is set to, and the last column is the type of values that are valid for that option.

Construction	=Radius	(r/d/c)
Size	=4	('0 float)
Duplicate	=False	
Class	=0	(0'=int'=255)
Layer	=0	(0'=int'=255)
Pen	=0	(0'=int'=255)
Style	=Solid	(style)
Priority	=125	(0'=int'=255)

Both means both the Type and Debug values will be used. The Type message will be displayed first and then the Debug message.

SMoothing =

The value entered for this option controls how many segments are used when converting a curve into a line. It is an arbitrary percentage between 0 and 1. The higher the percentage, the greater the number of segments.

STOre =

This option determines if the current status of settings on all dialogs are to be saved so that they can be reinstated when TurboCAD is next run.

None means that the settings of all dialogs will startup at the factory default values.

Quit or **Exit** means the settings of all dialogs will be remembered when TurboCAD is quit. The dialogs will appear exactly as you last left them when the TurboCAD program is next run.

Immediately means the settings of all dialogs will be remembered right now (upon the execution of this command). The dialogs will appear exactly as they are right now when the TurboCAD program is next run.

SAve=

This option reminds you that you may want to save your drawing after a given number of commands executed since the last save. The Save value must be a number between 0 and 511. A value of zero deactivates the automatic reminder feature.

Base=

This option sets the Base or "Zero" angle from which the Angle Lock (<A>) locate key will work. Valid entries are Last, Polar, or Horizontal.

Last means that the angle defined by the last two points entered will be the Base angle. The <A> key will then snap to the angle increment entered for the "Lock=" option using this base angle as the Zero angle.

Polar means that the angle entered as the Zero polar angle in the "Polar=" option above will be the Base angle for operation of the Angle Lock feature.

Horizontal means that the Angle Lock feature will always work with the horizontal as the Base angle.

SCroll=

This switch value determines whether you want the vertical and horizontal scroll bars displayed on a Macintosh screen. Not displaying them will give you a slightly larger drawing area. Valid switch values are On or Off, Zero or One, True or False, and so on.

Vlewlocate=

This switch value determines whether you want the TurboCAD View and Locate menus displayed on the menu bar of a Macintosh computer. Valid switch values are On or Off, Zero or One, True or False, and so on.

Examples

Prefer ,, 0.33 , 5

The Zoom factor will be set to 33% and the Lock angle will be set to five degrees. Pressing the <Z> (zoom) key will zoom in displaying one third of the previous window. The <U> (unzoom) key will do the inverse. The <A> (angle lock) key will snap to the nearest five degree increment using the Base angle last set in the "Base=" option.

PRE GL=ALL SEARCH=.25

The Global option will be set to All and the Searcharea option will be set to 1/4 inches. Any TurboCAD command that can perform global edits will default to performing the edit to all copies if the Global option is not used in that command. When locating points on your drawing that picks or snaps onto existing objects, objects will be searched for up to one quarter of an inch on the screen away from the crosshairs.

pf polar=90 mark=plus

The Polar angle will be set to ninety degrees counterclockwise and the Marker type will be set to "Plus." All polar coordinates entered will assume that a zero angle is pointing up towards the top of your drawing. Points marked temporarily in graphics commands will be marked with a plus sign (+).

pref prompt=no

This command is typically entered at the beginning of a macro so that the command prompts are not displayed during its execution, making the macro run faster.

pref sc=no vi=no

This command will cause the scroll bars and View and Locate menus on a Macintosh screen to disappear.

PRiority or PR

2D & 3D

The Priority command sets the priority that you wish to draw objects in. Priority sets a precedence for picking objects on your drawing. When picking objects to move, for example, if there were two objects near the crosshairs, the one with the highest priority is the one that would get selected. See Locating Points and Common Options for a more detailed narrative on setting, changing, and using priorities.

There are 256 priorities on each drawing, numbered 0 through 255. Every object on your drawing has a priority assignment, just as it has a Class, Layer, Pen and Style assignment. Priority is a property of each group. A primitive that has not yet been made part of a group is, for all intents and purposes, a group of only one item. Therefore, it also has a priority assigned to it. But when several primitives are grouped together, they all assume the same priority. All members of a group have the same priority assignment because the priority is assigned to the

P Commands

entire group, not the individual members. If the group was later broken, the members would still retain the priority assignment they had when part of the group.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Work	= 0 thru 255	1	last	%PR.W	=125

Definition of Options

Work =

Sets the current work priority. The work priority is the default priority that new objects will be drawn in if the Priority option is not used in a command that creates graphic primitives. You must enter a whole number from 0 through 255

Examples

pr 4

Your work priority will be set to four.

Priority Work=13

Your work priority will be set to thirteen.

PROject or PJ

3D

The Project command lets you view 3-dimensional objects in one of several different types of projections. Projection types include Normal (parallel projection), Oblique and Perspective.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Type	= PArallel or Oblique or Perspective	1	last	%PJ.T	= Parallel
Distance	= number	2	last	%PJ.D	= %NW.W
Height	= number	3	last	%PJ.H	= %NW.H
Angle	= angle	4	last	%PJ.A	= 45
DEpth	= number	5	last	%PJ.DE	= 0.75

Definition of Options

Type =

Sets the type of projection desired. Valid Type values are Parallel, Oblique and Perspective.

Parallel indicates a parallel projection. The viewing angle of this projection is determined by the View or Rotate World commands. The Distance, Height, Angle and Depth options are ignored with this type of projection.

Oblique is a type of parallel projection where one plane is parallel to the viewplane. The Angle and Depth options should be set to define the view. The Distance and Height options will be ignored.

Perspective is a type of projection that renders depth of three dimensional objects on a two dimensional plane (your screen) by using suggested lines that intersect in the background to delimit relative size from the background to foreground. But I'm sure you knew that. The Distance and Height options are required to define the perspective. The Angle and Depth options are ignored.

Distance =

Sets the distance from which you are viewing the 3-dimensional data in your drawing. The distance is measured from the world pivot point as set by the Rotate World command. This option is only used when the Type option is set to Perspective.

Height =

Sets the height from which you are viewing the 3-dimensional data in your drawing. The height is measured from the current viewing angle. This option is only used when the Type option is set to Perspective.

Angle =

Sets the angle of an oblique view. A valid Angle value is any number between -360 and 360. This option is only used when the Type option is set to Oblique. See the **Common Options** section for more information on entering angle values.

DEpth =

Sets the foreshortening scale of an oblique view. A valid Depth value is any number greater than zero. This option is only used when the Type option is set to Oblique.

Examples

Project Perspective Dist=100 Height=10

The 3 dimensional data on your drawing will be projected as a perspective that is being viewed from a distance of 100 units and a height of 10 units. The angle that the data is being viewed from can be controlled using the View or Move World commands.

P Commands

PJ o A=30

The 3 dimensional data on your drawing will be projected as an oblique. That Angle of the oblique view will be thirty degrees. The depth will default to the last used value (which is 3/4 at startup).

PROMpt or PM

Macro

When executing macros, it is useful to be able to issue prompts (or requests) to the user to request user input. This prompt may be just a message or may require the user to make a decision or enter data that will be used in the macro. In this way, a single macro may be used in many different situations. The Prompt command allows you to issue such prompts. This command will also remember the response of the operator (if any) in a variable for later use. The operator response can be in the form of a decision, text, a number, or the location of a point. This command can be used at any time, but has little value when used outside of a macro.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Prompt	= string	1	last	%PM.P	=``
Variable	= name	2	last	%PM.V	=A
Type	= NOne or String or SWitch or Number or Coordinate or Object	3	last	%PM.T	=None
String coordinates	= string	4	last none	%PM.S none	=`>>`

Definition of Options

Prompt =

Defines the prompt will appear on the Prompt line or in a Macintosh dialog box, depending on the value entered for the "Type=" option. The String value can be any alphanumeric string up to 32 characters in length. Be sure to quote strings that contain spaces or special characters (like this` or \like this\).

Variable =

This option specifies the name of the variable that will remember the operator's response to the prompt. The operator may respond by typing in information or locating a point. See the Macro Language section for information on variable names. TurboCAD will automatically use the Global Variable "A" if no variable name is entered.

Type =

This option sets the the type of response which will be required of the user. This response will be stored in the variable specified above. Valid Type values are None, String, Switch, Number, Coordinate and Object. See the **Macro Language** section for more information on variable types.

None means no value will be stored into a variable. This value is used if you just wish to issue a prompt and need no user input. The prompt will appear on the TurboCAD command line.

String means the operator response will be typed in and remembered as a character string. If the variable defined already exists and contains a string, the existing string will be the default. The prompt defined above, along with a text entry box, will appear in a screen dialog.

Switch means the operator response must be a switch value, or a yes/no decision. A screen dialog will appear with the appropriate Yes and No buttons. Switch values are: True or False, Yes or No, On or OFF, All or None. The value stored into the variable will be 1 for True, Yes, On and All, and 0 otherwise.

Number means the operator response will be typed in and remembered as a number. If the variable defined already exists and contains a number, the existing number will be the default. The prompt defined above, along with a number entry box, will appear in a screen dialog.

Coordinate means the operator response will be to locate a point. The crosshairs are automatically displayed so the operator may locate a point. The X and X axis components and the key are remembered in the same variable. The X, Y and key are also remembered in independent variables. For example, a coordinate variable called "Point", also automatically sets the variables Point.x, Point.y and Point.k. The variable Point can be used by itself to represent an entire point wherever one is required.

Object means the operator response will be to locate an existing object on the drawing. The crosshairs are automatically displayed so the operator may pick the object. The internal object identification number is remembered in the variable. This identification number is only valid until the next "Display Regenerate". This variable can be used like a coordinate.

String =

This option allows you to temporarily change the prompt line string while a macro is executing. It is sometimes desirable to have a prompt that is different than the standard prompt string so the operator can tell when he is in various stages of a macro. See the Preferences command information on the standard prompt string. [This feature is not available on Macintosh computers.](#)

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

P Commands

Examples

Prompt Prompt=`Locate the Origin` Var=Origin Type=Coord

The prompt string "Locate the Origin" will appear on the Prompt line and the crosshairs will be displayed. When the operator locates a point, its coordinates will be remembered in the variable "Origin".

Pm , `Enter Door Size:` , &DoorSize , Number

The prompt string "Enter Door Size:" will appear in a screen dialog along with a numerical entry box. When the operator enters a value and exits the dialog, the value will be remembered in the variable "&DoorSize" as a number.

pm \Continue? (y/n)\ Typ=sw

The prompt string "Continue? (y/n)" will appear on a screen dialog along with Yes and No buttons. The operator's response will be remembered in the default variable "A" as variable type "Switch".

prompt `The Macro is Working ...` Ty=none

The prompt string "The Macro is Working ..." will appear on the command line. This type of command is a typical command in a macro where the command prompts have been turned off via the Preferences command (for faster execution) and the user wants a relevant prompt to appear during macro execution.

PUt or PU

2D & 3D

The Put command stores objects from your drawing onto disk. Objects stored on disk are called "symbols."

By using the Scale option you can specify if the symbol is to be automatically scaled upon retrieval to match the scale of the drawing you are working on.

Symbols are retrieved from disk for placement on your drawing with the Get command. See the Get command for more information.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Symbol	= name	1	none	%PU.S	=none
Scale			none	%PU.SC	
Force			none	%PU.F	
coordinates			none		

Definition of Options

Symbol =

This option specifies the file name of the symbol when it is stored on disk. The symbol name can be any alphanumeric character string up to 32 characters. Remember to quote symbol names that contain spaces or special characters (like this` or \like this\). Symbols will be saved into the Current directory unless a complete path is entered.

Scale

This keyword-only option specifies that the symbol being saved is a scaled symbol. A scaled symbol is one that is expected to change in size as it used on drawings of different scales. An example of a scaled symbol might be a door. A three foot door symbol is twice as large on a 1/4"=1'0" scale architectural drawing than on a 1/8"=1'0" scale drawing. An example of a non-scaled symbol might be a callout bubble. One would expect a callout bubble to be the same size on drawings of any scale.

Force

This keyword-only option forces the saving of the symbol by bypassing the overwrite safety question. Use of this option will automatically overwrite any existing file on disk of the same name without warning you that it is doing so. This option is most commonly used in macros where operator intervention is not desired.

Warning: Use of this option can cause loss of data with no warning.

coordinates

See Locating Points and **Entering Coordinates** for more information.

Examples

Put Scale Symbol=\3x6 Desk\

You will be prompted to pick an existing object on your drawing. It will be saved to disk into the Current directory with the file name "3x6 Desk" as a scaled symbol.

pu HardDisk:Symbols:valve

You will be prompted to pick an existing object on your drawing. It will be saved to disk with the file name "valve" as a non-scaled symbol into the folder "Symbols" on the volume "HardDisk".

Quit or QT

2D & 3D

The Quit command clears your workarea and exits (terminates) the TurboCAD program. If changes have been made to your drawing since you last saved it to disk, you will be asked if you wish to save the changes before exiting.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Force			none	%QT.F	

Definition of Options

Force

Using this keyword-only option will force TurboCAD to terminate immediately, bypassing the safety message that gives you the opportunity to save any changes that have been made since your drawing was last saved to disk.

Warning: Use of this option is very dangerous as it can cause loss of data with no warning.

Examples

Qu

TurboCAD will be terminated. You will be given an opportunity to save your current workarea if you have made changes to the drawing since the last time you saved it.

q,f

TurboCAD will be terminated immediately. The current drawing workarea will not be saved. You will not be warned that your drawing is being thrown away.

READline or RL

Macro

It is sometimes necessary, especially in macros, to be able to "read in" and use the contents of a text file external to TurboCAD. The Readline command does just that, one line at a time. It will read from the keyboard, an existing string, or an ASCII file on disk and place the contents into the Current Arguments. The Current Arguments are a set of ten pre-defined TurboCAD variables into which the contents are placed.

When this command reads a line, spaces and tabs are read as delimiters. If the line contains variables, variable substitution is always performed. That is, the contents of the variable are placed into the Current Argument rather than the variable name. Calculator expressions are not performed before the Current Arguments are set unless specifically asked for by setting the Evaluate option.

The Current Arguments are global variables named %Arg0 through %Arg9. The first value (up to the first delimiter) read is placed in the variable %Arg1, the second in %Arg2, and so on up to %Arg8. The variable %Arg9 will contain the remainder of the line being read, including delimiters. %Arg0 will be set to the number of values read. The Current Arguments remain set until the next time the Readline command is used. If reading from a text file, each consecutive use of this command will cause the next line in the file to be read. When the end of the file is reached, the argument %Arg0 will be set to "-1."

Refer to the Writeline and File commands for related topics.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Filename	= name	1	none	%RL.F	= none
Prompt	= string	2	last	%RL.P	= none
Statement	= string	3	none	%RL.S	= none
STRing	= string	4	last	%RL.STR	= `>>`
Evaluate	= switch	5	No	%RL.E	= No

Definition of Options

Filename =

Specifies the disk file that will be read from. If the complete path to the file is entered, only the specified folder (directory) will be searched for the file. If only the file name is entered, and not the path, then the folders (directories) specified as the Current directory and the one specified for Macros in the Path command will be searched. If the file name contains spaces or special characters it must be quoted (`like this` or \like this).

A file must be "open" to TurboCAD in order for it to be read from. See the File command for information on opening and closing text files. If not already open, this command will cause the file to be opened automatically.

The first line of the file will be read and the values read will be placed in the Current Arguments. The second time the same file is called, the second line of the file will be read; and so on. If another read is attempted after the entire file has been read %Arg0 will be set to -1.

If this option is not used, input will be read from the keyboard. When reading from the keyboard, it is a good idea to use the Prompt option to send a prompt to the operator so he knows that keyboard entry is expected. This option will override the "Statement=" option below.

Q-R Commands

Prompt =

Sends a prompt to the operator. This option should be used when the user wishes to type in (from the keyboard) the line to be read. It is used to prompt the operator that now is the time to type in the line. If the Prompt string contains spaces or special characters it must be quoted (like this` or \like this\).

Statement =

Specifies the string that will be read. The contents of the string will be read and the values will be placed in the Current Arguments. If the Filename and Statement options are both used, the Filename option wins; this option will be overridden by the Filename option. This option is most commonly used when the user wishes to put the contents of a line into the Current Arguments "on the fly."

STRing =

Defines the TurboCAD command line prompt string. The prompt string appears at the beginning of the command line ahead of the commands that are typed in. The default Readline prompt is " >> ." This may be changed to any prompt string you desire up to ten characters. If numbers or spaces are to be used, the string should be quoted (like this` or \like this\) so it will not be interpreted as a calculation or another option. Macintosh computers do not support this feature.

Evaluate =

This option determines if the line being read should be evaluated for calculator expressions. The Evaluate value must be a switch value. Switch values are: True or False, Yes or No, On or Off, All or None. A positive value means calculator expressions will be evaluated before the Current Arguments are set.

Examples

READ Prompt=`Enter your name:`

The string "Enter your name:" will appear on the prompt line. The program will wait for the operator to type in some information and a <Return>. Let's assume that the operator enters "Joe Bloe." The string "Joe" will be placed in the variable %Arg1 and "Bloe" will be placed in %Arg2. The variable %Arg0 will be set to the number 2 because two values were entered.

rl Addresses

TurboCAD will search for the file named "Addresses" in the Current Directory and in the directory specified for Macros in the Path command. If found, the file will be "opened" (made accessible to TurboCAD). The first line of the file will be read into the Current Arguments. Let's assume that the file looks like this:

Jane A. Doe, 5 Circle Drive, Boston, MA 23232 617/555-5678

Peter P. Piper, 1900 Cliff Blvd., Oakland, IL 12345 212/555-3112

Joe M. Bloe, 123 Main Street, Centerville, CA 98765 714/555-0098

The values read from line one of the file would be assigned as follows:

```
%Arg0 = 9
%Arg1 = Jane
%Arg2 = A.
%Arg3 = Doe,
%Arg4 = 5
%Arg5 = Circle
%Arg6 = Drive,
%Arg7 = Boston,
%Arg8 = MA
%Arg9 = 23232 617/555-5678
```

You can see that the line was divided at each space and the values placed in successive Current Arguments. Note that %Arg9 contains the remainder of the line even though spaces are present.

With the Evaluate option set the values would be assigned as follows:

```
%Arg0 = 9
%Arg1 = Jane
%Arg2 = A
%Arg3 = .
%Arg4 = Doe
%Arg5 = 5
%Arg6 = Circle
%Arg7 = Drive
%Arg8 = Boston
%Arg9 = , MA 23232 617/555-5678
```

Tip: If you have a condition like the one above where there is more information on the line than will fit in eight arguments, here is a trick for reading the rest of the line. Move the contents of Current Arguments 1 through 8 into different variables (Example: Number1 = %Arg1, etc.) and then use the Statement option to read the contents of %Arg9 back into %Arg1 through %Arg9. The line in the macro might look like: "If (%Arg0 == 9) Readline Statement=%Arg9". See the next example.

Q-R Commands

read,,, \%Arg9\

Assuming that this example follows the previous one, with Evaluate active the contents of variable %Arg9 would be read and assigned as follows: %Arg0= 2, %Arg1=23232 and %Arg2=-5676.8883. The value stored in %Arg2 is the result of the calculation 617 divided by 555 minus 5678. With no evaluation %Arg2=617/555-5678.

Rea S=%ca.string

The result of the last calculator expression will be placed in the variable %Arg1. The variable %Arg0 will be set to 1.

RECOrd or RC

2D & 3D

The Record command will record all commands and points that are executed by the operator and write them to an ASCII text file on disk. This file can then be executed as a macro to play back the operations originally performed. Only one recording can be made at a time. Starting a new recording will automatically terminate a recording that is currently being done.

Because the recorded file is in ASCII format, it can be edited with any word processor. It can also be edited using the TurboCAD Modifytext command. Looking at a recorded macro file is very helpful in learning how macros are formatted. It is also a good starting point for developing a more complex macro.

Tip: You can use the Tracking and Keys options of the Preferences command to play back a macro in simulated real time so that you can see each step of the macro being executed.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Active	= switch	1	last	%RC.A	=none
Filename	= name	2	last	%RC.F	=none
FOrce			none	%RC.FO	

Definition of Options

Active =

Turns the recorder on and off. The Active value must be a switch value. Switch values are: True or False, Yes or No, On or Off, All or None.

Filename =

Specifies the disk file that will be written to. If the filename already exists on disk, you will be asked if it should be overwritten. If the file name contains spaces or special characters it must be quoted (`like this` or \like this\).

FOrce

This keyword-only option, when used, will override the safety question if you specified a file name which already exists. This file will be overwritten.

Warning: Using this option may cause loss of data.

Examples**Record On `Architectural Demo`**

The recorder will be turned on. Starting with the command following this one and ending when the recorder is turned off, all operations will be written into a disk file named "Architectural Demo". Any recording that may currently be ongoing will be terminated.

rc n

The current recording will be turned off.

Rectangle or RE**2D**

The Rectangle command creates rectangles on your drawing. The size and angle of the rectangle can be defined by locating points or entering values.

Each undo (<%) key) will remove the last point located on the current rectangle. Each Undo past the current rectangle will remove one complete rectangle. The Undo buffer will remember the last 63 rectangles drawn with this use of the command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Width	= number	1	none	%RE.W	=1
Height	= number	2	none	%RE.H	=1
Angle	= angle	3	none	%RE.A	=0
	or Points (or Pts)				
Construction	= Diagonals or Centerpoint	4	Diagonals	%RE.C	=Diagonals

Q-R Commands

Class	= 0 thru 255	5	%WK.C	%WK.C	=0
Layer	= 0 thru 255	6	%WK.L	%WK.L	=0
Pen	= 0 thru 255	7	%WK.P	%WK.P	=0
STyle	= style	8	%WK.S	%WK.S	=Solid
PRiority	= 0 thru 255	9	%WK.PR	%WK.PR	=125
WOrkset	= workset	10	%WK.W	%WK.W	=none
Mimic			none	%RE.M	
Xsize	= number		none	%RE.W	=none
Ysize	= number		none	%RE.H	=none
coordinates			none		

Definition of Options

Width =

Defines the width (X axis size) of the rectangle. You may enter any non-zero value. If no height value is entered, it will be set the same as the width value entered. If the Construct option is set to Diagonals, the rectangle is placed by locating one corner. Otherwise, the center of the rectangle is located.

Height =

Defines the height (Y axis size) of the rectangle. You may enter any non-zero value. If no Width value is entered, it will be set the same as the Height value entered. If the Construct option is set to Diagonals, the rectangle is placed by locating one corner. Otherwise, the center of the rectangle is located.

Angle =

This option sets the rotation angle of the rectangle from horizontal. A valid Angle value is any number between -360 and 360, or -2^1 and 2^1 radians or the word "points" (or Pts). If points is entered, then you will be prompted to locate a point (typically the second point entered) to determine the angle. This will be the second point entered. See the **Common Options** section for more information.

Construction =

This option sets how the rectangle will be constructed. Valid Construction values are:

Diagonals means the rectangle will be constructed by locating its diagonally opposite corners. The angle will be determined by the value entered for the "Angle=" option above. If the Width and Height options are set, only one corner point is located, followed by a point for the angle if the "Angle=" option is set to points.

Centerpoint means the rectangle will be created by locating its centerpoint and one corner. The angle will be determined by the value entered for the "Angle=" option above. If the Width and Height options are set, only the centerpoint is located, followed by a point for the angle if the "Angle=" option is set to points.

CLass =

See **Common Options** for more information.

Layer =

See **Common Options** for more information.

Pen =

See **Common Options** for more information.

STyle =

See **Common Options** for more information.

PRiority =

See **Common Options** for more information.

WOrkset =

See **Common Options** for more information.

Mimic

See **Common Options** for more information.

Xsize =

This option is an alias for the Width option and it performs exactly the same function. It is here because some of us are used to typing X and Y keywords.

Ysize =

This option is an alias for the Height option and it performs exactly the same function. It is here because some of us are used to typing X and Y keywords.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

RECTANGLE

Rectangles will be drawn by two points that are the diagonally opposite corners of the rectangle. The Angle value will default to zero.

re h=2 x=4 c=c a=45

A rectangle four units wide and two units high will be drawn at an angle of forty-five degrees by locating its centerpoint.

re h=2 x=4 c=c a=Pts

This is the same example as the one above, except that the angle of the rectangle will be set by entering a point. You will be prompted to locate the centerpoint, then you will be prompted to locate a point to define the angle. The angle will be the angle defined by the two points you have entered.

Q-R Commands

R,-12.5

A rectangle twelve and one half units square will be drawn unrotated by locating its upper right corner.

REPlace or RP

2D & 3D

The Replace command will substitute one object on your drawing with another. The new object can be picked from your drawing or retrieved from disk. If the object being replaced has multiple copies, you can replace one, some, or all of the copies. The objects being replaced can be specified by selecting them or by their tag. See the Tag and Group commands for more information on tags.

The rotation and scale of the object being replaced will automatically be applied to the symbol replacing it. Objects are replaced by overlaying the origins of the old and new object.

Warning: The results may be unpredictable if you attempt to replace a 2-dimensional object with a 3-dimensional one, or vice versa, because the Viewplane and World coordinate systems are not coincidental.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Symbol	= name	1	none	%RP.S	=none
Tag	= name	2	none	%RP.O	=none
SElect	= select	3	%PF.S	%RP.SE	=Pick
Global	= global	4	%PF.G	%RP.G	=Ask
Field	= switch	5	No	%RP.F	=No
coordinates			none		

Definition of Options

Symbol =

Specifies the name of the symbol that will be retrieved from disk. This is the symbol that you want to replace others on your drawing. If the complete path to the symbol is entered, only the specified folder (directory) will be searched for it. If only the symbol name is entered, and not the path, then the folders (directories) specified as the Current directory and the one specified for Symbols in the Path command will be searched. If the symbol name contains spaces or special characters it must be quoted (`like this` or \like this\). If this option is not used, you will be prompted to pick an object from your drawing.

Tag =

Specifies the tag of the objects that are to be replaced. If the tag contains spaces or special characters it must be quoted (like this` or \like this\). If this option is not used, you will be prompted to select the objects on your drawing that you want replaced.

SElect =

See **Common Options** for more information.

Global =

See **Common Options** for more information.

Field

This option determines if the text fields that are part of the replacing object are to be filled. The Field value must be a switch value. Switch values are: True or False, Yes or No, On or Off, All or None.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples**Replace**

You will be prompted to pick the object that you want to replace others, and then the others to be replaced. You will not be given an opportunity to fill the text fields that may be part of the new object.

rp GlobeValve GateValve

TurboCAD will search the Current Directory and the directory set for symbols for the symbol "GlobeValve". If found, the symbol "GlobeValve" will be retrieved from disk and will replace all objects on your drawing that have the tag "GateValve". You will not be given an opportunity to fill the text fields that may be part of the new object.

Rep S=\HardDisk:Diffusers:500cfm Diffuser\ Se=In F=T

TurboCAD will search the folder (directory) "Diffusers" within the volume "HardDisk" for the symbol "500cfm Diffuser". If found, the symbol "500cfm Diffuser" will be retrieved from disk. It will replace all objects that are completely inside a box that you define by locating its diagonally opposite corners. You will be prompted to fill the text fields of the new symbol as it replaces each object on your drawing.

RETurn or RN**Macro**

The Return command terminates the macro currently being executed and returns control to the next higher level. The next higher level could be a macro that called the current macro, or the TurboCAD command line.

Options

This command has no options.

Examples

Ret

Terminates the current macro. Control is returned to TurboCAD or the macro that called the current macro.

REVolve or RV

3D

The Revolve command will make repeated copies of an object radially about any one of, or combination of, the three principal (X, Y, Z) axes. If a 2-dimensional object is selected for revolution, it will automatically be converted into a wireframe. Up to 127 copies can be made at any angular increment. The nodes (vertices) of the copies can be connected with wireframes, similar to the Extrude command.

A simple example is that of taking a semicircle in the X-Y plane, whose diameter coincides with the Y axis, and revolving it about the Y axis. If you were to make 35 copies of it, each at a 10 degree increment, you would end up with something which looks like a sphere, especially if you chose to connect the vertices. Before doing the revolution, this command would convert the semicircle into a wireframe.

Each undo (<%> key) will remove the last copy or set of copies made. The Undo buffer will remember the last 128 revolved copies with this use of the command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Xangle	= angle	1	0	%RV.X	=0
Yangle	= angle	2	0	%RV.Y	=0
Zangle	= angle	3	0	%RV.Z	=0
Scale	= number	4	1	%RV.S	=1
Pivot	= Point (or Pt) or Plck	5	Point	%RV.P	=Point
SYstem	= Viewplane or World	6	Viewplane	%RV.SY	=Viewplane
Mode	= Relative	7	Relative	%RV.M	=Relative

	or Overall				
Order	= XYZ	8	last	%RV.0	=XYZ
	or XZY				
	or YXZ				
	or YZX				
	or ZXY				
	or ZYX				
Repeat	= 1 thru 127	9	1	%RV.R	=1
Connect	= switch	10	Yes	%RV.C	=Yes
coordinates			none		

Definition of Options

Xangle =

Sets the incremental angle about the X axis that the selected object will be revolved. This angle is the angle between each copy or the overall distance across all copies as set by the Mode option. The X axis that you are revolving about can be in the Viewplane or World coordinate system as set by the System option. The Xangle value can be any number between -360 and 360 degrees, or -2¹ and 2¹ radians. The "Right Hand Rule" applies to this rotation angle. That is, with your right hand in a fist and your thumb extended along the positive direction of the axis, then positive rotations are in the direction of curvature of your fingers. See the **Common Options** section for more information on angle values.

Yangle =

This option is the same as the Xangle option, except about the Y axis.

Zangle =

This option is the same as the Xangle option, except about the Z axis.

Scale =

This option sets the value that the selected object will be scaled equally in all axes when revolved. The Scale value must be a non-zero number. The Scale value is incremental: you are defining the amount of scaling that is to be done from one copy to the next, not the overall scale from the first copy to the last.

Pivot =

Sets the point that object will be revolved about. Valid Pivot values are:

Pick means the point picked on each object to select it is the point that will be used as the pivot point. If objects are selected by box, the center of the box will be used as the pivot point.

Point (or **Pt**) means you will be prompted to locate a common point about which all objects will be pivoted.

System =

Sets the type of coordinate system that will be used. Valid System values are Viewplane and World. See the **Common Options** section for more information.

Mode =

This option determines how revolved copies are to be spaced.

Relative means that the copies will be spaced apart by the angle specified in the Xangle, Yangle, and Zangle options.

Overall means that the angle specified will be divided by the number of copies specified in the Repeat option.

Order =

This option determines the order that object will be revolved about the X, Y and Z axis. The order that an object is revolved can sometimes have an effect on the outcome. Enter the letters X, Y and Z in the order that you would like the rotation for the corresponding axes to be performed.

Repeat =

This option specifies the number of copies that will be made. The Repeat value must be a number from 1 through 127.

Connect =

This option determines if wireframes will connect the nodes (vertices) of the object being revolved. The Connect value must be a switch value. Switch values are: True or False, Yes or No, On or Off, All or None.

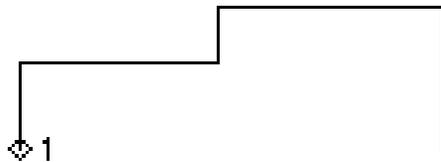
coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

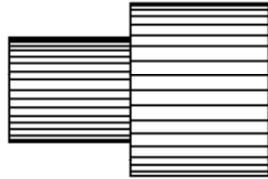
Revolve X=10 Rep=35 piv=pick

You will be prompted to pick an object. Assume the line primitive shown was picked at the point labeled "1."

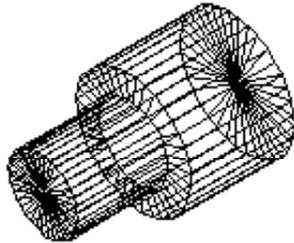


Q-R Commands

The Line will automatically be converted into a wireframe and revolved about point "1." Thirty-five copies, each spaced ten degrees, will be made of the object. The nodes of each copy will be connected with wireframes. The result would look like:

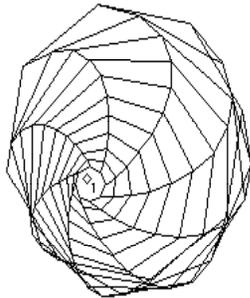


In a SouthWest Isometric view it looks like:



`rv,z=180,rep=10,sc=1.5, mod=ov`

You will first be prompted to locate the pivot point and then to pick the object to be revolved. The object will be copied ten times for an overall angle of 180 degrees (18 degrees between each copy). Each copy will be one and on half times as large as the previous one.



In the example, an octagon was revolved about the point labeled "1".

ROTate or RT

2D

The Rotate command changes the angle (orientation) of 2-dimensional objects on your drawing. In other words, it rotates objects about the Z axis of the Viewplane coordinate system. Rotation can be done dynamically or by specifying an angle. The angle of rotation can be relative (incremental) or absolute.

Q-R Commands

Only complete groups or primitives can be rotated with this command. Members of a group cannot be individually rotated.

Pressing the <Esc> (escape) key will allow you to drag the last object rotated to a new location. The object will be automatically picked by its origin.

Each undo (<%> key) will return the last rotated object or box of objects to their original rotation. The Undo buffer will remember the last 128 objects rotated with this use of the command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Angle	= angle	1	0	%RT.A	=90
Select	= select	2	%PF.S	%RT.S	=Pick
Pivot	= Point (or Pt) or Plck or Object	3	Point	%RT.P	=Point
Mode	= Relative or Absolute	4	Relative	%RT.M	=Relative
Object coordinates	= object	5	%PF.0 none	%RT.0	=Any

Definition of Options

Angle =

This option sets the rotation angle to be applied to the object(s) selected. If no value is entered for this option, you will be able to dynamically rotate the object(s) on your screen. A valid Angle value is any number between -360 and 360 degrees, or -2^1 and 2^1 radians. See **Common Options** for more information on entering angles.

Select =

Sets the way in which objects are to be selected for rotation. See **Common Options** for more information.

Pivot =

Sets the point that object will be rotated about. Valid Pivot values are:

Pick means the point picked on each object to select it is the point that will be used as the pivot point. If objects are selected by box, the center of the box will be used as the pivot point.

Point (or **Pt**) means you will be prompted to locate a common point about which all objects selected will be pivoted.

Object means the origin of each selected object will be used as the pivot point.

Mode =

This option defines the zero angle from which the objects are to be rotated.

Relative means the objects will be rotated incrementally from their current angle.

Absolute means the objects will be rotated using the angle at which they were originally created as the base (or "zero") angle. The current rotation angle, if any, will be disregarded.

Object =

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples**RT**

You will first be prompted to locate the pivot point about which all objects selected will be rotated. Each object picked will become attached to the crosshairs for dynamic rotation on the screen. Clicking the mouse will place each object selected at the current rotation angle.

Rotate Angle=45 piv=pi

You will be prompted to pick the objects you wish to rotate. Each object will be rotated a positive forty-five degrees from its current orientation about the point picked on the object.

Rot 0 M=A S=A

All objects on your drawing will be returned to their originally created orientation.

rt,-90,i,po

You will first be prompted to locate the pivot point about which all objects selected will be rotated. You will then be prompted to define a box by locating its diagonally opposite corners. All objects completely inside the box will be rotated a negative ninety degrees about the pivot point.

ROTate Three or RTT**3D**

The Rotate Three command rotates individual 3-dimensional objects on your drawing. Its complement for 2D objects is the Rotate command. This command should not be confused with the Rotate World command, which rotates the entire 3D world for viewing from a different vantage point.

Each undo (<%> key) will remove the last object or box of objects rotated. The Undo buffer will remember the last 128 objects with this use of the command.

This command can also be entered as "ROTATE_Three" or "RT Three".

Q-R Commands

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Xangle	= <i>angle</i>	1	0	%RTT.X	=0
Yangle	= <i>angle</i>	2	0	%RTT.Y	=0
Zangle	= <i>angle</i>	3	0	%RTT.Z	=0
Pivot	= Point (or Pt) or Pick	4	Point	%RTT.P	=Point
System	= Viewplane or World	5	Viewplane	%RTT.S	=Viewplane
Order	= XYZ or XZY or YXZ or YZX or ZXY or ZYX	6	last	%RTT.O	=XYZ
Global coordinates	= <i>global</i>	7	%PF.G none	%RTT.G	=Ask

Definition of Options

Xangle =

Sets the angle about the X axis that the selected object will be rotated. The X axis that you are revolving about can be in the Viewplane or World coordinate system as set by the "System=" option below. The Xangle value can be any number between -360 and 360. See the **Common Options** for more information on angle values.

Yangle =

This option is the same as the Xangle option, except about the Y axis.

Zangle =

This option is the same as the Xangle option, except about the Z axis.

Pivot =

Sets the point that object will be rotated about. Valid Pivot values are:

Point (or **Pt**) means you will be prompted to locate a common point about which all objects picked will be pivoted.

Pick means the point picked on each object to select it is the point that will be used as the pivot point.

System =

Determines which of the two coordinate systems will be used as the basis for the rotation. Valid System values are Viewplane and World. See **Common Options** for more information.

Order =

This option determines the order that object will be rotated about the X, Y and Z axis. The order that an object is rotated can sometimes have an effect on the outcome. Enter the letters X, Y and Z in the order that you would like the rotation for the corresponding axes to be performed.

Global =

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples**Rotate Three Z=45 piv=pick**

You will be prompted to pick the objects you wish to rotate. Each object will be rotated a positive forty-five degrees about the Z axis of the Viewplane coordinate system. It will be pivoted about the point at which it was picked.

RTT, 90,,90 s=w g=a

You will first be prompted to locate a pivot point and then to pick the objects you wish to rotate. Each object will be rotated a positive ninety degrees about the X and Z axes or the World coordinate system. All picked objects will pivot about the same pivot point. If there are copies of a rotated 3-dimensional group, they will also be rotated.

rotate_t Pivot=point System=World Order=zyx

You will first be prompted to locate a pivot point and then to pick the objects you wish to rotate. Then nothing will happen because you forgot to give an X, Y or Z axis angle value. Oh, yeah.

ROTate World or RTW**3D**

The Rotate World command rotates the entire 3-dimensional world, allowing you to view it from a different angle. Only 3-D data is affected (i.e. wireframes). Rotations can be performed using either the Viewplane or World coordinate system. See **Common Options** for more information on the two systems.

The effects of this command cannot be undone using the <%> key.

This command can also be entered as "ROTATE_World" or "RT World."

Q-R Commands

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Xangle	= angle	1	0	%RTW.X	=0
Yangle	= angle	2	0	%RTW.Y	=0
Zangle	= angle	3	0	%RTW.Z	=0
Pivot	= CUrrent or Center or Point (or Pt)	4	Current	%RTW.P	=Current
System	= Viewplane or World	5	Viewplane	%RTW.S	=Viewplane
Mode	= Relative or Overall	6	Relative	%RTW.M	=Relative
Order	= XYZ or XZY or YXZ or YZX or ZXY or ZYX	7	last	%RTW.O	=XYZ
	coordinates		none		

Definition of Options

Xangle =

Sets the angle about the X axis that all 3-dimensional data will be rotated. The X axis that you are rotating about can be in the Viewplane or World coordinate system as set by the "System=" option below. The Xangle value can be any number between -360 and 360. See the **Common Options** for more information on angle values.

Yangle =

This option is the same as the Xangle option, except about the Y axis.

Zangle =

This option is the same as the Xangle option, except about the Z axis.

Pivot =

Sets the point the World will be rotated about. Valid Pivot values are:

CUrrent means the World will rotate about the current world pivot point. This is the same pivot point as was last used.

Center means the centerpoint of all 3-dimensional data will be calculated and used as the pivot point. This point will become the new world pivot point.

Point (or **Pt**) means you will be prompted to locate a pivot point. This point will become the new world pivot point.

System =

Determines which of the two coordinate systems will be used as the basis for the rotation. Valid System values are Viewplane and World. See **Common Options** for more information.

Mode =

This option defines the zero (or base) angle from which the world is rotated.

Relative means the world will be rotated incrementally from its current angle.

Overall means the world will be rotated from its absolute zero angle. When all axes of the World coordinate system are set to zero, they are perfectly aligned with the Viewplane Coordinate system axes. In other words, "View Plan" is the absolute zero setting for the World coordinate system.

Order =

This option determines the order that world will be rotated about the X, Y and Z axis. The order of the rotation can sometimes have an effect on the outcome. Enter the letters X, Y and Z in the order that you would like the rotation for the corresponding axes to be performed.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Rotate World Y=90

All 3-dimensional data in your drawing will be rotated a positive ninety degrees about the Y axis of the Viewplane coordinate system, which is vertical on the plane of your screen. The world will be pivoted about the current world pivot point (the last one used). The world will rotate to the right (which means you will be looking at the left side of the world data).

rtw z=-45 sys=w piv=p

You will be prompted to locate a pivot point. All 3-dimensional data in your drawing will be rotated a negative forty-five degrees about the Z axis of the World coordinate system.

rot w 0 0 0,,a

The World coordinate system will be aligned with the Viewplane coordinate system. You are now looking at the Plan view.

Rounding or RD

2D

The Rounding command sets the distance increment to which the TurboCAD crosshairs will "jump" or "snap" to on the screen. This is commonly referred to as cursor rounding. Cursor rounding is independent of reference grids, although they can be made to coincide. Cursor rounding can be toggled active/inactive at any time. In effect, cursor rounding controls the location of points placed randomly (i.e. not with any of the "locate keys", such as <V>, <M>, etc.) with the crosshairs. When used in conjunction with the "locate keys" and with the ability to toggle rounding active/inactive, Cursor Rounding should be used to facilitate accurate drawing rather than limit it. For more information, refer to the discussion of Rounding and Grids in the **Setting Up a Drawing** chapter in Basics.

Warning: Because rounding always originates at the plot origin of your drawing (the lower left corner) and grids originate at the work origin as set by the Origin command, it is possible to get your grids and rounding out of sync. This means that your crosshairs will not land exactly on the visible grids. To alleviate this problem use "Origin Reset" and always relocate your drawing origin with rounding turned on.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Spacing	= number	1	last	%RD.S	=.125" (.1mm)
Yspacing	= number	2	last	%RD.Y	=.125" (.1mm)
Type	= Orthogonal or Isometric	3	last	%RD.T	=Orthogonal
Active	= switch	4	last	%RD.A	=On
Xspacing	= number		last	%RD.X	=.125" (.1mm)

If a new drawing is created in metric plot units, the spacing value will startup at .1mm instead of .125".

Definition of Options

Spacing =

Sets the distance, in work units, that the crosshairs will jump in both axes. This option will override and reset the Xspacing and Yspacing options. This option sets the spacing for both types of rounding.

Yspacing =

Sets the distance, in work units, the crosshairs will jump along the Y axis.

Type =

Sets the type of grids to be displayed. Valid rounding Type values are:

Orthogonal means that rounding is done along the X and Y axis of your screen. This is the type of rounding that corresponds to Full and Ruler type grids.

Isometric means that rounding is done at 30 and -30 degree angles. The Spacing option sets the spacing of isometric grids as measured along the 30 degree grid line. This is the type of rounding that corresponds to Isometric type grids. This type of rounding is not currently supported on Macintosh computers.

Note: Changing the rounding type does not automatically change the grid type. If you are changing from orthogonal to isometric rounding, or vice versa, be sure to set your grid to the proper type.

Active =

Turns rounding on and off. The Active value must be a switch value. Switch values are: True or False, Yes or No, On or Off, All or None. The rounding values are remembered for when the rounding is activated again. Internally this is accomplished by setting the values to a negative number. The numbers are automatically made positive when the rounding is reactivated. This option performs the same function as pressing the <R> key while in the Graphic mode to toggle the rounding on and off.

Xspacing =

Sets the distance the crosshairs will jump along the X axis when no value is entered for the "Spacing=" option and you want to specify a different value for rounding along each axis.

Examples**Round 1/4**

Rounding will be set to a distance of 0.25 work units in both the X and Y axes. The type will remain unchanged.

Round 1/4"

Rounding will be set to a distance of 1/4" in both the X and Y axes regardless of the current work units. The type will remain unchanged.

rd,1,2,0

Rounding will be set to one work unit along the X axis and two work units along the Y axis. Rounding type will be set to Orthogonal.

ro a=n

Rounding will be deactivated. The current Type and Spacing values are remembered for when rounding is reactivated.

Save or SV

2D & 3D

The Save command stores your current drawing on disk. The complete drawing or any specific combination of classes can be saved.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Drawing	= name	1	last	%SV.D	=none
Classes	= classes or All or Display or Used or Edit	2	All	%SV.C	=All
Force			none	%SV.F	

Definition of Options

Drawing =

This options specifies the file name of your drawing when it is stored on disk. The drawing name can be any alphanumeric character string up to 60 characters. If the complete path to the file is entered (For example, Save Drawing=HD:Drawings:Schematic), it will be saved in the specified folder (directory). If only the file name is entered, it will be saved in the current folder. Remember to quote file names that contain spaces or special characters (like this` or \like this\).

Classes =

Sets the classes that are to be saved. A valid Classes value is a list of classes or the words "All", "Display", "Edit" or "Used".

classes is a list of the classes that you wish to have saved. There are 256 classes on each drawing, numbered 0 through 255. List the classes you wish to have saved, separating each number with a space or comma. Ranges of classes can be entered by separating the starting class and ending class with an ellipsis (three periods, like this: 10...20). Individual classes and ranges of classes are negated by preceding them with an up arrow (shift-6 on the keyboard, like this: ^).

All means that all classes will be saved.

Display means only the classes currently being displayed will be saved.

Edit means only the classes that are currently editable will be saved.

Used means only the classes that currently contain data will be saved.

Force

This keyword-only option forces the saving of the drawing by bypassing the overwrite safety question. Use of this option will automatically overwrite any existing file on disk of the same name. This option is most commonly used in macros where operator intervention is not desired.

Warning : Use of this option can cause loss of data with no warning.

Examples

Save \Floor Plan\

All classes of your current workarea will be saved to disk as "Floor Plan" in the current directory.

sa `HardDisk:Project A:PID-0125` 100...200,250

Classes 100 through 200 and class 250 will be saved as "PID_0125" in the folder "Project A" on volume "HardDisk."

SV,Fo,Cl=Ed,Dr=Elevation

Only the editable classes will be saved as "Elevation" in the current folder. If a file exists with that name, it will be overwritten without warning you.

Scale or SC

2D

The Scale command changes the size of 2-dimensional objects on your drawing. Objects can be scaled differently in each axis, if desired. Only complete groups or primitives can be scaled with this command. Members of a group cannot be individually scaled.

Pressing the <Esc> (escape) key will allow you to drag the last object scaled to a new location. The object will be automatically picked by its origin.

Each undo (<%> key) will return the last scaled object or box of objects to their original size. The Undo buffer will remember the last 128 objects scaled with this use of the command.

Note: Scaling is always done along the object's X and Y axes, not the drawing's axes. When an object is rotated its axes are also rotated. Regrouping an object will realign its axes with the drawing's axes because it is then considered a new object that has not yet been rotated.

S Commands

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Scale	= number	1	1	%SC.S	=1
SElect	= select	2	%PF.S	%SC.SE	=Pick
Pivot	= Point (or Pt) or Plck or Object	3	Point	%SC.P	=Point
Mode	= Relative or Absolute	4	Relative	%SC.M	=Relative
Object	= object	5	%PF.O	%SC.O	=Any
Xscale	= number		1	%SC.X	=1
Yscale	= number		1	%SC.Y	=1
coordinates			none		

Definition of Options

Scale =

This option sets the scale that selected objects will be scaled along the object's X and Y axes. A valid Scale value is any number, except zero. Values greater than 1 mean enlargement, and values less than 1 mean making smaller. Negative values mean that the object will be mirrored about the pivot point (defined below) as well as scaled by the absolute value of the value entered. This option will override and reset the Xscale and Yscale options.

If no value is entered for this option and for the Xscale and Yscale options, all other options will be disregarded. In this case, you will be prompted to select an object for scaling. It will appear with "handles" on a Macintosh screen. You may then grab any of these handles and scale the object dynamically.

SElect =

This option determines how objects are to be selected. Valid Select values are Pick, Inside, Outside, Touching, Notinside, NOTOutside, NOTTouching and All. See **Common Options** for more information.

Pivot =

Sets the point that object will be scaled about. Valid Pivot values are:

Point (or **Pt**) means you will be prompted to locate a common point about which all objects will be scaled.

Pick means the point picked on each object to select it is the point that will be used as the pivot point. If objects are selected by box, the center of the box will be used as the pivot point.

Object means the origin of each selected object will be used as the pivot point.

Mode =

This option defines the scale from which further scaling will be done.

Relative means the objects will be scaled from their current scale.

Overall means the objects will be scaled from their originally created scale.

Object =

See **Common Options** for more information.

Xscale =

This option sets the value that objects will be scaled along the object's X axis. A valid Xscale value is any number other than zero.

Yscale =

This option sets the value that objects will be scaled along the object's Y axis. A valid Yscale value is any number other than zero.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples**Scale 2**

You will first be prompted to locate a pivot point. Then you will be prompted to pick the objects you wish to scale. Each object will be scaled to twice its current size (the Mode option defaults to Relative) about the common pivot point which you located.

SC Sel=In Piv=Ob X=1/2

You will be prompted to define a box by locating its diagonally opposite corners. All objects completely inside the box will be scaled to one half their current size along along its X axis. Their size along the Y axis will be unaffected and will remain the same. The scaling will be done about the origin of each object.

sca, .75,all,ob,ab ob=t

All text primitives on your drawing will be scaled to three-quarters their original size. Scaling will be done about the individual origin of each text primitive.

Scale Three or SCT**3D**

The Scale Three command changes the size of 3-dimensional objects on your drawing. Scaling is done equally in the X, Y and Z axis.

Only complete groups or primitives can be scaled with this command. Members of a group cannot be individually scaled.

S Commands

Pressing the <Esc> (escape) key will allow you to drag the last object scaled to a new location. The object will be automatically picked by its origin.

Each undo (<%> key) will return the last scaled object to its original size. The Undo buffer will remember the last 128 objects scaled with this use of the command.

This command can also be entered as "SCALE_Three" or "SC Three."

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Scale	= number	1	1	%SCT.S	=1
Pivot	= Point (or Pt) or Pick	2	Point	%SCT.P	=Point
Global coordinates	= global	3	%PF.G none	%SCT.G	=Ask

Definition of Options

Scale =

This option sets the scale that picked object will be scaled about the X, Y and Z axes. A valid Scale value is any number except zero. Values greater than 1 mean enlargement, and values less than 1 mean making smaller. Negative values mean that the object will be mirrored about the Pivot Point (defined below) as well as scaled by the absolute value of the value entered.

Pivot =

Sets the point that object will be scaled about. Valid Pivot values are:

Point (or **Pt**) means you will be prompted to locate a common point about which all picked objects will be scaled.

Pick means the point picked on each object to select it is the point that will be used as the pivot point.

Global =

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Scale Three 2

You will first be prompted to locate the pivot point, then you will be prompted to pick the objects you wish to scale. Each object will be scaled to twice its current size about the pivot point.

SCT Piv=Pi s=1/2

You will be prompted to pick the objects to be scaled. The picked objects will be scaled to one half their current size. The scaling will be done about the point picked on each object.

SElect or SL

Macro

The Select command displays a dialog containing a list of choices and prompts the operator to select one of them by highlighting it with the mouse or cursor. This allows you to present a multiple choice question to the operator and use the operator selection for whatever purpose, such as to branch in to a specific label in the macro to to execute another macro. This command can also be used to display the list without requiring operator response; i.e. for informational purposes only.

The list of choices is user definable. You can consider this a logical extension of a "switch" type prompt in the Prompt command, which only allows for a yes/no response.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Name	= string	1	last	%SL.N	= none
Filename	= name	2	none	%SL.F	= none
Type	= None or List or NUmber	3	last	%SL.T	= None
Prompt	= string	4	last	%SL.P	= none
Result	= string	5	last	%SL.R	= none

Definition of Options**Name =**

This option titles the dialog that will appear on your screen. The Name value can be any alphanumeric character string. If the string contains spaces or special characters it must be quoted (^like this` or \like this\).

Filename =

If this option is used, the contents of the specified file will be displayed as the list of choices. The practical limit to the number of lines is how many will fit on your screen. If this option is not used, input is expected from the keyboard or macro. In this case, each line is displayed as one choice, beginning with the first line after the one which contains the Select command, until a blank line is encountered. The lines will be read from a macro only if this command is

being executed from within a macro. All lines will have variable substitution performed, but calculator expressions will not be processed.

Type =

Specifies the type of list that is to be displayed. Valid Type values are None, List and Number.

Number means the choices in the list will be numbered. The operator is expected to select one of the choices by clicking on it with the cursor or by entering one of the numbers and hitting a <Return>. If the operator enters a number that is not one of those listed, none of the entries will be highlighted. The number corresponding to the operator's choice is remembered in the Current Status Word %SL.Result.

List is the same as number except that the list is not numbered. The operator is expected to select one of the choices by clicking on it with the cursor typing in a character string. The actual string of the choice is remembered in the Current Status Word %SL.Result.

None produces the same type of list as the List value, but without requiring an operator response. This value is used when a list is to be displayed for the operator's information, but no response from the operator is required.

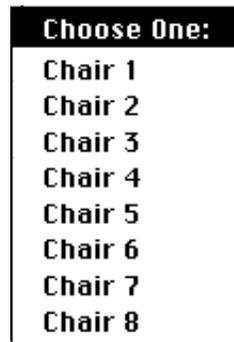
Prompt =

This option defines the prompt that will appear on the Prompt line at the bottom of your screen when the list of choices is displayed. The Prompt value can be any alphanumeric character string up to 32 characters in length. If the string

Examples

Select N=`Choose one:` F=`Chairs` T=L

TurboCAD will search for the file named "Chairs" in the Current Directory and in the directory set for macros. If found, the contents of the file will be displayed in a dialog that is titled "Choose one." The operator is expected to make a choice by clicking on his/her choice with the mouse, by moving to that choice with the cursor and hitting a <Return>, or by entering the first letter(s) of the choice followed by a <Return>. The operator's answer (i.e. the actual text of the choice) is remembered in the Current Status Word "%SL.Result"



S Commands

Suppose part of a macro looked like:

SL N=Branch to...` T=L

Diodes

Capacitors

Resistors

Inductors

/%SL.R



A dialog would appear as shown above. After making the choice as in the first example, the macro would branch to (execute) the macro with the name of the choice since the contents of the choice are stored in the Current Status Word %SL.R.

SEtup or SU

2D & 3D

The Setup command changes the original drawing workarea parameters. These parameters include the drawing height, width, scale and the work and plot units. The drawing name and description can also be changed.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Width	= number	1	last	%SU.W	=34
Height	= number	2	last	%SU.H	=22
Scale	= number	3	last	%SU.S	=1
Mscale	= number	4	last	%SU.M	=1
Work	= unit	5	last	%SU.WO	=Inches
Plot	= unit	6	last	%SU.P	=Inches
Resolution	= Fractional or Decimal or number	7	last	%SU.R	=Fractional
Drawing	= name	8	last	%SU.D	= `` (Untitled)
Description	= string	9	last	%SU.DE	= ``

Definition of Options

Width =

This option changes the width of the "sheet of paper" you are currently working on. We call this the workarea. The width and height of the workarea are measured in plot units (also called sheet units), which are either inches or millimeters depending on the setting of the "Plot=" option. Drawing scale has no affect on the width value. Adjustments to the width of the workarea result in movement of the right side of the sheet in or out.

Height =

This option changes the height of the "sheet of paper" you are working on. Other than that, it is the same as width. Adjustments to the height result in movement of the top of the sheet up or down.

Scale =

This option changes the scale that you wish to work in. Scale is the ratio you want between plot units and work units. A scale of "1" means that there is one work unit on your drawing for each unit on the sheet of paper when plotted. A scale of ".0625" means there are sixteen work units for each sheet unit. This option and the Mscale option below are mutually exclusive. Setting one of them will cause TurboCAD to automatically calculate and set the other. See the New command and the **Setting Up a Drawing** section in Basics for more information on setting a drawing scale.

Changing the Scale does not change the scale of any existing data on your drawing, it sets the scale that you will be working in from this point forward.

Mscale =

This option changes the scale that you wish to work in. Mscale is the ratio you want between the size of an object in real life and its size on your drawing, and is commonly referred to as the Absolute scale. An mscale of "1" means that means that you want to draw objects life size. A Scale (as defined above) of 1/8"=1' corresponds to an Mscale of 1:96. This option and the Scale option above are mutually exclusive. Setting one of them will cause TurboCAD to automatically calculate and set the other. See the New command and the Setting Up a Drawing section in Basics for more information on setting a drawing scale.

Changing the Mscale does not change the scale of any existing data on your drawing, it sets the scale that you will be working in from this point forward.

WORk =

This option changes the units that you will work in. Work units are the units that most distances and sizes are measured in. Valid Work values are:

MILs	or ML	MILLimeters	or MM
Inches	or IN	Centimeters	or CM
Feet	or FT	Meters	or M
Miles	or MI	Kilometers	or KM

S Commands

Changing the work unit name does not change the size of a unit on your drawing, it only changes the name of that unit. For example, if you changed your work unit from inches to millimeters, an object that was previously one inch long is now one millimeter long. In other words, a work unit is work unit, and it doesn't matter what you call it.

Plot =

This option changes the units that you will plot in. Plot units are the units that your drawing sheet is measured in. Valid Plot values are the same as for work units. The most commonly used plot units are inches and millimeters.

Changing the plot unit effectively changes the size of your drawing. This is because the plot units are the ones that define the units that you will be plotting in.

Drawing =

Names your drawing workarea. This is the name that will be used as the default file name when saving your drawing to disk. The drawing name can be any alphanumeric character string up to 60 characters. File names containing spaces or special characters should be quoted ('like this' or \like this\) so they are not read as separate options to the command. Entering a complete path name will ensure that the drawing will be saved in the directory (folder) of your choice.

DEscription =

This is a free field for your use. It can be used for the drawing's revision level, creator's name or what have you. The description can be any alphanumeric character string up to 32 characters. Remember to quote ('like this' or \like this\) descriptions containing spaces or special characters. You can look up the current description by executing the "Status Setup" command.

Examples

Setup Scale=1/4

The scale of your drawing will be set to one quarter plot unit equals one work unit. Existing data will not be rescaled. No other drawing setup parameters will be changed. The Mscale, or absolute scale, will be automatically calculated and set.

SET 40,30

Your sheet size will be changed to forty units wide by thirty units high. No other drawing setup parameters will be changed.

su d='Connection Details' de='Rev. B, 12JUN88'

Your drawing name will be changed to "Connection Details". This is the name that will now be used by the Save command. The Description will be set to "Rev. B, 12JUN88". No other drawing setup parameters will be changed.

The Sketch command creates multi-segment lines on your drawing by automatically recording points as the cursor is moved. The sampling distance can be adjusted to control the smoothness of curves.

Hold down the mouse key to start sketching and release it to stop. If the sketched line would contain more than the maximum number of segments allowed in a line primitive, a new one will be started automatically.

Each undo (<%) key) will remove the last line primitive created. The Undo buffer will remember all Line primitives created with this use of the command.

Rounding is automatically disabled while sketching. View functions can be performed in the Sketch command, but only while not in the process of actually sketching. Locate functions are disabled in this command and cannot be used to locate points. Coordinate entry cannot be used to sketch a line. Sketched lines cannot be recorded into a macro.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Distance	= number	1	last	%SK.D	= .0625
Class	= 0 thru 255	2	%WK.C	%WK.C	= 0
Layer	= 0 thru 255	3	%WK.L	%WK.L	= 0
Pen	= 0 thru 255	4	%WK.P	%WK.P	= 0
Style	= style	5	%WK.S	%WK.S	= Solid
PRiority	= 0 thru 255	6	%WK.PR	%WK.PR	= 125
WORkset	= workset	7	%WK.W	%WK.W	= none
Mimic			none	%SK.M	

Definition of Options

Distance =

Sets the, sampling distance, or minimum length that each segment of the line will be. As soon as the cursor is moved more than this distance from the previously recorded point, a new point is recorded. If the cursor is moved rapidly, points will be recorded farther apart because of computer speed limitations. The Distance value must be a number greater than zero.

Class =

See **Common Options** for more information.

Layer =

See **Common Options** for more information.

Pen =

See **Common Options** for more information.

Style =

See **Common Options** for more information.

PRiority =

See **Common Options** for more information.

WOrkset =

See **Common Options** for more information.

Mimic

See **Common Options** for more information.

Examples**sk 2**

Depress the mouse button to start sketching a line. Points will be recorded at a minimum sampling distance of 2 work units. The Class, Layer, Pen, and Style options will default to Status palette values. Release the mouse button to stop sketching.

SPLine or SP**2D**

The Spline command creates curved lines on your drawing. Splines are created by locating points through which a third degree polynomial spline is drawn.

Each undo (<%) key) will remove the last point located. The Undo buffer will remember the last 256 points located with this use of the command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Class	= 0 thru 255	1	%WK.C	%WK.C	=0
Layer	= 0 thru 255	2	%WK.L	%WK.L	=0
Pen	= 0 thru 255	3	%WK.P	%WK.P	=0
Style	= style	4	%WK.S	%WK.S	=Solid
PRiority	= 0 thru 255	5	%WK.PR	%WK.PR	=125
WOrkset	= workset	6	%WK.W	%WK.W	=none

S Commands

Mimic	none	%SP.M
coordinates	none	

Definition of Options

Class =

See **Common Options** for more information.

Layer =

See **Common Options** for more information.

Pen =

See **Common Options** for more information.

Style =

See **Common Options** for more information.

PRiority =

See **Common Options** for more information.

WOrkset =

See **Common Options** for more information.

Mimic

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Spline

You will be prompted to locate points to create a spline. The Class, Layer, Pen, and Style options will default to Status Palette values.

sp,77 st=cl

You will be prompted to locate points to create a spline. The Spline will be drawn on Class 77 and the Centerline line style. The Layer, Pen and Priority options will default to Status Palette values.

Spl,M,p=10

You will first be prompted to pick an existing spline on your drawing and then to locate points to create a spline. The Spline will be drawn with Pen 10. The Class, Layer, Style and Priority options will default to those of the picked spline.

STatus or SS**2D & 3D**

The Status command looks up and displays the last used settings of a command's options.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Command	= command	1	last command	%SS.C	=none

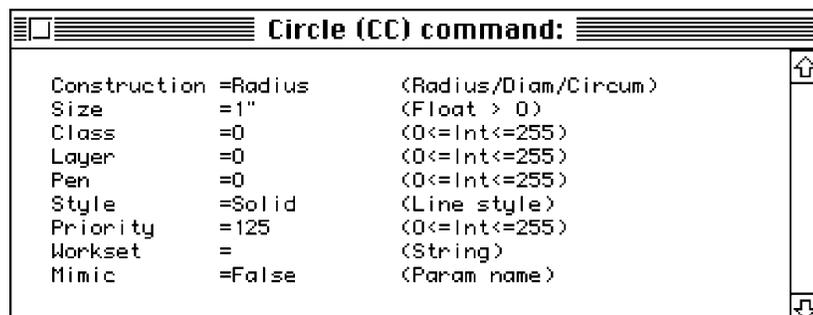
Definition of Options**Command =**

The value for Command can be any TurboCAD command name, abbreviation or code. The last entered values for each option of the command will be displayed.

Last entered means the last time an option was actually entered, not defaulted. Default values will not be shown. Therefore, the values shown are not necessarily the values that will be used when the command is next run. They are the values that were last entered, even if that command was executed since then.

Examples**Status Command=Circle**

The values contained in the Current Status Words for the Circle command will be displayed on your screen in an information window. Shown below is the message you would see on a Macintosh computer.

**STA CIR**

Same as the above example.

SS SS

Displays the last used values of the Status command.

STOp or SO

Macro

The Stop command terminates the macro currently being executed and returns control to the TurboCAD Command Line. All higher levels of macro execution will also be terminated. This means that if a macro called the current macro, control will not return to the calling macro.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
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This command has no options.

Examples

STOP

All levels of macro execution will be terminated. Control will be returned to the TurboCAD interactive command level.

STRetch or SR

2D

The Stretch command moves vertices much like the Move Vertex command, except that you select the vertices by box, thereby allowing you to move multiple vertices at once. If all of the vertices of an object are either inside or outside the box (your choice), then the whole object will move. If some of the vertices of an object are either inside or outside, then only those will move. Objects and/or vertices are selected for stretching by defining a box, and you can affect either vertices inside or vertices outside the box. This makes it possible to move multiple vertices, or objects and vertices, in one edit function. The Convert option allows you to specify if complete objects or partial objects are to be moved.

Each undo (<%> key) will return the last moved objects and/or vertices to their original locations. The Undo buffer will remember the last 40 edits performed with this use of the command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Xdistance	= number	1	0	%SR.X	= 0
Ydistance	= number	2	0	%SR.Y	= 0

Select	= Inside or Outside	3	last	%SR.S	= Inside
Global	= global	4	%PF.G	%SR.G	= Ask
Convert	= switch	5	True	%SR.C	= True
Vector			none	%SR.V	
Distance	= number	6	none	%SR.D	= 0
Angle	= angle	7	none	%SR.A	= 0
coordinates			none		

Definition of Options

Xdistance =

Sets the distance along the drawing (viewplane) X axis that the selected objects will be moved. The Xdistance value can be any positive or negative number. This option is mutually exclusive of the Distance option. If no Xdistance or Ydistance value is entered, the distance will be defined by dragging the box to the new location.

Ydistance =

Sets the distance along the drawing (viewplane) Y axis that the selected objects will be moved. The Ydistance value can be any positive or negative number. This option is mutually exclusive of the Distance option. If no Xdistance or Ydistance value is entered, the distance will be defined by dragging the box to the new location.

Select =

This option determines how vertices/objects are to be selected.

Inside means objects/vertices inside the box will be moved. The box is defined by its diagonally opposite corners. Only objects completely inside the box will be moved unless the Convert option is used.

Outside means objects/vertices outside the box will be moved. The box is defined by its diagonally opposite corners. Only objects completely outside the box will be moved unless the Convert option is used.

Global =

See **Common Options** for more information.

Convert =

This option will convert group members that are rectangles, polygons, arcs or ellipses into a line so that part of a group can be stretched. The Convert value must be a switch value: True or False, Yes or No, On or Off, All or None. A positive value means that group members will be converted into lines so that those members can be stretched at the point that the selection box intersects them. A negative value means that only complete groups will be moved.

S Commands

Vector

This keyword-only option requires the distance and angle of the stretch to be specified by locating two points with the crosshairs. An error will be issued if this option is used at the same time as the X or Y distance options or the Distance and Angle options below.

Distance =

This option, along with the Angle option below, allows you to specify the distance vector numerically. Any positive or negative distance value is valid. An error will be issued if this option is used at the same time as the X or Y distance options or the Vector option.

Angle =

This option works with the Distance option to specify the distance vector numerically. An error will be issued if this option is used at the same time as the X or Y distance options or the Vector option. See **Common Options** for more information on entering angles.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Stretch

You will be prompted to define a box by locating its diagonally opposite corners. You then drag the box to the desired new location. All vertices and whole groups that are completely inside the box will be moved.

Sr,5'3+1/2",,c

You will be prompted to define a box by locating its diagonally opposite corners. All vertices and whole groups that are completely inside the box will be moved five feet three and one half inches to the right. Groups that are bisected by the box will have their members converted into lines as necessary so the portion of the group that falls within the box can be moved.

SR S=I V

You will first be prompted to locate two points to define the distance vector, then you will be prompted to define a box by locating its diagonally opposite corners. All vertices and whole groups that are completely inside the box will be moved by the distance and angle defined by the distance vector. Groups that are bisected by the box will have their members converted into lines as necessary so the portion of the group that falls within the box can be moved.

SR d=3 a=30

You will be prompted to define a box by locating its diagonally opposite corners. All vertices and whole groups that are completely inside the box will be moved a distance of 3 work units at an angle of 30 degrees. Groups that are bisected by the box will have their members converted into lines as necessary so the portion of the group that falls within the box can be moved.

STyle or ST

2D & 3D

The Style command sets the style of line that you wish to draw in. The Style command also defines what a line style looks like.

The current work style is displayed on the Status palette at the bottom of your screen.

There are 8 styles on each drawing, numbered 0 through 7 and named as shown below. Every object on your drawing has a style assignment, just like it has a Class, Layer, Pen and Priority assignment. Styles are a property of each primitive (or group member).

Line styles are defined by setting the length of a long dash, a gap, an optional short dash and the number of repeats for each dash type. Style definitions are measured in plot units, so they are unaffected by your drawing scale.

Options

<u>Keyword</u>	<u>Value</u>		<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Work	= Solid	or SO or 0	1	last	%ST.W	=0
	or Dotted	or DO or 1				
	or Centerline	or CL or 2				
	or Dashed	or DA or 3				
	or Longdash	or LD or 4				
	or Phantom	or PH or 5				
	or LONGCenterline	or LC or 6				
	or LONGGap	or LG or 7				
Long	= number		2	last	%ST.L.n	=0
Gap	= number		3	last	%ST.G.n	=0
Short	= number		4	last	%ST.S.n	=0
LRepeat	= number		5	last	%ST.LR.n	=0
SRepeat	= number		6	last	%ST.SR.n	=0

S Commands

Definition of Options

Work =

Sets the line style that new objects will be drawn in. This is also the style that will be defined if any other options of this command are used, and you can only define one at a time. You must enter a whole number from 0 through 7 or one of the abbreviations or names shown above.

Long =

Sets the length of the long dash. Enter a number greater than zero.

Gap =

Sets the length of the gaps that appear between dashes. Enter a number greater than zero.

Short =

Use this option if you would like a short dash in addition to the long dash (like a centerline, for example). Enter a number greater than zero. Long dashes and short dashes will alternate.

LRepeat =

Sets the number of long dashes that will appear between short dashes. Enter a whole number between 0 and 7. This option is usually set to one if the Short option is not used.

SRepeat =

Sets the number of short dashes that will appear between long dashes. Enter a whole number between 0 and 7. This option is only needed when using the Short option.

Examples

STYLE CENTERLINE

Your work style will be set to Centerline.

st 3 1/4 1/8

Your work style will be set to dashed. The Dashed style will be defined as a repeating pattern of long dashes. Each dash will be one quarter plot unit in length separated by gaps of one eighth plot unit.

Sty,Ph,3,,2,,3,,3

Your work style will be set to phantom. The Phantom style will be defined as a repeating pattern of one long dash and three short dashes. The long dash will be three plot unit in length and the short dashes will be 3/10 plot units long. All dashes will be separated by gaps of 2/10 plot units.

SUBstitute or SB**2D**

The Substitute command replaces one text string with another on your drawing. A search string and replace string are entered. For example, you may want to replace every occurrence of the word "chair" with the word "chair 1". The first occurrence of the search string (chair) will be replaced with the replace string (chair 1) in a paragraph where a match was found. You can also specify a specific line in a paragraph for the search and replace. Strings can also be deleted using this command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Old	= string	1	none	%SB.O	= none
New	= string	2	none	%SB.N	= none
Line	= number	3	0	%SB.L	= 0
Select	= select	4	%PF.S	%SB.S	= Pick
Global coordinate	= global	5	%PF.G none	%SB.G	= Ask

Definition of Options**Old =**

This option specifies the string that will be searched for and replaced. If this option is omitted and the New option is used, the entire line specified will be replaced. If no line was entered, the entire paragraph is replaced.

New =

This option specifies the string that will replace the search string. If this option is omitted and the Old option is used, the search string will be deleted. If the old option is not used with this option, an entire line or paragraph will be replaced with this string, depending on whether the Line option was also used.

Line =

Specifies which line of the text paragraph the search and substitution should be made on. If this option is not entered the search and substitution will start with the first line of the paragraph. If the Old option is also not used, the entire paragraph will be replaced with the New string.

Select =

See **Common Options** for more information.

S Commands

Global =

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Sub Old=`Part A` New=`Part G`

You will be prompted to pick a paragraph of text on your drawing. The first occurrence of the string "Part A" will be replaced with the string "Part G."

sb,,\For future use.\,3,all

Line three of all ungrouped paragraphs of text on your drawing will be replaced with the string "For future use.".

TAg or TG

2D & 3D

The Tag command allows you assign an "invisible" name to groups on your drawing. The only way to select an object for tagging is by picking it with the crosshairs. Depending on the setting of the Global option, the tag can be applied to all, some, or one of multiple copied objects by picking one of them. Tagged groups can be counted using the List and Count commands.

Do not confuse the tag attached to a group on the drawing with the name of a symbol stored on disk. Although it is common to make the group's tag and its save name on disk the same, they are not the same thing. Storing a symbol on disk does not "tag" it.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Tag	= name	1	none	%TG.T	= ``
Global	= switch	2	last	%TG.G	= No
coordinates		none	none		

Definition of Options

Tag =

Sets the tag of the groups you pick. The tag can be any alphanumeric string up to ten characters long. Tags that contain spaces or special characters must be quoted (‘like this’ or \like this\) so they are read as one string and not multiple options of the Tag command.

Global =

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Tag Switch

You will be prompted to pick the objects you would like tagged with the name "Switch." The Global option will default to its last used value.

tg t=\ g=0

You will be prompted to pick the objects you would like tagged. Each object picked will have its existing tag removed because the tag entered was a null string (no characters, empty, nothing, zip). Only the objects picked will be affected, even if there are copies.

The Text command creates paragraphs of graphic text on your drawing. This command can also create text fields and text attributes for a group. Text fields are like text placeholders - their location is predefined, but their verbiage is filled in later. Text attributes are used to attach textual information to a group that you would like to show up in a bill of materials type listing.

This command gives you full control over text height, slant, vertical spacing, horizontal spacing, character width, boldness, font, justification, angle and more.

Each undo (<%> key) will remove the last point located or line of text entered. Each text primitive is limited to 512 characters. A new text primitive is automatically started at a line break when this limit is exceeded.

Note: TurboCAD fonts differ from Macintosh fonts in that they are made of vectors (straight line segments) instead of pixels (dots on the screen). Although fancy vector fonts are a little more difficult to define, they do have several advantages over pixel fonts -mainly that they can be rotated, scaled, mirrored, etcetera, just like any other object on your drawing.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Justification	= justification	1	last	%TX.J	= LC
Height	= number	2	last	%TX.H	= .125
Angle	= angle	3	0	%TX.A	= 0
	or Points (or Pts)				
Slant	= -45 thru 45	4	last	%TX.S	= 0.0
Offset	= number	5	last	%TX.O	= .0625
Width	= number	6	last	%TX.W	= .75
Vertical	= number	7	last	%TX.V	= 1.5
HOrizontal	= number	8	last	%TX.HO	= 1.0
Font	= font	9	last	%TX.F	= 'default'
ATtribute	= 1 thru 127	10	0	%TX.AT	= 1
Placement	= Points (or Pts)	11	last	%TX.P	= Points
	or Drag				

					T Commands
Type	= Paragraph or Single	12	Paragraph	%TX.T	= Paragraph
Field	= switch	13	Off	%TX.FI	= Off
FiLename	= name	14	none	%TX.FIL	= none
STatement	= string	15	none	%TX.ST	= none
Class	= 0 thru 255	16	%WK.C	%WK.C	= 0
Layer	= 0 thru 255	17	%WK.L	%WK.L	= 0
PEn	= 0 thru 255	18	%WK.P	%WK.P	= 0
PRiority	= 0 thru 255	19	%WK.PR	%WK.PR	= 125
WORkset	= workset	20	%WK.W	%WK.W	= none
Mimic			none	%TX.M	
Reset			none	%TX.R	
Dialog			none	%TX.D	
coordinates			none	none	

The Startup values shown here are in inches. If you create a new drawing and select a metric work unit, text eight will be set to 3mm and offset will be set to 1.5mm. Most of the other options that require a numeric value are ratios of the height, and are therefore unaffected by the type of work unit.

Definition of Options

Justification =

This option determines where the text paragraph will appear relative to the point or points you locate. The justification value is a two character code, where one character defines the horizontal relationship of the text to the point, while the other character defines the vertical relationship. Horizontal justifications are Left, Centered and Right, and vertical justifications are Above, Centered and Below. The two characters can be entered in either order, either vertical then horizontal or vice versa. It is also possible to enter only a single character for the justification code. In this case the missing justification is assumed to be Centered. Valid Justification values are:

<u>Code</u>	<u>Meaning</u>
LA or AL	Flush Left, Above the Point
LC or CL or L	Flush Left, Centered on the Point
LB or BL	Flush Left, Below the Point
CA or AC or A	Centered Above the Point
CC or C	Centered on the Point

T Commands

CB	or BC	or B	Centered Below the Point
RA	or AR		Flush Right, Above the Point
RC	or CR	or R	Flush Right, Centered on the Point
RB	or BR		Flush Right, Below the Point

Depending on the setting of the Angle option, you will be prompted to locate either one or two points to locate the text. If one point is located, the text is justified to that one point (centered on the point, flush left to the point, etc.). If two points are located, the text is justified to both points (centered between them, flush left to first point, etc.).

The following example should help clarify the nine different possible justifications and how they relate to the point or points located.

In this illustration, the diamond represents a single point justification and the plus marks represent a two point justification.

Note that Left means flush left (ragged right). Flush left text appears on the right side of a single point located. The same statement goes for flush Right justified text. This gets some people confused. Probably because they are thinking the justification AL means "above and to the left of the point". It does not. It means "above the point, flush left".

RA CA LA
+ ♦ + ♦ + ♦ +

RC CC LC
+ ♦ + ♦ + ♦ +

RB CB LB
+ ♦ + ♦ + ♦ +

Height =

This option sets the height of the text in plot units. Plot units are units measured on the plotted paper. The text height is not expressed in work units and is not affected by the drawing scale. The Height value must be a number greater than zero and less than half the height of your drawing.

Angle =

This option sets the angle of the text paragraph on your drawing. The angle value is any number between -360 and 360. See Common Options for more information on entering angle values.

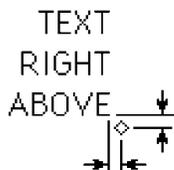
Points (or **Pts**) means that you will be required to locate two points to define the angle of the text paragraph. These two points will also be used as the justification points.

Slant =



This option sets the slant of the text characters. The Slant value must be a whole number from -45 through 45 degrees. Negative values are a back slant and positive values are a forward slant.

Offset =



This option sets the distance that the text will be offset from the point or points located. The Offset value can be any positive number or zero. The offset is both a vertical and horizontal distance.

Width =

GFEDCBA

This option sets the width of text characters. The Width value is a ratio of the text height and can be any non-zero number. Entering a negative number will create horizontally mirrored characters like in the example shown.

Vertical =

This option sets the vertical spacing between lines of text. The Vertical value is a ratio of the text height and can be any non-zero number. A positive number places lines one below another, while a negative number places lines one above another. The example shown here has a vertical value of -1.5.

HOrizontal =

This option sets the horizontal distance between text characters. The Horizontal value is a ratio of the text height and can be any non-zero number. Entering a negative number will create text whose characters are placed right to left on the line. Using this option in conjunction with the Width option can create some unique variations of text. The first example shown here has a positive Width value and a negative Horizontal value, and the second example has both negative Width and Horizontal values.

Font =

This option selects the font (typeface style) you wish to use. The Font value can be the name of an existing font or a number from 0 through 7. Up to eight fonts can be attached to each drawing. Once attached to a drawing, a font is assigned to one of 8 numbered slots. Fonts can then be accessed by their slot number or their name. If a font is specified for use with the Text command that is not already attached to the drawing, it will be automatically attached and placed in the next available slot.

See the Font command for information on attaching, substituting and looking up fonts.

T Commands

ATtribute =

This option specifies that text is to be considered an attribute of a group. The purpose of an attribute is to give the ability to attach descriptive information to a group that will appear on a bill of materials type printout. An example of attribute information might be a part number, item cost, department number, etc. The Attribute value can be a number from 1 through 127. Attribute 0 is the same as regular text.

Here is the way attributes are attached to groups and later referenced. Create text attributes just as you would any other piece of text on your drawing, except that the Attribute option is set to some number. An attribute can be a single line of text only, not a paragraph. Now make the attributes part of a group by using the Group command. The attributes will now appear in printouts created using the List command if you have set the Type option of that command to Attribute. The attributes will appear in the printout in the order they were made a part of the group, not in order of attribute number. See the List command for an example of an attribute printout.

It is a good idea to put attributes on a unique layer so that you can not plot or not display them easily. You might even put different attributes on different layer in the event you would like to selectively plot or view attribute information.

Placement =

This option determines how text will be placed on the drawing. Placement values are Points (or Pts) and Drag.

Points means you will first locate a point or points on your drawing and then type the text. The text will appear relative to the points based on the value entered for the Justification option.

Drag means you will first type the text and then drag it to the desired location. This method is a little slower, but can be very helpful in placing text in a congested area.

Type =

This option sets the type of text paragraph that will be created. Type values are Single and Paragraph.

Paragraph means multiple lines of text will be entered for each point or set of points located. Two consecutive <Returns> are needed to terminate a text paragraph. Each paragraph is a primitive.

Single means only a single line of text can be created for each point or set of points located. This is useful when you are entering a bunch of one line callouts at different locations on the drawing because it saves the double <Return> needed to terminate a paragraph. Each line of text is a primitive.

Field =

This option specifies that text is to be considered a text field. The field value must be a switch value. Switch values are: True or False, Yes or No, On or Off, All or None.

A text field is like a text placeholder. It establishes the location and all the parameters of text except its verbiage. When a field is filled it is converted into either regular text or an attribute.

Fields can be used two ways; as text on a drawing or as text that is part of a group (or symbol). You create a field the same way as you would create regular text or an attribute, except that you have entered the Field option. Where you would normally type in the text wording, type in a question that you would like presented to the operator when it comes time to fill in the field with some textual information. The question will appear on your drawing just like regular text, except that it will not plot.

To "fill in" a field on your drawing, use the Field command. The Field command allows you to pick the text fields on your drawing that you wish to fill. When a field is picked, its question is displayed to the operator on the Prompt line at the bottom of the screen and the field blinks on the screen so it is obvious which field was picked.

Although fields can be created and filled anywhere on your drawing as described in the last two paragraphs, it is much more common and useful to build fields into groups and symbols. A simple example of a field in a symbol might be a callout or column bubble, where the graphics of the symbol never changes but the textual information does. When a group contains fields, picking the group while in the Field command will run through all of the fields of the group prompting the operator to fill each field. The prompts will appear in the order the fields were grouped. See the Field command for more information.

Fields that are part of symbols are even handier ("symbol" is what we call a group that is stored on disk). When the symbol is retrieved from disk with the Get command, you have the option of filling the fields as the symbol is placed on your drawing. See the Get command for more information.

As with attributes, you should consider a Layer scheme to control the display and plot of fields. Keep in mind what layer you want the information to be on after you fill a field. That is the layer the field should be created on.

Filename =

This option specifies the name of an ASCII file to be used as input to the Text command. If the complete path to the file is entered, only the specified folder (directory) will be searched for the file. If only the file name is entered, and not the path, then the folders (directories) specified as the Current directory and the one specified for Macros in the Path command will be searched. If the file name contains spaces or special characters it must be quoted ('like this' or \like this\). If found, the contents of the file will be placed on your drawing as a text paragraph. Variable and tab substitution will be performed on the file before it is placed on the drawing. The Type option will be ignored. This option cannot be used with the Dialog option.

SStatement =

This option specifies a character string to be used as input to the Text command. The character string will be placed on your drawing as a text paragraph. This is most often used inside a macro when you want the macro to put a single line of text on your drawing. Variable and tab substitution will be performed on the string before it is placed on the drawing. The Type option will be ignored. This option cannot be used with the Dialog option.

Class =

See **Common Options** for more information.

T Commands

Layer =

See **Common Options** for more information.

PEn =

See **Common Options** for more information.

PRiority =

See **Common Options** for more information.

WOrkset =

See **Common Options** for more information.

Mimic

See **Common Options** for more information.

Reset

This option sets the values of the Width, Vertical and Horizontal options back to their Startup values. (Just in case you get them all fouled up.)

Dialog

This option is used on Macintosh computers to bring up a dialog in which to type the text. Systems other than Macintoshes with a Command line only do not use this option for anything.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Text

All options will default to their last used values, except the Angle, Attribute, and Field. The Angle will be 0, the Attribute number will be 0 (i.e. not an attribute), and the Field switch will be set to Off. The Mimic, Reset and Dialog options will not be used. The Type option will be set to Paragraph. The Class, Layer, Pen and Priority will default to the values displayed on the Status palette.

You will be prompted to locate one or two points for the location/justification of the text, depending on the last used value for Justification, then you will enter the text wording on the Command line until you enter two consecutive returns.

TX C .2 0 AT=50 L=50 RES

Justification will be set to CC, text height to 1/5 of a plot unit and the text angle to zero. The text entered will be attribute number 50 and will be created on layer 50. The Width, Horizontal and Vertical options will be reset to their startup values. All other options will default normally.

t m a=p t=s v=2 p=d dialog

Because the Mimic option was used, you will be prompted to pick an existing piece of text on your drawing. The options of the text command will assume the same values. The Angle

option is set to Points, requiring that two points be located for each line of text. Type is set to Single and Vertical spacing between lines is twice the current text height. On a Macintosh, you will see a standard text entry dialog into which you will type the wording. Placement has been set to Drag, meaning that the text will be typed first, then dragged into position after you exit the text entry dialog.

Timer or TI

2D & 3D

The Timer command tracks the time that has been spent on a drawing.

The timer is automatically started each time a drawing is loaded. The elapsed time since loading the drawing is remembered in the Current Status Word "%TI.Current." When the drawing is cleared, the current time is added to the previous time and stored in "%TI.Previous" and is saved with the drawing. The timer can be stopped and started while working on a drawing. Furthermore, the Current, Previous and Total times for a specific drawing can be viewed at any time while working on the drawing. Only the Current and Previous times can be modified.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Function	= List or Pause	1	List	%TI.F	= List
Current	= hh:mm:ss	2	%TI.C	%TI.C	= 00:00:00
Previous	= hh:mm:ss	3	%TI.P	%TI.P	= 00:00:00
Total			%TI.T	%TI.T	= 00:00:00

Definition of Options

Function =

Since the timer is always "on," this option only serves to view or temporarily suspend the timer. Valid values are List and Pause.

List causes the values for Current, Previous and Total times to be displayed on your screen. Any key action will cause this display to disappear.

Pause temporarily stops the timer. A message will be displayed informing you that the timer is on hold. You will not be able to execute TurboCAD commands while the timer is stopped. Press any key to start the timer and resume work on your drawing. If you are trying to keep an accurate record of the time spent on a drawing, this option lets you turn the timer off during breaks or interruptions.

T Commands

Current =

This option allows you to alter or change the Current elapsed time for the drawing you are currently working on. Valid values must be in "HH:MM:SS" (hours:minutes:seconds) format and all numbers must be positive.

Previous =

This option allows you to alter or change the Previous elapsed time for the drawing you are currently working on. Valid values must be in "HH:MM:SS" (hours:minutes:seconds) format and all numbers must be positive.

Total

This is not a valid option and is only listed to illustrate the fact that Total time is always the addition of Current and Previous times. You cannot alter or change Total time.

Examples

ti l

The values for Current, Previous, and Total times will be displayed until the user hits any key.

ti P=02:30:00

The value for Previous time for the current drawing will be changed to two hours and thirty minutes. The Current time will be unaffected and the Total time will be recalculated.

TRansform or TF

2D

The Transform command rotates, scales and mirrors objects on your drawing. Objects can be selected for transformation by picking them or by defining a box. The point about which the objects are transformed can also be controlled. This command does, in one command, everything that the Scale and Rotate commands do.

Members of groups cannot be individually transformed, only whole objects. Objects will always be scaled and mirrored before they are rotated, regardless of the order the options were entered.

The <^> or <Esc> key (<clear> on the Macintosh Plus) will attach the last transformed object to the crosshairs and allow you to drag it to a new location.

Each undo (<%> key) will return the last transformed object to its original orientation and scale. The Undo buffer will remember the last 128 objects transformed with this use of the command.

Note: Transformations are always done about the object's X and Y axes, not the drawing axes. When an object is rotated its axes are also rotated. Regrouping an object will realign its axes with the drawing axes because it is then considered a new object that has not yet been rotated.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Angle	= angle	1	0.0	%TF.A	=90
Scale	= number	2	1.0	%TF.S	=1.0
Mirror	= None or X or Y or Both	3	None	%TF.M	=None
SElect	= select	4	%PF.S	%TF.SE	=Pick
Pivot	= Point (or Pt) or Plck or Object	5	Point	%TF.P	=Point
M0de	= Relative or Absolute	6	Relative	%TF.MO	=Relative
Object	= object	7	%PF.O	%TF.O	=Any
Xscale	=number		1.0	%TF.X	=1.0
Yscale	=number		1.0	%TF.Y	=1.0
coordinates				none	none

Definition of Options

Angle =

This option sets the rotation angle to be applied to the object(s) selected. If no value is entered for this option and for the "Scale=" option below, you will be able to dynamically rotate and scale the object(s) on your screen about the pivot point you choose in the Pivot=" option below. A valid Angle value is any number between -360 and 360 degrees, or -2^1 and 2^1 radians. See **Common Options** for more information on entering angles.

Scale =

This option sets the scale that selected objects will be scaled along the object's X and Y axes. A valid Scale value is any number, except zero. Values greater than 1 mean enlargement, and values less than 1 mean making smaller. Negative values mean that the object will be mirrored about the pivot point (defined below) as well as scaled by the absolute value of the value entered. This option will override and reset the Xscale and Yscale options.

T Commands

If no value is entered for this option, for the Angle option, and for the Xscale and Yscale options, you will be able to dynamically rotate and scale the object(s) on your screen. The pivot point for both rotation and scaling will be that which you choose for the pivot point in the "Pivot=" option below.

Mirror =

Sets the axis that objects will be flipped about.

None means no mirroring will occur.

Xaxis means objects will be mirrored about the object's X axis.

Yaxis means objects will be mirrored about the object's Y axis.

Both means objects will be mirrored about both axes.

SElect =

This option determines how objects are to be selected. Valid Select values are Pick, Inside, Outside, Touching, Notinside, NOTOutside, NOTTouching and All. See the **Common Options** section in Basics for more information.

Pivot =

Sets the point that object will be transformed about. Valid Pivot values are:

Point (or **Pt**) means you will be prompted to locate a common point about which all objects will be transformed.

Pick means the point picked on each object to select it is the point that will be used as the pivot point. If objects are selected by box, the center of the box will be used as the pivot point.

Object means the origin of each selected object will be used as the pivot point.

MOde =

This option defines the base angle and scale from which the objects are to be transformed

Relative means the objects will be transformed from their current angle and scale.

Overall means the objects will be transformed from their originally created angles and scale.

Object =

See the **Common Options** section in Basics for more information.

Xscale =

This option sets the value that objects will be scaled along the object's X axis. A valid Xscale value is any number other than zero.

Yscale =

This option sets the value that objects will be scaled along the object's Y axis. A valid Yscale value is any number other than zero.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

TF

After locating the pivot point, each object you pick will be dynamically rotated about that point and scaled about the point you pick it with.

Transform Angle=45 Scale=.5

After locating the pivot point, each object you pick will be rotated a positive 45 deg. about that point and scaled to one half of its current scale about the point you pick it with.

TRANS,M=Y,SEL=IN,P=OB

All objects completely inside a box that you define by locating its diagonally opposite corners will be mirrored about their Y axis. Each object will be mirrored about its individual origin.

tf,90,,x,t,p

After locating the pivot point and defining a box, all objects touching the box will first be mirrored about the X axis and then rotated ninety degrees about the pivot point.

TRansform Three or TFT

3D

The Transform Three command rotates and scales 3-dimensional objects on your drawing. It does, in one command, what the Rotate Three and Scale Three commands do.

Each undo (<%> key) will remove the last object or box of objects transformed. The Undo buffer will remember the last 128 objects with this use of the command.

This command can also be entered as "TRANSFORM_Three" or "TF Three".

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Xangle	= angle	1	0	%TFT.X	=0
Yangle	= angle	2	0	%TFT.Y	=0
Zangle	= angle	3	0	%TFT.Z	=0
Scale	= number	4	1	%TFT.S	=1
Pivot	= Point (or Pt) or Pick	5	Point	%TFT.P	=Point
SYstem	= Viewplane or World	6	Viewplane	%TFT.SY	=Viewplane
Order	= XYZ	7	last	%TFT.O	=XYZ

T Commands

	or XZY				
	or YXZ				
	or YZX				
	or ZXY				
	or ZYX				
Global	= global	8	%PF.G	%TFT.G	=Ask
coordinates			none		

Definition of Options

Xangle =

Sets the angle about the X axis that the selected object will be transformed. The X axis that you are rotating about can be in the Viewplane or World coordinate system as set by the System option. The Xangle value can be any number between -360 and 360. See the **Common Options** section for more information on angle values.

Yangle =

This option is the same as the Xangle option, except about the Y axis.

Zangle =

This option is the same as the Xangle option, except about the Z axis.

Scale =

This option sets the scale that picked object will be scaled about the X, Y and Z axes. A valid Scale value is any number, except zero. The same scale value will be used for all axes.

Pivot =

Sets the point that object will be transformed about. Valid Pivot values are:

Point (or **Pt**) means you will be prompted to locate a common point about which all objects will be transformed.

Pick means the point picked on each object to select it is the point that will be used as the pivot point. If objects are selected by box, the center of the box will be used as the pivot point.

System =

Determines which of the two coordinate systems will be used as the basis for the rotation. Valid System values are Viewplane and World. See **Common Options** for more information.

Order =

This option determines the order that object will be rotated about the X, Y and Z axis. The order that an object is rotated can sometimes have an effect on the outcome. Enter the letters X, Y and Z in the order that you would like the rotation for the corresponding axes to be performed.

Global =

See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples**Transform Three Z=45 Sc=1.5 P=PI**

You will be prompted to pick the objects you wish to rotate. Each object will be rotated a positive forty-five degrees about the Z axis of the Viewplane coordinate system and scaled to one and one half its current size. It will be transformed about the point at which it was picked.

TFT, 90,,90 sy=w g=a

You will first be prompted to locate a pivot point and then to pick the objects you wish to transform. Each object will be rotated a positive ninety degrees about the X and Z axes of the World coordinate system. All picked objects will pivot about the same pivot point. If there are copies of a rotated 3-dimensional group, they will also be transformed.

Ungroup or UG

2D & 3D

The Ungroup command will break apart (unglue, smash, explode) previously created groups. If the group is nested, any number of tiers may be ungrouped starting with the highest tier and working towards the lowest. As groups tiers are ungrouped they are also untagged. See the Group command for information on how group tiers and their tags are defined.

After entering the command and the optional number of tiers, you will be prompted to pick the groups to be broken.

Each undo (<%) key) will deselect the last group picked. Entering the <E> key will ungroup the groups selected thus far. The <Return> key can also be used to initiate the ungrouping. However, this will also terminate the command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Tiers	= 1 thru 255 or All	1	1	%UG.T	=1
coordinates			none		

Definition of Options

Tiers =

This option determines how many tiers will be broken. The Tiers value must be an integer from 1 through 255 or the word "All." Tiers are broken starting from the highest tier and working towards the lowest tier.

All means all group tiers will be dissolved to primitives.

For more information on grouping and group tiers, see the Group command.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Ungroup

You will be prompted to pick the groups you wish to have broken. Only the top tier of each group will be ungrouped.

u 2

You will be prompted to pick the groups you wish to have broken. The top two tiers of each group will be ungrouped.

View or VW

3D

The View command allows you to define the viewplane or display predefined views of 3-Dimensional space. A specific viewplane can also be defined by locating three points (i.e. 3 points define a plane). All views are based on the world coordinate system.

Although the same views can be accomplished using the Rotate World command, this command provides an easy way to get to a known view without having to know its X, Y and Z rotational values. Once at one of these known views, you can then rotate the world around by using the Rotate World command.

Options

Keyword	Value	Order	Default	CSW	Startup
Function = P	Ointsor PTs	1	Points	%VW.F	=Plan
	or PAn				or Top
	or ReVerse				or Bottom
	or North				or Rear
	or South				or Front
	or East				or Right
	or West				or Left
	or NORTHEast				or NE or RIGHTRear or RR
	or NORTHWest				or NW or LEFTRear or LR
	or SOUTHEast				or SE or RIGHTFront or RF
	or SOUTHWest				or SW or LEFTFront or LF
Isometric		2	none	%VW.I	
coordinates			none		

Definition of Options

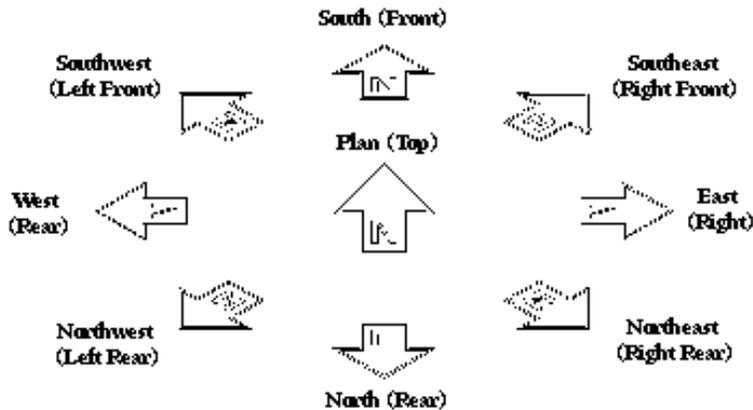
Function =

This option defines the function that is to be performed. The function is either to define a new viewplane or to display one of the pre-defined world coordinate system views.

U-Z Commands

The **Points** (or **Pts**) value means a new viewplane will be defined by locating three points. The three points correspond to the lower right, lower left and upper left corners of your screen, in that order.

The other values of the Function option are pre-defined views. **North** means you are looking at the north elevation (standing on the north side looking south). The other compass headings work the same way. **Plan** means you are looking down on the world and **Reverse** means you are looking up. This example demonstrates the Isometric views of the symbol shown in Plan in the center.



Isometric

An isometric view is one that depicts an object in a rotated and inclined position to give it a pictorial appearance. The rotation and inclination are such as to result in a view with three equally divided axes about a center point. TurboCAD accomplishes this by inclining the view selected with the Function option by approximately $35^{\circ}16'$.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

View Plan

The Plan, or Top view, of the 3-dimensional world will be made parallel to the viewplane (your screen). This means you are on the top of your object looking down, and that your World and Viewplane coordinate systems will be in the same orientation.

View South

The South elevation of the 3-dimensional world will be made parallel to the viewplane (your screen). This means you are on the south side of your object looking north.

v nw i

The world will be rotated so that you are looking at the northwest corner of it in an isometric view.

vw pts

You will be prompted to locate three points. The 3-dimensional world will be rotated so that the plane defined by the three points is parallel to the viewplane (your screen). The points must be entered in the order of: lower left, lower right, upper right.

The Wall command draws parallel line segments. The segments can be any specified distance apart and offset from the points you locate by any amount. There are options for a centerline and automatic closure of the line ends. The Wall command has the ability to automatically gap any object (except text) that the wall terminates on.

The user should be cognizant of the differences, and similarities between, walls and doublelines, which are parallel line primitives created with the Doubleline command. Wall is different from doubleline in that it is just an automated way to draw a line primitive such that there are two parallel segments. You can imagine a wall as a multi-segment line primitive which makes a 180 degree turn at one end and doubles back on itself with segments parallel to the first ones at a specified distance. The Doubleline command creates a unique type of TurboCAD graphic primitive called a doubleline. A doubleline primitive is remembered by its imaginary centerline, but displayed as two parallel lines. Because of this, edits to a doubleline primitive are limited to editing both sides of a doubleline as one unit. The Wall command creates a multi-segment line primitive with segments at both ends which are either invisible (gap segments) or visible, and which can be edited the same as any other line. Using the Lookup command on a wall will tell you it is just a line primitive.

Pressing the <Esc> key immediately after locating the first point of a wall tells it that you want the existing object at that point gapped (if any). The gapping will happen when the line is terminated. Pressing the <Esc> any time after the second point of a wall has been located will terminate the wall at the last point located and gap the object that is at the end of the wall (if any).

Pressing the key performs a similar function to the <Esc> key, except that it locates the point and performs the <Esc> key function all at once. This is merely a shortcut to clicking the mouse key and then the <Esc> key. <Shift>-click on Macintosh computers performs the same function as the key.

Each undo (<%> key) will remove the last point located. The Undo buffer will remember the last 84 points located with this use of the command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Width	= number	1	last	%WA.W	=.5"
Offset	= number	2	0	%WA.O	=0
Centerline	= style or None	3	None	%WA.C	=None
CAp	= switch	4	Yes	%WA.CA	=Yes
Gap	= switch	5	Yes	%WA.G	=Yes
CLass	= 0 thru 255	6	%WK.C	%WK.C	=0
Layer	= 0 thru 255	7	%WK.L	%WK.L	=0
Pen	= 0 thru 255	8	%WK.P	%WK.P	=0
STyle	= style	9	%WK.S	%WK.S	=Solid
PRiority	= 0 thru 255	10	%WK.PR	%WK.PR	=125
WOrkset	= workset	11	%WK.W	%WK.W	= none
Mimic coordinates			none none	%WA.M	

Definition of Options

Width =

Sets the distance between the parallel line segments making up the wall. The Width value can be any number including zero.

Offset =

Sets the distance from the centerline (or imaginary line formed by the points located) to the left side of the wall. The offset value can be positive, negative or zero.

Centerline=

This option will draw a separate line primitive between the points located at the same time the wall is being drawn. The centerline is drawn between the points located. If an Offset value is used while drawing a wall, then the centerline will not necessarily be in the center of the wall, but will always be where the points are located.

CAp =

This option determines if wall ends are to be closed if they are not an end that is gapping an object. The Cap value must be a switch value. Switch values are: True or False, Yes or No, On or Off, All or None. A positive answer means the ends will be capped, while a negative answer means the ends will be left open. If the ends are open, this merely means that those segments will be gap (invisible) segments.

Gap =

This option determines if objects are to be gapped when selected to do so with the 'esc' or 'B' key. The Gap value must be a switch value. Switch values are: True or False, Yes or No, On or Off, All or None. A positive answer means the object will be gapped and the length of each side of the wall will be adjusted as necessary. A negative answer means the sides will be adjusted but the object will not be gapped.

Class =

See **Common Options** for more information.

Layer =

See **Common Options** for more information.

Pen =

See **Common Options** for more information.

Style =

See **Common Options** for more information.

PRiority =

See **Common Options** for more information.

WOrkset =

See **Common Options** for more information.

Mimic

The Mimic option for this command deviates somewhat from the traditional definition of mimic. Since the Wall command creates line primitives, there would be little for the mimic option to "mimic." When you use this option in this command, TurboCAD expects you to pick an existing Doubleline primitive to mimic. The Width, Class, Layer, Pen, Style, and Priority will be mimicked. The Offset will be set to 0, Centerline will not be mimicked, and both ends will be capped if the doubleline picked is capped either at the start or both ends of the line. See **Common Options** for more information.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples**Wall Width=1/8**

Walls will be drawn with a distance of one-eighth unit between the parallel lines. The Offset and Centerline options will default to None. The Class, Layer, Pen, and Style options will default to the values shown on the Status palette. Gap will default to Yes, meaning that use of the <Esc> or key will cause objects to be gapped. Cap will default to Yes, meaning that wall ends that are not gapping an object should be closed.

WA %tx.he o=%tx.he/2 g=n ca=f

Walls will be drawn with a width equal to the current text height. The Offset will be set to half that value meaning the doubleline will be drawn along its left face. Cap, Gap and Centerline options will be set to None. The Class, Layer, Pen, and Style will be Status palette values.

U-Z Commands

Wa,,,Cl,,,5,5

Walls will be drawn at the last used width with no offset. A centerline will be drawn in the Centerline line style. Cap and Gap will default to On. The Class and Layer will be set to five. The Pen, and Style options will default to the values shown on the Status palette.

WEight or WT

2D & 3D

The Weight command gives line weight (thickness) to any TurboCAD object (even individually within a group) by assigning a display and plot "width" to pens on your drawing. The number of retraces desired on the plotter to fill a given width can be automatically calculated or manually entered. Pens on your drawing can be assigned to pen holders on the plotter. The display of line weight on your screen is optional. Remember that TurboCAD also assigns a screen color to each pen, which is independent of the width.

Weights are remembered with a drawing. When you open a drawing, it will have the same line weight assignments as when it was last saved. In order to save time, TurboCAD automatically turns the display of weights OFF when loading a drawing. You can activate the display of weights any time after that via the "Display=" option of this command.

Weights are assigned based on the width that they should be when plotted, not based on a number of pixels on the screen. This gives much greater flexibility in defining your own line weights. To display weights on your screen, we calculate the number of pixels that it takes to approximate the specified width. As you zoom into your drawing, lines with weight will get bolder. This is very helpful in visualizing how different weights relate to one another. Realize though, that the plotter is capable of much finer differentiation in weights than the screen can display.

Weights can be altered by the Plot program when plotting a drawing on a pen plotter.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Pen	= 0 thru 15	1	%WK.P	%WT.P	=0
Width	= number	2	last	%WT.W.n	=.01
Traces	= 1 thru 127	3	last	%WT.T.n	=1
Holder	= 0 thru 15	4	last	%WT.H.n	=0
Display	= switch	5	last	%WT.D	=Yes

Definition of Options

Pen =

This option selects the pen that line weight is to be assigned to. The Pen value must be an integer from 0 through 255. If this option is not used, the current pen will be used. That's the one displayed on the Status palette at the bottom of your screen.

Width =

This option sets the width of the line. The Width value must be a number that is zero or greater. Width is measured in plot units. Plot units are the units in which you plot the drawing. In other words, actual units measured on the paper when the drawing is plotted. Your working units and the scale of your drawing have no affect on the width.

Traces =

This option specifies how many times the pen in the plotter will retrace over a line to obtain the specified width. If this option is not used, the number of traces will automatically be calculated to fill the width specified. The calculation is based on traces being .01" (1/100 of an inch) apart. Formula: Traces = FIX ((width * 100) + 1.5).

Note: It is possible to specify a large width and a small number of traces to get a double line, triple line or more. Although a pen plotter can handle this quite nicely, the lines will be displayed filled in solid on your screen.

Holder =

This option specifies which holder on the plotter will plot the objects drawn with a given pen. The Holder value must be a number from 0 through 15. If this option is not used, all pens on your drawing will be mapped to the first pen holder on the plotter.

If you do assign different pens on your drawing to different holders on the plotter you may still want to occasionally plot with a single pen. Don't worry, the Plot program has just such an option without having to do any reassignments here.

Note: The pen plotter manufacturers have not yet seen fit to establish a standard for numbering pen holders. Consequently, some pen blocks or pen carousels start with the number 1 while others start with the number 0. Be sure you know how your plotter works before you make all the wrong pen-to-holder assignments.

Display =

This option determines if the assigned weights are to be displayed on your screen. If your drawing is currently not displaying weights, then executing this command with this option set to "on," and then forcing a "redraw" (i.e. <D> graphic mode key) will cause weights to be displayed. If you are displaying the entire drawing, you may not be able to discern the weights. Try zooming in to see them. The Display value must be a switch value. Switch values are: True or False, Yes or No, On or Off, All or None. This option only controls the display of line weight on your screen, the Plot program controls whether weights will be plotted at the plotter.

U-Z Commands

Note: Calculating and displaying weights on large drawings with many different line widths can cause redraw times to be noticeably slower. It may help your productivity to work with line weight display turned off and turn it on only when you wish to verify weights.

Examples

Weight Width=1/16 Pen=12 Holder=3

Assuming the plot units are inches, pen 12 will be assigned a width of .0625" (one sixteenth of an inch). The number of traces will be automatically calculated and set to 7. Display of line weight on the screen will default to on. All objects drawn in pen 12 on your drawing will be plotted with the pen in holder number 3 on the plotter.

WE 2.03 2 2

Pen 2 on your drawing will be plotted with holder 2 on the plotter. Two traces will be plotted 0.03 plot units apart (a double line). Line weight on the screen will be 0.03 plot units as well, but will display as a solid bold line.

wt d=f w=2

The current pen (shown on the Status palette) will be assigned a width of 2 plot units. The number of traces will be automatically calculated. The on screen display of line weight will be turned off ("d=f" is interpreted as "display=false").

WHere or WH

2D & 3D

The Where command looks up the exact location of points on your drawing. The whereabouts of a point can be displayed in work units, plot units, data units, screen units or world coordinate system (3D) units.

After entering the command and the unit type, locate points whose coordinates you wish to know. If you use a "locate" key, such as <V>, <C>, etc., the coordinates will be those of the point snapped to and not the location of the crosshairs. The coordinate values for each point will be displayed on the Status palette and remembered in the following Current Status Words.

<u>CSW</u>	<u>Value Remembered</u>
	%CD.XLast Point - X axis coordinate component
	%CD.YLast Point - Y axis coordinate component
	%CD.ZLast Point - Z axis coordinate component
	%CD.XXNext to Last Point - X axis coordinate component
	%CD.YYNext to Last Point - Y axis coordinate component
	%CD.ZZNext to Last Point - Z axis coordinate component

%CD.AN Angle formed by last two points

%CD.A Entered Last Angle Entered

%CD.DI Distance between last two points

%CD.D Entered Last Distance Entered

Note: The Last Angle Entered and Last Distance Entered above are the last values entered for the Distance and Angle options in the Copy, Copy Member, Move, Move Member, Move Segment, and Stretch commands.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Units	= WORK = Plot = World	1	Work	%WH.U	=Work
coordinates			none	none	

Definition of Options

Units =

This option sets the type of units that you wish to have the coordinate location displayed in. Valid Unit types are shown below.

WORK means that the coordinate location of each point will be displayed in absolute work units from the drawing origin as set by the Origin command. Remember that work units take the drawing scale into account.

Plot means that the coordinate location of each point will be displayed in absolute plot units from the lower left corner of the drawing. Plot units are units on the plotted paper and are not affected by the drawing scale.

World means that the coordinate location of each point will be displayed in absolute world coordinate system (3D) units from the drawing origin as set by the Origin command. Remember that world units take the drawing scale into account.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Where

You will be prompted to locate points. As you do, the coordinates of each point will appear on the Status palette in work units.

U-Z Commands

wh,w

You will be prompted to locate points. As you do, the coordinates of each point will appear on the Status palette in world coordinate system units.

WInDow or WD

2D

The Window command allows you to define the size and location of the rectangular portion of the drawing you wish to view. Windows can also be remembered for later recall with this command.

If the Define or Remember options were used, which require that points be located, each undo (<%) key) will remove the last point located.

Note: TurboCAD provides a short-cut which allows you to define and recall "windows" at any time when the crosshairs are active. You do this by using the "/" and "\" characters followed by the number of the window. The "/" is used for recalling previously remembered windows and the "\" is used for remembering the current window. See *Locating Points* for more information.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Function	= Define or Remember or Current or rView or Previous or Actual or Fullview or REFresh	1	Define	%WD.F	=Define
Number	= 1 thru 9 or All	2	last	%WD.N	=1
coordinates			none		

Definition of Options

Function =

This option selects window function that is to be performed. These are:

Define means a new window will be defined by locating the diagonally opposite corners of the rectangular area you wish to view. If the window is not of the same proportions as the terminal screen, the axis of greatest length will be made to fit. The Number option cannot be used when this value is used.

Remember means you wish to define a window and have it remembered. This function requires that the Number option be used to assign a number that the window will be remembered as. The window is defined the same way as in the Define function. The Remember function only remembers windows, it does not display them. Use the View function to view the window you have just remembered. The <\/> key can also be used to remember windows while in the Graphic Mode.

Current means the current window will be remembered. This function requires that the Number option be used to assign a number that the window will be remembered as. This function works the same as the Remember function except that you do not have to locate any points to define the window to be remembered.

View means you wish to recall a previously remembered window. The Number option is required to specify which window is to be recalled. The <\/> key can also be used to recall remembered windows while in the Graphic mode.

Previous means that the window previous to the current one will be displayed. The Number option is ignored when this value is used.

Actual means that the drawing will be displayed on your screen at the scale at which it will plot. You will be required to locate a point that you want to be the center of the new display window. The Number option is ignored when this value is used.

Fullview means that the entire drawing will be displayed such that it fill the screen. The Number option is ignored when this value is used.

REfresh means that the current window will be repainted. This is used mainly to correct pixel blanking caused by extensive editing. Of course any other repaint function will also correct pixel blanking. The Number option is ignored when this value is used.

Number =

This option specifies which window number is to be remembered or recalled. The Number value can be a number from 1 through 9. If you are remembering windows, this option can also be set to "All," allowing you to define all nine remembered windows in one command. This option is only needed when using the Remember or View values for the Function option.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Window

The Function option will default to Define and you will be prompted to locate two points that are the diagonally opposite corners of the rectangular area you wish to view.

U-Z Commands

WI FUNC=REM NUMB=4

You will be prompted to locate two points to define a window. The window will be remembered as number four. The window will not be displayed, only remembered.

wd v 7

Previously remembered window number seven will be displayed.

wi p

The window that was viewed previous to the current one will be displayed.

Wd,Ref,6

The current window will be refreshed (repainted). The number will be ignored.

Wireframe or WF

3D

The Wireframe command creates wireframe primitives. Wireframes are really nothing more than multi-segment lines in three dimensional space. Just remember that the Line command draws 2D lines and the Wireframe command draws 3D lines.

Wireframes can be created by locating points or entering coordinates (like creating a line), by converting an existing 2-dimensional object into a wireframe (making it a 3-d object). When drawing a wireframe you can take advantage of the locate keys to snap to the existing vertices of 2-dimensional objects or other wireframes, their mid points, etc. If a point is located that is not snapped to some existing object, then its z-depth defaults to the value set by the Z option of this command.

Wireframes can be edited by using the 3-dimensional editing commands.

Each undo (<%> key) will remove the last point located or object picked to create the current wireframe. A wireframe cannot have more than 84 edges.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Z	= number	1	%ZD.D	%WF.Z	=0
Class	= 0 thru 255	2	%WK.C	%WK.C	=0
Layer	= 0 thru 255	3	%WK.L	%WK.L	=0
Pen	= 0 thru 255	4	%WK.P	%WK.P	=0
Style	= style	5	%WK.S	%WK.S	=Solid
PRiority	= 0 thru 255	6	%WK.PR	%WK.PR	=125
WORkset	= workset	7	%WK.W	%WK.W	=none

Convert	none	%WF.C
coordinates	none	

Definition of Options

Z =

This option sets the Z axis depth that you are working at. The Z depth is measured in work units. The Z value can be any positive or negative number. Positive values are towards you (out of the screen) and negative values are away from you (into the screen). Another way to think of this is to visualize that you are drawing in plan (looking down at the ground). A wall that is a positive eight feet tall is coming towards you. Plan happens to be the default view that all new drawings start at.

Any points that are located in free space (not snapped to some existing object) will assume the Z depth as the value for its Z axis coordinate in 3-dimensional space.

Class =

See **Common Options** for more information.

Layer =

See **Common Options** for more information.

Pen =

See **Common Options** for more information.

Style =

See **Common Options** for more information.

PRiority =

See **Common Options** for more information.

WOrkset =

See **Common Options** for more information.

Convert

This option allows you to convert existing objects (except Text and Points) on your drawing into wireframes. You will be prompted to pick an existing object on your drawing instead of locating points to create the wireframe.

Convert

This option allows you to convert existing objects (except Text and Points) on your drawing into wireframes. You will be prompted to pick an existing object on your drawing instead of locating points to create the wireframe.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

U-Z Commands

Examples

WIREFRAME

You will be prompted to locate points to draw a wireframe. The Class, Layer, Pen, and Style values will default to the values shown on the Status palette at the bottom of your screen. The Z depth will default to the value shown on the Coordinates palette.

WF -3'4" Pen=4 ST=da

You will be prompted to locate points to draw a wireframe. The Z depth is set to a negative three feet four inches (away from you). The wireframe will be drawn with pen 4 and a dashed line style. The Class, Layer and Priority values will default to the values shown on the Status palette.

wire,conv,l=200,p=5

You will be prompted to pick objects that you wish to have converted into wireframes. The wireframe will have layer 200 with pen 5. The Class, and Style values will default to those shown on the Status palette.

Work or WK

2D & 3D

The Work command sets the default working Class, Layer, Pen, and Style which are displayed on the TurboCAD Status palette. These can also be set by the Class, Layer, Pen, and Style commands. This command also has a Mimic option which allows you to set the work defaults to the same properties of an existing object on your drawing.

This command also allows you to create and define worksets. A workset is a combination of any specific Class, Layer, Pen, Style and Priority (you can define only some of these if you wish) to which you assign a name. You can later recall that specific combination by its name. Workset definitions are remembered with drawings.

Commands which create primitives and objects also have Class, Layer, Pen, Style and Priority as options as well as a Mimic option. Setting any one of these options from within one of these commands will temporarily override the default for that command only and will not affect the values displayed on the Status palette.

The current work Class, Layer, Pen, and Style are displayed on the Status palette at the bottom of your screen. These are the values that will be defaulted to when executing a command that has Class, Layer, Pen, Style or Priority as an option. Mostly these are commands that create new primitives on your drawing.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Workset	= workset	1	none	%WK.W	=none
Class	= 0 thru 255	2	last	%WK.C	=0
Layer	= 0 thru 255	3	last	%WK.L	=0
Pen	= 0 thru 255	4	last	%WK.P	=0
Style	= style	5	last	%WK.S	=Solid
PRiority	= 0 thru 255	6	last	%WK.PR	=125
Name	= name	7	none	%WK.N	=none
Delete	= name	8	none	%WK.D	=none
Llist			none	%WK.LI	
Mimic			none	%WK.M	
coordinates			none		

Definition of Options

Workset =

This option requires that you enter the name of a previously defined workset. The values for Class, Layer, Pen, Style and Priority which are part of that workset definition will become the default values on the Status palette. Workset definitions are remembered with you save a drawing.

Class =

Sets the Class that objects will be drawn in. The current class will be displayed on the Status palette at the bottom of your screen. The class value must be a number from 0 through 255.

Layer =

Sets the Layer that objects will be drawn on. The current layer will be displayed on the Status palette at the bottom of your screen. The layer value must be a number from 0 through 255.

Pen =

Sets the Pen that objects will be drawn with. The current pen will be displayed on the Status palette at the bottom of your screen. The pen value must be a number from 0 through 255.

Style =

Sets the Line Style that objects will be drawn in. The current style will be displayed on the Status palette at the bottom of your screen. Valid Style values are:

Solid	or SO	or 0
Dotted	or DO	or 1
Centerline	or CL	or 2
DAShed	or DA	or 3
Longdash	or LD	or 4
Phantom	or PH	or 5
LONGCenterline	or LC	or 6
LONGGap	or LG	or 7
PRiority =		

Sets the pick Priority that objects will have when drawn. The current priority will be displayed on the Status palette at the bottom of your screen. The priority value must be a number from 0 through 255.

Name =

This option is used to create a Workset. The name may be up to 32 characters in length, and names are not case sensitive. The values for Class, Layer, Pen, Style, and Priority entered for those options in this command will become a workset definition. Only the ones given a value will be remembered. You may, for example, only want to define worksets which assign names to specific layers. If the name entered is already defined, a "replace" question will be asked. You may have the same workset definition under several names, but a name may only appear once.

Delete =

This option is used to delete a currently defined workset. Enter the name of the workset and it will be removed from the list of currently available worksets.

List

Using this keyword-only option will cause a list of all currently defined worksets and their definitions to be displayed on the screen.

Mimic

This keyword-only option allows you to set the work defaults to the same properties as an existing object on your drawing. You will be prompted to pick an existing object. The Class, Layer, Pen, Style and Priority values will be set to those of the picked object.

coordinates

The only time that coordinates would be required in this command are if you are using the Mimic option. See **Locating Points** and **Entering Coordinates** for more information.

Examples

WK `FIRE WALLS`

The Class, Layer, Pen, Style and Priority will be set to those contained in the workset definition named "FIRE WALLS." If only some of these options are defined in the workset, then only those will be set and the remainder will remain the same on the Status palette.

Work Layer=122 Class=122 Name=`FIRE WALLS`

The working class as well as the working layer will be set to 122. Furthermore, these will be remembered in the workset named "FIRE WALLS" and will be added to the list of current worksets.

WK D=`FIRE WALLS`

The workset "FIRE WALLS" will be deleted from the list of currently available worksets.

WK,,45,76 Style=Dotted

The work class will be set to 45, the work layer to 76, and the style to Dotted. These values will be displayed on the Status palette.

wo m

You will be prompted to pick an existing object on your drawing. The work defaults will be set to the same Class, Layer, Pen, Style and Priority values as the object picked. These values will be displayed on the Status palette.

WORKPlane or WP**3D**

The Workplane command sets the plane that 3-dimensional objects will be drawn on. This command allows you to set the work plane to a plane that is not coincidental with the view plane.

Note: The View and Rotate World commands will automatically set the work plane parallel to the view plane. If you want your work plane to remain unchanged after changing views, use the "Previous" option of this command.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Function	= P0ints (or PTs) 1 =Viewplane or Previous or Viewplane		Points	%WP.F	
coordinates			none		

Definition of Options**Function =**

This option sets how the Workplane will be defined.

U-Z Commands

Points (or **P**Ts) means a new work plane will be defined by locating three points. You can take advantage of the TurboCAD "locate" keys to snap to existing points (2D or 3D) on your drawing.

Previous means the work plane will be set to the plane defined in the last use of the Workplane command. This value is used to set the work plane to its previous location after the 3D view has been changed.

Viewplane means the work plane will be aligned with the current viewplane.

coordinates

See **Locating Points** and **Entering Coordinates** for more information.

Examples

Workplane

You will be prompted to locate three points to define the plane you wish to work on. The view plane will not be changed.

wp f=p

The work plane will be set the same as the last use of the Workplane command.

WRiteline or WL

Macro

It is sometimes necessary, especially when you want to extract information from a drawing, to be able to "write" (or export) that information to a text file external to TurboCAD. The Writeline command does just that, one line at a time. This command will write information to the Command Area or an ASCII file on disk. This command is usually used from within a macro.

The Writeline command always writes information at the end of the file being written to. At your option, the information is either appended to the last line or becomes a new line.

Refer to the Readline and File commands for related topics.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Filename	= name	1	none	%WL.F	= none
Newline	= switch	2	Yes	%WL.N	= Yes
Statement	= string	3	last	%WL.S	= none

Definition of Options

Filename =

Specifies the disk file that will be written to. If the complete path to the file is entered, only the specified folder (directory) will be searched for the file. If only the file name is entered, and not the path, then the folders (directories) specified as the Current directory and the one specified for Macros in the Path command will be searched. If the file name contains spaces or special characters it must be quoted (like this` or \like this\).

A file must be "open" to TurboCAD in order for it to be written to. See the File command for information on opening and closing text files. If not found or already open, this command will cause a new file with the specified filename to be created and opened.

The information contained in the "Statement=" option below will be written to the file at the end of the file. If this option is not used, the command will write to the Command line.

Newline =

This option determines if multiple writes to a file should be placed on the same line or individual lines. The Newline value must be a switch value. Switch values are: True or False, Yes or No, On or OFF, All or None. A positive value means that each "write" to a file will start on a new line. A negative value will write at the end of the previous line.

Statement =

Specifies the string that will be written. The string can be any alpha numeric character string. Keep in mind that you can use this command to write the contents of Variables, Current Status Words, results of a List command, etc. to an external file.

Examples

Write Statement=\Here we go again!

The string "Here we go again!" will appear on the Command Area.

wl,Areas,No,%aa.area %aa.tarea

The values stored in the Current Status Words "%aa.area" and "%aa.tarea" will both be written on the same line to the file "Areas." These CSWs contain the last calculated area and total area, respectively.

Zdepth or ZD

3D

The Zdepth command sets the working depth along the Z axis of the Viewplane coordinate system. The current Zdepth value is displayed on the Status palette at the bottom of your screen.

The Zdepth value is the default Z axis depth that 3-dimensional objects will be drawn at. While in the Wireframe command, the Zdepth can also be changed "mid-stream" by pressing the <Esc> key and entering a new value for Zdepth.

Options

<u>Keyword</u>	<u>Value</u>	<u>Order</u>	<u>Default</u>	<u>CSW</u>	<u>Startup</u>
Depth	= number	1	last	%ZD.D	=0.0

Definition of Options

Depth =

Sets the depth along the Viewplane Z axis that 3-dimensional objects are to be drawn. The Depth value can be any floating point number. A positive value is towards you (out of the screen) and a negative value is away from you (into the screen).

Examples

ZDepth Depth=4'5"

The Viewplane Z axis depth will be set to a positive four feet five inches.

z -.2

The Viewplane Z axis depth will be set to a negative 2/10 of a work unit.

CHAPTER 7 - REFERENCE

Macros

The TurboCAD Macro language is a programming language designed to allow users to make the most effective use of the TurboCAD program. The main use for macros is to automate repetitive sequences of commands to save operator time. This also helps to establish drafting standards for a more consistent drawing appearance. macros let you tailor the system to the needs of your application.

A macro is simply an ASCII text file that contains TurboCAD commands. When executed the file is interpreted line by line. You can write a macro in any word processor that can save files in ASCII format (also called Text Only format). The TurboCAD Modifytext command can also write and edit macros. Some computer systems require that the macro file stored on disk have the extension ".mac" (and may be case sensitive). Macintosh computers do not have filename extensions nor are macro filenames case sensitive.

An easy way to get familiar with the format of macros is to use the Record command. As TurboCAD commands are executed interactively they will also be written into an ASCII file that can later be executed as a macro. You can look at and edit this file to get familiar with how macros work.

Macro Execution

Macros are executed by using the "Macro" command as described in the Commands chapter. The slash character (/) is a built-in alias for the Macro command.

All TurboCAD commands are valid in a Macro, except the Quit and Exit commands. Any valid entry of a command interactively is also valid in a Macro. However, it is advised that you establish a convention for entering commands in a Macro. We suggest you enter command names and options spelled out as fully as possible. This generally leads to fewer errors, and makes the Macro easier to read. This is especially helpful when trying to debug a Macro. Also, as more commands and options are added to TurboCAD in future upgrades, it is less likely that a conflict will arise due to a minimum abbreviation.

Macros can be "nested". That is, any line in a Macro can be another Macro, and that Macro can contain lines which are another Macros, and so forth. The user can control whether or not execution of a Macro continues after the nested Macro is finished. The maximum level of nesting (i.e. a macro within a macro within a macro ...) is controlled by the Macro option of the Preferences command.

Macro Commands

Although all TurboCAD commands can be executed interactively, some of the commands only have a useful purpose when used in a Macro. These commands generally control the execution of the Macro. These Macro commands are fully explained in the Commands

chapter and you should read through all of them. They are recognized by the word "Macro" in the upper left of the command's title sheet. Here is a brief description of the commands that control a macro's execution:

- Bell**
Rings the bell at the terminal to attract the operators attention to something.
- File**
Controls the file table used by the Readline and Writeline commands.
- Goto**
Branches the execution of the macro to another line in the macro. The line being branched to must have a label. This command is often executed by the If command.
- If**
Performs a function only if a statement is true. A conditional statement.
- Message**
Sends informational messages to the operator.
- Pause**
Temporarily halts the execution of the macro for a specified period of time.
- Prompt**
Issues prompts to the operator and remembers the response in a variable.
- Readline**
Reads from a file or from the keyboard.
- Return**
The Return command terminates the execution of the current macro and returns control the next higher macro level.
- Stop**
Terminates the execution of all macro levels and returns control the interactive level.
- Writeline**
Writes to a file or the Command line.

Macro Interpretation

Macros are interpreted one line at a time. The process of interpreting each line has three steps: substitution, syntax checking and execution.

In the substitution part, a line is read from the macro file and then checked for numeric and character expressions. If variables, Current Status Words or Current Arguments are found they are replaced with their values. For example, in the line (the Circle command is used for this example):

Circle Pen = &Oldpen + 1

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the variable &Oldpen (an example of a user created Local Variable) is replaced with its value. Assuming &Oldpen has a value of 4, line is changed to read:

```
Circle Pen = 4 + 1
```

After the substitution has been done, the syntax of the line is checked. In this example, this includes making sure that "Circle" is a valid command, that "Pen" is a valid option of the Circle command and that the equals sign is included. If calculator expressions are encountered during the syntax checking step, they are processed. The expression 4+1 is evaluated and its sum (five) is checked to see that it is within the range of valid pen numbers.

In the execution step, the command is actually performed. Continuing with the example above, the result would be a prompt to the operator to locate the circle's center, then to locate a point on the circle. The circle would be drawn with pen five.

If an error occurs during the execution of a macro, the macro will be halted. The error generated by the invalid command will be displayed on the Prompt line. In addition, a macro error message will be issued in the format:

```
Macro name halted at line number.
```

Where *name* is the name of the macro that was executing and *number* is the line number in the macro where the error was encountered.

Startup Macro

It is possible to have a macro run automatically when the TurboCAD program is run. This macro is most commonly used to set up command options and other parameters. The macro must be named "initpeg" and be in the same directory (folder) as the TurboCAD program.

Line Continuation

Occasionally, a command won't fit on one line in a macro, or it is easier to edit if broken up. The solution is to place the command on multiple lines by "continuing" it. Lines are continued by placing a backslash (\) at the end of the first line, and placing the remainder of the line on the next line in the macro. For example, the lines:

```
Dimension Extension=Point Angle=45 Type=String \  
Level=12 Pen=7 Gap=.125
```

would be interpreted as one long Dimension command. Multiple continuations are also possible. For example:

```
Line \  
Pen=5 \  
[a,4,5] \  
[r,3.90] \  
[:]
```

would execute the Line command, select Pen five, locate two points and exit the command.

Line continuations can also be entered interactively on the command line. The Prompt line will read "Continue" when a line continuation is in progress.

Comments

A "Comment" is a character string that you place in a macro that is for informational purposes—it is not processed as a command. Comments are most commonly used to document the way that a macro works. The use of comments is encouraged. A well documented macro is much easier to modify six months after it was written than one that was poorly (or not at all) documented. Remember, that someday someone else may have to figure out a macro you wrote.

Comments are placed in a macro by enclosing them in curly brackets {like this}. The comment starts at the left bracket and ends at the first right bracket or carriage return found. The macro interpreter ignores comments, so including comments will not cause a macro to execute any more slowly than it would without them.

Information that should always be included in a macro as comments would be the purpose and function of the macro, the input expected and output generated. The date the macro was written or last revised, and a list of variables used are also helpful to have.

Comments can also be entered interactively onto the command line. They will be ignored when the command line is processed, just like they are in macros.

Blank lines may also be placed anywhere within a macro without affecting execution, with one exception. A blank line after a line continuation will be read as a valid command line. Use blank lines freely to make your macros more readable.

Because leading spaces and tabs are ignored on the command line, they can be used to indent lines of your macro. This also makes the macro more organized and readable.

An example of a simple macro using comments, blank lines and line continuation might be:

```

{ This Macro creates circular or square columns
{ The operator must enter the column size
Prompt `Square or Round columns? Answer S or R.` \
  &Ans String
If (&Ans == `s` || &Ans == `S`), Goto Square:
If (&Ans == `r` || &Ans == `R`), Goto Round:
Stop

Round:
Prompt `Enter Round Column Radius` &Rad Num
Circle Radius Siz=&Rad {Draw round columns
Return
Square:
Prompt `Enter Square Column Size` &Siz Num
Rect C=Center W=&Siz H=&Siz {Draw square columns
Return

```

Current Status Words

A Current Status Word (CSW) is a TurboCAD system variable that remembers the value (setting) of a command's option. The CSW value is automatically set when a command is executed.

The main use of CSWs is so TurboCAD knows the value for options that default to their last used value. CSWs are also used to contain other values that are related to the command but are not actually an option of the command. Not all commands have CSWs associated with them, since not all commands "remember" their last used values.

CSWs are named according to the following rule:

% command code . option name

That is, a percent sign, command code, a period, and the option name. For example, the CSW for the current text height would be %TX.Height, and the CSW for the total area last calculated would be %aa.total.

Case is not significant in the name of a CSW—upper, lower, or mixed case may be used. The option name can be abbreviated to its minimum entry as shown capitalized below. For example, %fl.s, %FL.Size, and %fl.SIZ are all equivalent.

CSWs cannot be assigned a value by using the assignment statement. They can only be set by executing a command. This means that they may not appear to the left of an equal sign; you cannot assign a new Fillet Radius with the command "%FL.Rad = 2". They may, however, be used freely in expressions or as option values. In other words, Current Status Words are "read only" variables. For example, you can draw a circle whose radius is equal to the current doubleline width by entering "Circle Radius Size=%db.w".

Some CSWs have indices associated with them. These are given by following the CSW with a period and an index number. Some examples of this are the CSW for the width of pen five, as set by the Weight command: %WT.Width.5; and whether or not class 100 is being displayed: %CL.Display.100. If you don't include an index with a CSW that can use one, the default is the lowest possible index number, but an index included with a CSW that cannot use one is an error. CSWs with indices are shown in the following list by a .[n-m] after the CSW name, where "n" is the lowest index number allowed and "m" is the highest.

There are also CSWs associated with commands which do not have a corresponding option in the command. An example of this would be the CSW for the last area calculated by the Area command, %AA.Area. There is no Area parameter in the Area command, though there is a CSW for it so that, for example, there is a way to look up the last area calculated from within a macro.

Some CSWs are shared between more than one command. For example, the Drawing parameter in Load, New, Open and Setup each refer to the same CSW, though you can refer to it as either %LD.D, %NW.D, %OP.D or %SU.D. These duplicated CSWs are noted in the list below.

The list shows all of the CSWs, the command each is associated with, their startup value (what they are set to before their command is ever used), their data type, and the units that

data is in. The CSWs are listed by command in alphabetical order—not their positional input order. Capitalization is used to show the minimum abbreviation with the remainder of the CSW's name in lower case letters. The codes used to describe the type of data a CSW can contain are:

<u>Code</u>	<u>Data Type</u>
I	A whole number (like "9").
F	A decimal floating point number (like 3.1415).
S	Any alphanumeric character string (like "Overall").
V	A variable name (like A).
X	A switch value (like "On", "Off", "Yes", "No", etc.)
K	No value, a "keyword-only" option.

Some parameters will accept a mix, such as a number or the word "Points," so two data types will be shown because the CSW could have either value. You will need to refer to the specific command in the Commands chapter to find out what the actual values might be for a given CSW. The Units column will show a code for the type of unit that is stored in a CSW. The Units codes are:

<u>Code</u>	<u>Unit Type</u>
W	Work units on your drawing.
P	Plot units on the paper.
S	Inches measured on the physical screen.
A	Angular units (degrees or radians).
-	No units for this CSW.

<u>Command Name</u>	<u>CSW Name</u>	<u>Startup Value</u>	<u>Data</u>		
			<u>Type</u>	<u>Units</u>	<u>Notes</u>
Alias	%AS.Active	Yes	X	-	
Alias	%AS.Function	List	S	-	
Alias	%AS.Name	none	S	-	
Alias	%AS.Statement	none	S	-	
Align	%AL.Angle	Points	F,S	-	
Align	%AL.Object	Any	S	-	
Align	%AL.Select	Pick	S	-	
Angle	%AN.Angle	0.0	F	A	Read only
Angle	%AN.Difference	0.0	F	A	Read only
Angle	%AN.Included	0.0	F	A	Read only

CURRENT STATUS WORDS

Command Name	CSW Name	Startup	Data		
		Value	Type	Units	Notes
Angle	%AN.Previous	0.0	F	A	Read only
Angle	%AN.Silent	False	K	-	
Angle	%AN.Total	0.0	F	A	Read only
Angle	%AN.Type	Points	S	-	
Arc	%AR.COMplement	False	K	-	
Arc	%AR.Construction	Radius	S	-	
Arc	%AR.Mimic	False	K	-	
Arc	%AR.Size	1.0	F	W	
Area	%AA.Area	0.0	F	W	Read only
Area	%AA.Perimeter	0.0	F	W	Read only
Area	%AA.Silent	False	K	-	
Area	%AA.TArea	0.0	F	W	Read only
Area	%AA.TPerimeter	0.0	F	W	Read only
Area	%AA.Type	Points	S	-	
Bitsave	%BT.FileName	none	S	-	
Bitsave	%BT.FORce	False	K	-	
Boundary	%BD.Fill	0	I,S	-	
Boundary	%BD.Mimic	False	K	-	
Boundary	%BD.FRame	False	K	-	
Break	%BK.Convert	False	K	-	
Break	%BK.Type	Vertex	S	-	
Calculator	%CA.Result	none	S	-	Read only
Chamfer	%CF.Angle	Point	F,S	A	
Chamfer	%CF.COnvert	False	K	-	
Chamfer	%CG.Xsize	1.0	F	W	
Chamfer	%CG.Ysize	1.0	F	W	
Change	%CG.Global	Ask	S	-	
Change	%CG.Object	Any	S	-	
Change	%CG.Select	Pick	S	-	
Change_Group	%CGG.Class	0	I	-	
Change_Group	%CGG.Object	Any	S	-	
Change_Group	%CGG.Priority	125	I	-	
Change_Group	%CGG.Select	Pick	S	-	
Change_Member	%CGM.Globd	Ask	S	-	
Change_Member	%CGM.Layer	0	I	-	
Change_Member	%CGM.Object	Any	S	-	
Change_Member	%CGM.Pen	0	I	-	
Change_Member	%CGM.SElect	Pick	S	-	
Change_Member	%CGM.Style	Solid	S,I	-	
Change_Object	%CGO.Global	Ask	S	-	
Change_Object	%CGO.Object	Any	S	-	
Change_Object	%CGO.Type	Member	S	-	
Circle	%CC.Construction	Radius	S	-	
Circle	%CC.Mimic	False	K	-	
Circle	%CC.Size	1.0	F	W	
Class	%CL.Display.[0-255]	All	I,S	-	Same as %DP.C
Class	%CL.Edit.[0-255]	All	I,S	-	
Class	%CL.Used.[0-255]	Used	I,S	-	Read only

CURRENT STATUS WORDS

Command Name	CSW Name	Startup	Data		
		Value	Type	Units	Notes
Class	%CL.Work	0	I	-	Same as %WK.C
Clear	%CZ.Force	False	K	-	
Clip	%CX.Convert	Yes	X	-	
Clip	%CX.Delete	Inside	S	-	
Clip	%CX.Fence	Rectangle	S	-	
Clip	%CX.FRame	False	K	-	
Clip	%CX.Select	Touching	S	-	
Close	%CZ.Force	False	K	-	
Colors	%CS.Color	Blue	S	-	
Colors	%CS.Hue.[0-360]	0	I	-	
Colors	%CS.Item	Field	I,S	-	
Colors	%CS.Lightness.[0-100] 50	I	-	-	
Colors	%CS.Noupdate	False	K	-	
Colors	%CS.Reset	False	K	-	
Colors	%CS.Saturation.[0-100] 100	I	-	-	
Coordinate	%CD.Active	Yes	X	-	
Coordinate	%CD.ANgle	0.0	F	A	Read only
Coordinate	%CD.AEntered	0.0	F	A	Read only
Coordinate	%CD.CLear	False	K	-	
Coordinate	%CD.Cursor	False	K	-	
Coordinate	%CD.Distance	0.0	F	W	Read only
Coordinate	%CD.DEntered	0.0	F	W	Read only
Coordinate	%CD.X	0.0	F	W	Read only
Coordinate	%CD.XDistance	0.0	F	W	Read only
Coordinate	%CD.XEntered	0.0	F	W	Read only
Coordinate	%CD.XX	0.0	F	W	Read only
Coordinate	%CD.Y	0.0	F	W	Read only
Coordinate	%CD.YDistance	0.0	F	W	Read only
Coordinate	%CD.YEntered	0.0	F	W	Read only
Coordinate	%CD.YY	0.0	F	W	Read only
Coordinate	%CD.Z	0.0	F	W	Read only
Coordinate	%CD.ZDistance	0.0	F	W	Read only
Coordinate	%CD.ZEntered	0.0	F	W	Read only
Coordinate	%CD.ZZ	0.0	F	W	Read only
Copy	%CP.Angle	0.0	F	A	
Copy	%CP.Distance	0.0	F	W	
Copy	%CP.Mode	Relative	S	-	
Copy	%CP.Object	Any	S	-	
Copy	%CP.Repeat	1	I	-	
Copy	%CP.Select	Pick	S	-	
Copy	%CP.Vector	False	K	-	
Copy	%CP.Xdistance	0.0	F	W	
Copy	%CP.Ydistance	0.0	F	W	
Copy	%CP.Zdistance	0.0	F	W	
Copy_Member	%CPM.Angle	0.0	F	A	
Copy_Member	%CPM.Distance	0.0	F	W	
Copy_Member	%CPM.Mode	Relative	S	-	
Copy_Member	%CPM.Object	Any	S	-	

CURRENT STATUS WORDS

Command Name	CSW Name	Value	Startup Type	Units	Data Notes
Copy_Member	%CPM.Repeat		1	I	-
Copy_Member	%CPM.Vector		False	K	-
Copy_Member	%CPM.Xdistance		0.0	F	W
Copy_Member	%CPM.Ydistance		0.0	F	W
Copy_Parallel	%CPP.Distance		1.0	F,S	W
Copy_Parallel	%CPP.Mode		Relative	S	-
Copy_Parallel	%CPP.Repeat		1	I	-
Copy_Radial	%CPR.Angle		Points	F,S	A
Copy_Radial	%CPR.Mode		Relative	S	-
Copy_Radial	%CPR.Object		Any	S	-
Copy_Radial	%CPR.Repeat		1	I	-
Copy_Radial	%CPR.ROtate		Yes	X	-
Copy_Radial	%CPR.Select		Pick	S	-
Count	%CT.Classes.[0-255]		All	S,I	-
Count	%CT.Tag		`	S	-
Crosshatch	%CR.Angle		0.0	F	A
Crosshatch	%CR.Frame		False	K	-
Crosshatch	%CR.Mimic		False	K	-
Crosshatch	%CR.Origin		Boundary	S	-
Crosshatch	%CR.Pattern		Lines	S	-
Crosshatch	%CR.SElect		False	K	-
Crosshatch	%CR.Scale		1.0	F	-
Delete	%DL.Object		Any	S	-
Delete	%DL.Select		Pick	S	-
Delete_Edge	%DLE.Global		Ask	S	-
Delete_Gap	%DLG.Global		Ask	S	-
Delete_Gap3	%DL3.Global		Ask	S	-
Delete_Member	%DLM.Global		Ask	S	-
Delete_Node	%DLN.Global		Ask	S	-
Delete_Segment	%DLS.Global		Ask	S	-
Delete_Vertex	%DLV.Global		Ask	S	-
Digitizer	%DG.Baudrate		9600	I	-
Digitizer	%DG.Databits		8	I	-
Digitizer	%DG.Keys		`vcezup%kojntxya`	S	-
Digitizer	%DG.Match		One	S	-
Digitizer	%DG.PArity		None	S	-
Digitizer	%DG.POrt		Modem	S	-
Digitizer	%DG.Protocol		None	S	-
Digitizer	%DG.Scren		None	S	-
Digitizer	%DG.STEpsize		1000	I	-
Digitizer	%DG.STopbits		2	I	-
Dimension	%DM.Angle		Points	F,S	A
Dimension	%DM.Dimension		0.375"	F	P
Dimension	%DM.Extension		Points	F,S	P
Dimension	%DM.EXTStyle		Solid	S,I	-
Dimension	%DM.Format		Fractional	S	-
Dimension	%DM.Gap		0.0625"	F	P
Dimension	%DM.Kind		Linear	S	-

CURRENT STATUS WORDS

<u>Command Name</u>	<u>CSW Name</u>	<u>Startup Value</u>	<u>Data</u>		
			<u>Type</u>	<u>Units</u>	<u>Notes</u>
Dimension	%DM.Leader	Arrow	S	-	
Dimension	%DM.LENgtH	0.125"	F	P	
Dimension	%DM.Mimic	False	K	-	
Dimension	%DM.Overrun	0.0625"	F	P	
Dimension	%DM.RADial	Multiple	S	-	
Dimension	%DM.Rounding	0.0625"	F	W	
Dimension	%DM.SHow	On	X	-	
Dimension	%DM.SMart	On	X	-	
Dimension	%DM.Spacing	0.25"	F	P	
Dimension	%DM.TExt	Centered	S	-	
Dimension	%DM.Type	Incremental	S	-	
Dimension	%DM.Units	Inches	S	-	
Dimension	%DM.Width	0.125"	F	P	
Display	%DP.Classes.[0-255]	All	S,I	-	Same as %CL.D
Display	%DP.Layers.[0-255]	All	S,I	-	Same as %LA.D
Display	%DP.Pen	0	I	-	Same as %PN.D
Display	%DP.Regenerate	False	X	-	
Distance	%DS.Angle	0.0	F	A	Read only
Distance	%DS.Distance	0.0	F	W	Read only
Distance	%DS.Silent	False	K	-	
Distance	%DS.TDistance	0.0	F	W	Read only
Distance	%DS.Type	Points	S	-	
Distance	%DS.Xdistance	0.0	F	W	Read only
Distance	%DS.Ydistance	0.0	F	W	Read only
Doubleline	%DB.CAp	None	S	-	
Doubleline	%DB.Centerline	None	S,I	-	
Doubleline	%DB.COnvert	False	K	-	
Doubleline	%DB.Mimic	False	K	-	
Doubleline	%DB.Offset	0.0	F	W	
Doubleline	%DB.Width	0.5	F	W	
Ellipse	%EL.Angle	Points	S,F	A	
Ellipse	%EL.Construction	Box	S	-	
Ellipse	%EL.Included	180.0	F	A	
Ellipse	%EL.Major	2.0	F	W	
Ellipse	%EL.MIMic	False	K	-	
Ellipse	%EL.MInor	1.0	F	W	
Ellipse	%EL.Projection	30.0	F	A	
Ellipse	%EL.Ratio	0.5	F	-	
Ellipse	%EL.Starting	0.0	F	A	
Error	%ER.Bell	On	X	-	
Error	%ER.Limit	1	I	-	
Error	%ER.Number	99	I	-	
Extend	%ET.Convert	False	K	-	
Extend	%ET.Function	Drag	S	-	
Extend	%ET.Global	Ask	S	-	
Extend	%ET.Length	1.0	F	W	
Extend	%ET.Mode	Relative	S	-	
Extrude	%EX.Depth	1.0	F	W	

CURRENT STATUS WORDS

<u>Command Name</u>	<u>CSW Name</u>	<u>Startup Value</u>	<u>Type</u>	<u>Data Units</u>	<u>Notes</u>
Extrude	%EX.Repeat	2	I	-	
Extrude	%EX.Surfaces	None	S	-	
Extrude	%EX.Z	0.0	F	W	
Field	%FD.Global	Ask	S	-	
File	%FF.FileName	none	S	-	
File	%FF.Function	List	S	-	
Fillet	%FL.COnvert	False	K	-	
Fillet	%FL.Mimic	False	K	-	
Fillet	%FL.Size	1.0	F	W	
Fillet_Existing	%FLE.Extend	Yes	X	-	
Fillet_Existing	%FLE.Mimic	False	K	-	
Fillet_Existing	%FLE.Size	1.0	F	W	
Font	%FT.NAme.[0-7]	default	S	-	
Font	%FT.Number	0	I	-	
Gap	%GA.Convert	False	K	-	
Gap	%GA.Global	Ask	S	-	
Gap	%GA.Pick	True	X	-	
Gap	%GA.Size	Points	S,F	W	
Gap_Arc	%GAA.Global	Ask	S	-	
Gap_Arc	%GAA.Pick	True	X	-	
Gap_Edge	%GAE.Global	Ask	S	-	
Gap_Edge	%GAE.Pick	True	X	-	
Gap_Edge	%GAE.Size	Points	S,F	W	
Get	%GE.Angle	0.0	F	A	
Get	%GE.ASsume	0	I,S	-	
Get	%GE.Class	0	I	-	
Get	%GE.Field	True	X	-	
Get	%GE.FOnt	default	I,S	-	
Get	%GE.Mirror	None	S	-	
Get	%GE.PLacement	Drag	S	-	
Get	%GE.Repeat	500	I	-	
Get	%GE.SCaLe	1.0	F	-	
Get	%GE.Symbol	none	S	-	
Get	%GE.Xscale	1.0	F	-	
Get	%GE.Yscale	1.0	F	-	
Goto	%GO.Label	none	S	-	
Grid	%GD.Display	Yes	X	-	
Grid	%GD.Large	1.0	F	W	
Grid	%GD.Small	0.25	F	W	
Grid	%GD.Type	Full	S	-	
Grid	%GD.XLarge	1.0	F	W	
Grid	%GD.Xsmall	0.25	F	W	
Grid	%GD.YLarge	1.0	F	W	
Grid	%GD.Ysmall	0.25	F	W	
Group	%GP.Class	0	I	-	
Group	%GP.Object	Any	S	-	
Group	%GP.Priority	125	I	-	
Group	%GP.Select	Pick	S	-	

CURRENT STATUS WORDS

<u>Command Name</u>	<u>CSW Name</u>	<u>Startup Value</u>	<u>Type</u>	<u>Data Units</u>	<u>Notes</u>
Group	%GP.Tag	none	S	-	
Help	%HE.Command	none	S	-	
If	%IF.Condition	False	X	-	Read only
If	%IF.Statement	none	S	-	
Insert	%IN.Distance	1.0	F	W	
Insert	%IN.Field	Yes	X	-	
Insert	%IN.Gap	1.0	F	W	
Insert	%IN.SScale	1.0	F	-	
Insert	%IN.Symbol	none	S	-	
Insert	%IN.Xscale	1.0	F	-	
Insert	%IN.Yscale	1.0	F	-	
Insert_Node	%INN.Global	Ask	S	-	
Insert_Vertex	%INV.Convert	False	K	-	
Insert_Vertex	%INV.Global	Ask	S	-	
Layer	%LA.Display.[0-255]	All	S,I	-	Same as %DP.L
Layer	%LA.Used.[0-255]	Used	I,S	-	Read only
Layer	%LA.Work	0	I	-	Same as %WK.L
Leaderline	%LL.Leader	Arrow	S	-	
Leaderline	%LL.LENght	0.125	F	P	
Leaderline	%LL.Mimic	False	K	-	
Leaderline	%LL.Placement	End	S	-	
Leaderline	%LL.Smart	Yes	X	-	
Leaderline	%LL.Width	0.0625	F	P	
Line	%LI.Function	Multiple	S	-	
Line	%LI.Mimic	False	K	-	
List	%LS.Classes.n	All	S,I	-	
List	%LS.Delimiter	none	S	-	
List	%LS.Filename	none	S	-	
List	%LS.Tag	none	S	-	
List	%LS.TYpe	none	S	-	
Load	%LD.Drawing	none	S	-	Same as %NW.D
Load	%LD.Force	False	K	-	and %SU.D
Log	%LG.Current	0	I	-	Read only
Log	%LG.Filename	none	S	-	
Log	%LG.Lines	0	I	-	
Lookup	%LU.Angle	0.0	F	A	Read only
Lookup	%LU.ATtribute	0	I	-	Read only
Lookup	%LU.Class	0	I	-	Read only
Lookup	%LU.CAp	None	S	-	Read only
Lookup	%LU.COpies	0	I	-	Read only
Lookup	%LU.COUnt	0	I	-	Read only
Lookup	%LU.Dimension	0.0	F	P	Read only
Lookup	%LU.Extension	0.0	S,F	P	Read only
Lookup	%LU.EXTStyle	Solid	S,I	-	Read only
Lookup	%LU.Fill	0	S,I	-	Read only
Lookup	%LU.FIEld	Off	X	-	Read only
Lookup	%LU.FOnt	0	S,I	-	Read only
Lookup	%LU.FORmat	Fractional	S,I	-	Read only

CURRENT STATUS WORDS

<u>Command Name</u>	<u>CSW Name</u>	<u>Startup Value</u>	<u>Type</u>	<u>Data Units</u>	<u>Notes</u>
Lookup	%LU.Gobject	Any	S	-	Read only
Lookup	%LU.GAp	0.0	F	W	Read only
Lookup	%LU.GXscale	0.0	F	-	Read only
Lookup	%LU.GXMaximum	0.0	F	W	Read only
Lookup	%LU.GXMinimum	0.0	F	W	Read only
Lookup	%LU.GXOrigin	0.0	F	W	Read only
Lookup	%LU.GXSize	0.0	F	W	Read only
Lookup	%LU.GYscale	0.0	F	-	Read only
Lookup	%LU.GYMaximum	0.0	F	W	Read only
Lookup	%LU.GYMinimum	0.0	F	W	Read only
Lookup	%LU.GYOrigin	0.0	F	W	Read only
Lookup	%LU.GYSize	0.0	F	W	Read only
Lookup	%LU.GZorigin	0.0	F	W	Read only
Lookup	%LU.Height	0.0	F	W,P	Read only
Lookup	%LU.HORizontal	0.0	F	-	Read only
Lookup	%LU.Included	0.0	F	A	Read only
Lookup	%LU.Justification	Cl	S	-	Read only
Lookup	%LU.Layer	0	I	-	Read only
Lookup	%LU.LEader	Arrow	S	-	Read only
Lookup	%LU.LLength	0.0	F	P	Read only
Lookup	%LU.LWidth	0.0	F	P	Read only
Lookup	%LU.MAajor	0.0	F	W	Read only
Lookup	%LU.MARker	Dot	S	-	Read only
Lookup	%LU.Members	0	I	-	Read only
Lookup	%LU.MInor	0.0	F	W	Read only
Lookup	%LU.MOobject	Any	S	-	Read only
Lookup	%LU.MTtag	none	S	-	Read only
Lookup	%LU.MTler	0	I	-	Read only
Lookup	%LU.NOhighlight	none	K	-	
Lookup	%LU.Oobject	Any	S	-	
Lookup	%LU.OFfset	0.0	F	P	Read only
Lookup	%LU.OVerrun	0.0	F	P	Read only
Lookup	%LU.Pen	0	I	-	Read only
Lookup	%LU.PRiority	125	I	-	Read only
Lookup	%LU.PLacement	End	S	-	Read only
Lookup	%LU.Rotate	0.0	F	A	Read only
Lookup	%LU.ROunding	0.0	F	W	Read only
Lookup	%LU.SEgment	0	I	-	Read only
Lookup	%LU.SHow	On	X	-	Read only
Lookup	%LU.Silent	False	K	-	
Lookup	%LU.SIze	0.0	F	W	Read only
Lookup	%LU.SLant	0.0	F	A	Read only
Lookup	%LU.SPacing	0.0	F	P	Read only
Lookup	%LU.STArting	0.0	F	A	Read only
Lookup	%LU.STyle	Solid	S,I	-	Read only
Lookup	%LU.Tag	none	S	-	Read only
Lookup	%LU.THeight	0.0	F	P	Read only
Lookup	%LU.TIers	0	I	-	Read only

CURRENT STATUS WORDS

Command Name	CSW Name	Startup Value	Data		
			Type	Units	Notes
Lookup	%LU.TWidth	0.0	F	P	Read only
Lookup	%LU.TYpe	Incremental	S	-	Read only
Lookup	%LU.Units	Inches	S	-	Read only
Lookup	%LU.Vertical	0.0	F	-	Read only
Lookup	%LU.VERTEx	0	I	-	Read only
Lookup	%LU.Width	0.0	F	W	Read only
Lookup	%LU.Xorigin	0.0	F	W	Read only
Lookup	%LU.Yorigin	0.0	F	W	Read only
Lookup	%LU.Zorigin	0.0	F	W	Read only
Macro	%MA.Arguments	none	S	-	
Macro	%MA.FileName	none	S	-	
Merge	%MG.Class	0	I	-	
Merge	%MG.Drawing	none	S	-	
Merge	%MG.Origin	False	K	-	
Merge	%MG.Scale	False	K	-	
Message	%MS.Statement	none	S	-	
Message	%MS.Time	5	I	-	
Mirror	%MR.Axis	Points	S	-	
Mirror	%MR.Copy	False	K	-	
Mirror	%MR.Object	Any	S	-	
Mirror	%MR.Pivot	Axis	S	-	
Mirror	%MR.Select	Pick	S	-	
Modifytext	%MT.FileName	none	S	-	
Modifytext	%MT.Global	Ask	S	-	
Move	%MV.Angle	0.0	F	A	
Move	%MV.Distance	0.0	F	W	
Move	%MV.Object	Any	S	-	
Move	%MV.Select	Pick	S	-	
Move	%MV.Vector	False	K	-	
Move	%MV.Xdistance	0.0	F	W	
Move	%MV.Ydistance	0.0	F	W	
Move	%MV.Ydistance	0.0	F	W	
Move_arc	%MVA.Global	Ask	S	-	
Move_Edge	%MVE.Global	Ask	S	-	
Move_Edge	%MVE.System	Viewplane	S	-	
Move_Edge	%MVE.Vector	False	K	-	
Move_Edge	%MVE.Xdistance	0.0	F	W	
Move_Edge	%MVE.Ydistance	0.0	F	W	
Move_Edge	%MVE.Zdistance	0.0	F	W	
Move_Member	%MVM.Angle	0.0	F	A	
Move_Member	%MVM.Distance	0.0	F	W	
Move_Member	%MVM.Global	Ask	S	-	
Move_Member	%MVM.Object	Any	S	-	
Move_Member	%MVM.Vector	False	K	-	
Move_Member	%MVM.Xdistance	0.0	F	W	
Move_Member	%MVM.Ydistance	0.0	F	W	
Move_Node	%MVN.Global	Ask	S	-	
Move_Node	%MVN.System	Viewplane	S	-	

CURRENT STATUS WORDS

Command Name	CSW Name	Startup Value	Type	Data Units	Notes
Move_Node	%MVN.Vector	False	K	-	
Move_Node	%MVN.Xdistance	0.0	F	W	
Move_Node	%MVN.Ydistance	0.0	F	W	
Move_Node	%MVN.Zdistance	0.0	F	W	
Move_Parallel	%MVP.Distance	0.0	F	W	
Move_Parallel	%MVP.Global	Ask	S	-	
Move_Segment	%MVS.Angle	0.0	F	A	
Move_Segment	%MVS.Convert	False	K	-	
Move_Segment	%MVS.Distance	0.0	F	W	
Move_Segment	%MVS.Global	Ask	S	-	
Move_Segment	%MVS.Vector	False	K	-	
Move_Segment	%MVS.Xdistance	0.0	F	W	
Move_Segment	%MVS.Ydistance	0.0	F	W	
Move_Vertex	%MVV.Angle	0.0	F	A	
Move_Vertex	%MVV.Convert	False	K	-	
Move_Vertex	%MVV.Distance	0.0	F	W	
Move_Vertex	%MVV.Global	Ask	S	-	
Move_Vertex	%MVV.Vector	False	K	-	
Move_Vertex	%MVV.Xdistance	0.0	F	W	
Move_Vertex	%MVV.Ydistance	0.0	F	W	
New	%NW.Drawing	none	S	-	Same as %LD.D
New	%NW.Force	False	K	-	and %SU.D
New	%NW.Height	22.0	F	P	Same as %SU.H
New	%NW.MScale	1.0	F	-	Same as %SU.MS
New	%NW.Plot	Inches	S	-	Same as %SU.P
New	%NW.Resolution	Fractional	F,S	-	
New	%NW.Scale	1.0	F	-	Same as %SU.S
New	%NW.Width	34.0	F	P	Same as %SU.W
New	%NW.WOrk	Inches	S	-	Same as %SU.WO
Open	%OP.Drawing	none	S	-	Same as %LD.D
Open	%OP.Force	False	K	-	Same as %LD.F
Origin	%OR.Point	False	K	-	
Origin	%OR.Reset	False	K	-	
Origin	%OR.X	0.0	F	W	
Origin	%OR.Y	0.0	F	W	
Overlay	%OV.Object	Any	S	-	
Overlay	%OV.Select	Pick	S	-	
Path	%PA.Current	varies	S	-	
Path	%PA.DELimeter	`	S	-	
Path	%PA.Drawings	varies	S	-	
Path	%PA.Macros	varies	S	-	
Path	%PA.Patterns	varies	S	-	
Path	%PA.Symbols	varies	S	-	
Pause	%PS.Time	5	I	-	
Pen	%PN.Display	0	I	-	Same as %DP.P
Pen	%PN.Used.[0-15]	Used	I	-	Read only
Pen	%PN.Work	0	I	-	Same as %WK.P
Point	%PT.Distance	1.0	F	W	

CURRENT STATUS WORDS

Command Name	CSW Name	Startup Value	Data		
			Type	Units	Notes
Point	%PT.DIVide	4	I	-	
Point	%PT.Function	Points	S	-	
Point	%PT.Marker	X	S	-	
Point	%PT.MImic	False	K	-	
Point	%PT.Percent	50.0	F	-	
Polygon	%PG.Angle	0.0	F	A	
Polygon	%PG.Construction	Radius	S	-	
Polygon	%PG.Mimic	False	K	-	
Polygon	%PG.Sides	6	I	-	
Polygon	%PG.SIZe	1.0	F	W	
Preferences	%PF.Angle	Degrees	S	-	
Preferences	%PF.Base	Last	S	-	
Preferences	%PF.Clockwise	False	X	-	
Preferences	%PF.COmmand	Off	X	-	
Preferences	%PF.CUrsor	Crosshairs	S	-	
Preferences	%PF.Format	Float	S	-	
Preferences	%PF.Global	Ask	S	-	
Preferences	%PF.Highlight	Fuzzy	S,I	-	
Preferences	%PF.Keys	Off	X	-	
Preferences	%PF.Lock	15.0	F	A	
Preferences	%PF.MACro	5	I	-	
Preferences	%PF.Marker	Diamond	S,I	-	
Preferences	%PF.Object	Any	S	-	
Preferences	%PF.Polar	0.0	F	A	
Preferences	%PF.PRompts	On	S	-	
Preferences	%PF.Rounding	1/1024	F	W	
Preferences	%PF.SAVe	50	I	-	
Preferences	%PF.SCroll	On	X	-	
Preferences	%PF.SEArcharea	0.13	F	S	
Preferences	%PF.Select	Pick	S	-	
Preferences	%PF.SMoothing	0.5	F	-	
Preferences	%PF.STOre	None	S	-	
Preferences	%PF.STring	>	S	-	
Preferences	%PF.TAbs	8	I	-	
Preferences	%PF.Tracking	None	S	-	
Preferences	%PF.Verbose	None	S	-	
Preferences	%PF.VIewlocate	On	X	-	
Preferences	%PF.Zoom	0.5	F	-	
Priority	%PR.Work	125	I	-	Same as %WK.P
Project	%PJ.Angle	45.0	F	A	
Project	%PJ.DEpth	0.75	F	W	
Project	%PJ.Distance	%NW.W	F	W	Dwg. width
Project	%PJ.Height	%NW.H	F	W	Dwg. height
Project	%PJ.Type	Parallel	S	-	
Prompt	%PM.Prompt	none	S	-	
Prompt	%PM.String	>>	S	-	
Prompt	%PM.Type	None	S	-	
Prompt	%PM.Variable	A	V	-	

CURRENT STATUS WORDS

Command Name	CSW Name	Startup	Data		
		Value	Type	Units	Notes
Put	%PU.Force	False	K	-	
Put	%PU.SCaLe	False	K	-	
Put	%PU.Symbol	none	S	-	
Quit	%QT.Force	False	K	-	
Readline	%RL.Evaluate	No	X	-	
Readline	%RL.FileName	none	S	-	
Readline	%RL.Prompt	none	S	-	
Readline	%RL.Statement	none	S	-	
Readline	%RL.STRing	>>	S	-	
Record	%RC.Active	Off	S	-	
Record	%RC.FileName	none	S	-	
Record	%RC.FORce	False	K	-	
Rectangle	%RE.Angle	0.0	S,F	A	
Rectangle	%RE.Construction	Diagonals	S	-	
Rectangle	%RE.Height	1.0	F	W	
Rectangle	%RE.Mimic	False	K	-	
Rectangle	%RE.Width	1.0	F	W	
Replace	%RP.Field	No	X	-	
Replace	%RP.Global	Ask	S	-	
Replace	%RP.SElect	Pick	S	-	
Replace	%RP.Symbol	none	S	-	
Replace	%RP.Tag	none	S	-	
Revolve	%RV.Connect	Yes	X	-	
Revolve	%RV.Mode	Relative	S	-	
Revolve	%RV.Order	XYZ	S	-	
Revolve	%RV.Pivot	Point	S	-	
Revolve	%RV.Repeat	1	I	-	
Revolve	%RV.Scale	1.0	F	-	
Revolve	%RV.SYstem	Viewplane	S	-	
Revolve	%RV.Xangle	0.0	F	A	
Revolve	%RV.Yangle	0.0	F	A	
Revolve	%RV.Zangle	0.0	F	A	
Rotate	%RT.Angle	90.0	F	A	Same as %TF.A
Rotate	%RT.Mode	Relative	S	-	
Rotate	%RT.Object	Any	S	-	
Rotate	%RT.Pivot	Point	S	-	
Rotate	%RT.Select	Pick	S	-	
Rotate_Three	%RTT.Global	Ask	S	-	
Rotate_Three	%RTT.Order	XYZ	S	-	
Rotate_Three	%RTT.Pivot	Point	S	-	
Rotate_Three	%RTT.System	Viewplane	S	-	
Rotate_Three	%RTT.Xangle	0.0	F	A	Same as %TFT.X
Rotate_Three	%RTT.Yangle	0.0	F	A	Same as %TFT.Y
Rotate_Three	%RTT.Zangle	0.0	F	A	Same as %TFT.Z
Rotate_World	%RTW.Mode	Relative	S	-	
Rotate_World	%RTW.Order	XYZ	S	-	
Rotate_World	%RTW.Pivot	Current	S	-	
Rotate_World	%RTW.System	Viewplane	S	-	

CURRENT STATUS WORDS

Command Name	CSW Name	Startup	Data	
		Value	Type	Units Notes
Rotate_World	%RTW.Xangle	0.0	F	A
Rotate_World	%RTW.Yangle	0.0	F	A
Rotate_World	%RTW.Zangle	0.0	F	A
Rounding	%RD.Active	On	X	-
Rounding	%RD.Spacing	0.125	F	W
Rounding	%RD.Type	Orthogonal	S	-
Rounding	%RD.Xspacing	0.125	F	W
Rounding	%RD.Yspacing	0.125	F	W
Save	%SV.Classes.[0-255] All	S,I	-	-
Save	%SV.Drawing	none	S	-
Save	%SV.Force	False	K	-
Scale	%SC.Mode	Relative	S	-
Scale	%SC.Object	Any	S	-
Scale	%SC.Pivot	Point	S	-
Scale	%SC.Scale	1.0	F	- Same as %TF.S
Scale	%SC.SElect	Pick	S	-
Scale	%SC.Xscale	1.0	F	- Same as %TF.X
Scale	%SC.Yscale	1.0	F	- Same as %TF.Y
Scale_Three	%SCT.Global	Ask	S	-
Scale_Three	%SCT.Pivot	Point	S	-
Scale_Three	%SCT.Scale	1.0	F	- Same as %TFT.S
Select	%SL.Filename	none	S	-
Select	%SL.Name	none	S	-
Select	%SL.Prompt	none	S	-
Select	%SL.Result	none	S	- Read only
Select	%SL.Type	None	S	-
Setup	%SU.DDescription	none	S	-
Setup	%SU.Drawing	none	S	- Same as %LD.D
Setup	%SU.Height	22.0	F	P Same as %NW.H
Setup	%SU.Mscale	1.0	F	- Same as %NW.M
Setup	%SU.Plot	Inches	S	- Same as %NW.P
Setup	%SU.Scale	1.0	F	- Same as %NW.S
Setup	%SU.Width	34.0	F	P Same as %NW.W
Setup	%SU.WOrk	Inches	S	- Same as %NW.WO
Sketch	%SK.Distance	0.0625	F	W
Sketch	%SK.Mimic	False	K	-
Spline	%SP.Mimic	False	K	-
Status	%SS.Command	none	S	-
Stretch	%SR.Angle	0.0	F	A
Stretch	%SR.Convert	True	X	-
Stretch	%SR.Distance	0.0	F	W
Stretch	%SR.Global	Ask	S	-
Stretch	%SR.Select	Inside	S	-
Stretch	%SR.Vector	False	K	-
Stretch	%SR.Xdistance	0.0	F	W
Stretch	%SR.Ydistance	0.0	F	W
Style	%ST.Gap.n	0	F	P
Style	%ST.Long.n	0	F	P

CURRENT STATUS WORDS

Command Name	CSW Name	Startup Value	Data		
			Type	Units	Notes
Style	%ST.LRepeat.n	0	I	-	
Style	%ST.Short.n	0	F	P	
Style	%ST.SRepeat.n	0	I	-	
Style	%ST.Work	Solid	S,I	-	Same as %WK.S
Substitute	%SB.Global	Ask	S	-	
Substitute	%SB.Line	0	I	-	
Substitute	%SB.New	none	S	-	
Substitute	%SB.Old	none	S	-	
Substitute	%SB.Select	Pick	S	-	
System	%SM.Statement	none	S	-	
Tag	%TG.Global	No	S	-	
Tag	%TG.Tag	none	S	-	
Text	%TX.Angle	Points	S,F	A	
Text	%TX.ATtribute	1	I	-	
Text	%TX.Dialog	False	K	-	
Text	%TX.Field	False	X	-	
Text	%TX.FILename	none	S	-	
Text	%TX.Font	default	S,I	-	
Text	%TX.Height	0.125	F	P	
Text	%TX.HORizontal	1.0	F	-	
Text	%TX.Justification	LC	S	-	
Text	%TX.Mimic	False	K	-	
Text	%TX.Offset	0.0625	F	P	
Text	%TX.Placement	Points	S	-	
Text	%TX.Reset	False	K	-	
Text	%TX.Slant	0	F	A	
Text	%TX.Type	Paragraph	S	-	
Text	%TX.Vertical	1.5	F	-	
Text	%TX.Width	0.75	F	-	
Timer	%TI.Current	0:00:00	S	-	
Timer	%TI.Function	List	S	-	
Timer	%TI.Previous	0:00:00	S	-	
Timer	%TI.Total	0:00:00	S	-	Read only
Transform	%TF.Angle	90.0	F	A	Same as %RT.A
Transform	%TF.Mirror	None	S	-	
Transform	%TF.MOde	Relative	S	-	
Transform	%TF.Object	Any	S	-	
Transform	%TF.Pivot	Point	S	-	
Transform	%TF.Scale	1.0	F	-	Same as %SC.S
Transform	%TF.SElect	Pick	S	-	
Transform	%TF.Xscale	1.0	F	-	Same as %SC.X
Transform	%TF.Yscale	1.0	F	-	Same as %SC.Y
Transform_Three	%TFT.Global	Ask	S	-	
Transform_Three	%TFT.Order	XYZ	S	-	
Transform_Three	%TFT.Pivot	Point	S	-	
Transform_Three	%TFT.Scale	1.0	F	-	Same as %SCT.S
Transform_Three	%TFT.SYstem	Viewplane	S	-	
Transform_Three	%TFT.Xangle	0.0	F	A	Same as %RTT.X

CURRENT STATUS WORDS

Command Name	CSW Name	Startup	Data		
		Value	Type	Units	Notes
Transform_Three	%TFT.Yangle	0.0	F	A	Same as %RTT.Y
Transform_Three	%TFT.Zangle	0.0	F	A	Same as %RTT.Z
Ungroup	%UG.Tiers	1	I,S	-	
View	%VW.Function	Plan	S	-	
View	%VW.Isometric	False	K	-	
Wall	%WA.CAp	Yes	S	-	
Wall	%WA.Centerline	None	S,I	-	
Wall	%WA.Gap	Yes	S	-	
Wall	%WA.Mimic	False	K	-	
Wall	%WA.Offset	0.0	F	W	
Wall	%WA.Width	0.5	F	W	
Weight	%WT.Display	Yes	S	-	
Weight	%WT.Holder.[0-15]	0	I	-	
Weight	%WT.Pen	0	I	-	
Weight	%WT.Traces.[0-15]	1	I	-	
Weight	%WT.Width.[0-15]	0.0	F	P	
Where	%WH.Units	World	S	-	
Window	%WD.Function	Define	S	-	
Window	%WD.Number	1	I,S	-	
Wireframe	%WF.Convert	False	K	-	
Wireframe	%WF.Z	0.0	F	W	
Work	%WK.Class	0	I	-	Same as %CL.W
Work	%WK.Delete	none	S	-	
Work	%WK.Layer	0	I	-	Same as %LA.W
Work	%WK.List	False	K	-	
Work	%WK.Mimic	False	K	-	
Work	%WK.Name	none	S	-	
Work	%WK.Pen	0	I	-	Same as %PN.W
Work	%WK.PRiority	125	I	-	Same as %PR.W
Work	%WK.Style	Solid	S,I	-	Same as %ST.W
Work	%WK.Workset	none	S	-	
Workplane	%WP.Function	Viewplane	S	-	
Writeline	%WL.FileName	none	S	-	
Writeline	%WL.Newline	Yes	X	-	
Writeline	%WL.Statement	none	S	-	
Zdepth	%ZD.Depth	0.0	F	W	
-	%_.Command	0	I	-	Read only
-	%_.Date	mm/dd/yy	S	-	Read only
-	%_.Error	0	I	-	Read only
-	%_.Last	0	I	-	Read only
-	%_.Lines	0	I	-	Read only
-	%_.Name	PEG2	I	-	Read only
-	%_.Previous	0	I	-	Read only
-	%_.Score	100	I	-	Read only
-	%_.SEconds	0	I	-	Read only
-	%_.Time	hh:mm:ss	S	-	Read only
-	%_.Version	3.4	S	-	Read only

Variables

A variable is a user-defined storage location that may contain either a coordinate, a floating point number or a character string of not more than thirty-two characters. A variable's name begins with the special character for the type of variable (described below) followed by an alphanumeric string. Variable names can be up to ten characters in length and must start with a letter (a thru z). Case is not important in variable names. For example, "&NUMBER" and "&number" are both names for the same variable.

Variables are not declared formally, as with some programming languages. When a variable is required for storage, just enter a valid variable name. The macro interpreter will automatically assign a part of the computer memory to that name. There are four types of variables. This table shows the names we have given each type and its format.

<u>Variable Type</u>	<u>Format</u>
Local	&name
Global	name
System	A thru Z

Local variables begin with an ampersand (&) and have up to a ten character name. Their range is only within the macro they are used in. The values that local variables remember cannot be passed between macros. For example, if Macro A executes Macro B, and they both have a local variable named &Number, the two variables are separate and distinct, and neither macro has access to the other's &Number variable.

Global variables begin with a pound (number) sign (#) and also have up to a ten character name. Their range is within the macro they are used in and any lower level macro. A lower level macro is a macro that is executed by another macro. In the example used above, if Macro A used a variable named HEIGHT, Macro B would have access to the same HEIGHT variable.

System variables are a special class of global variables names A through Z. These 26 variables are always available, even at the interactive level outside of a macro. Be careful when using the variable A. This system variable is used (as a default) by some commands, such as the Prompt command.

Remember that variable names can contain numbers and letters, but must start with a letter. Also remember that case is unimportant. Try to keep your variable names descriptive of the value being stored. It is also a good idea to set a case standard (upper case, lower case, or mixed) for your variables as it makes them easier to spot in the Macro. Some examples of good variable names are: Width, &Answer, &Quantity, Name, Xsize.

Variables are defined and set in an Assignment Statement. See the Calculator command, specifically the paragraph on Assignment Statements for more information.

Current Arguments

Current Arguments are temporary system variables that automatically remember values generated by a command. Only two commands access the Current Arguments - Macro and Readline. How these commands use the Current Arguments is explained in their documentation.

There are ten Current Arguments, named %Arg0 through %Arg9. Like all variables, case is unimportant. Current Arguments can contain character strings or floating point numbers. %Arg0 will always contain a number from 1 through 9 that indicates the number of Current Arguments that have been set by the last command. %Arg1 through %Arg9 will contain 0 on startup. Once a Current Argument is used it will always contain the last value assigned to it. There is no automatic initialization of the Current Arguments. If a command were to set the value of %Arg0 and %Arg1, the contents of %Arg2 through %Arg9 would be unchanged.

One use for Current Arguments within a macro is to pass numeric or character values from a calling macro to a called macro in an argument list. This lets you write macros that accept data from the user or another macro without having to set global or system variables.

As an example, let's write a macro to set the current pen number, line style and point type to the same user entered value. Let's call the macro "SetInput". It would look like this:

```
Input Pen=%Arg1 Style=%Arg1
Point Marker=%Arg1
Return
```

Now to set these values to 7, for example, the user would pass the value to the macro by entering the number after the macro name. Like this:

```
Macro SetInput 7
```

The number seven will be remembered in %Arg1 for use in the macro. %Arg0 would be set to 1 because only one Current Argument was set by this command. The values currently in %Arg2 through %Arg9 will not be affected.

Glossary

The following is an alphabetical listing of the technical terminology used in this manual, accompanied by an explanation. All definitions and examples refer to the item's meaning in reference to the TurboCAD CAD package. Other meanings may be possible or common outside of this environment.

Alphanumeric string

A string of characters containing any combination of letters, numbers and other printable characters. Examples: "Jeff's Junk", "2nd Floor Plan", "234-56.3b", "Chair45", "\$%^*(".

Bitmap

An image on your screen that is defined by pixels (dots) that are black or white, or some color.

Box

Many edit commands allow you to select objects by enclosing them in a rectangular area that we call a "box." These commands include: Move, Move Vertex, Copy, Delete, Mirror, Group and more. The option is called Select and has values that will allow you to select objects that are inside, outside, touching, not inside, not outside or not touching.

Centerpoint

The point in the center of a graphic primitive like an arc, circle or rectangle.

Click

The act of placing the cursor at a desired location and pressing the mouse key.

Command

Any one of the TurboCAD commands as listed in the Command List. The command that you are currently in is displayed on the right side of the Prompt line.

Command Line

The area on the Status palette where you can enter commands.

Command Mode

This is the name for the state TurboCAD is in when it expects a command to be entered. Any time that the crosshairs are not displayed, you are in the Command mode.

Cursor

This is the pointer that appears on your screen that tracks with the movement of the mouse. The cursor can take on several different forms in TurboCAD. The crosshairs cursor indicates that you are in the Graphic mode and is used for locating points on your drawing. The standard pointer indicates that you are in the Command mode awaiting the execution of the next command. On Macintosh computers, the pointer cursor is also used when selecting from a menu. The I-beam cursor is used for editing text wording, and the wristwatch cursor's meaning needs no explanation.

Crosshairs

Those horizontal and vertical crossing lines on your screen that you use to locate points when you are in the Graphic mode of TurboCAD.

Current Status Word

A special type of variable that is automatically set by the command when executed. See the **Current Status Words** section for more information.

Default

A default is an unspecified value; the value that is used if none was given explicitly. On startup, many TurboCAD commands default to some predetermined values.

Dialog

A titled box that is displayed by some commands to show you their options.

Delimiter

The character used to separate options on the command line. Valid TurboCAD delimiters are a comma, tab or space. See the Entering Commands section for more information.

Double-Click

Pressing the mouse key twice in rapid succession.

Drag

The act of dynamically moving an object across the screen. Many commands in TurboCAD drag objects by attaching them to the crosshairs so they can be moved with the mouse. You could consider the object being dragged as a new type of temporary cursor.

Endpoint

The last point located to create a graphic primitive like an arc, line, doubleline, etc.

Enter

The term "Enter" when seen in this manual or as a prompt on your screen, means that information is fed to the program by typing it on the keyboard.

Execute

To activate a command. This is done by typing the command and desired options followed by a <Return>.

Exit

To terminate a command. This is done by entering a <Return>.

Filename

The name of a file stored on disk. Like a TurboCAD drawing or symbol, for example.

Global

To make one change effective across several objects or the entire drawing, as opposed to singular changes. See the **Common Options** section for more information.

Graphic Mode

This is the name for the state TurboCAD is in when it expects graphic input, such as the location of a point. You are in the Graphic mode any time that the crosshairs are displayed.

GLOSSARY

Group

Objects that are treated as a unit (like they are glued together). Groups are created using the Group command.

Interactive

Conversation between the operator and the program. The operator "talks" to the program through the keyboard and mouse, and the program responds with prompts to the operator. This back and forth conversation is known as interactive use.

Keyword

The name of a command's option. An option's value is set by entering the keyword, an equals sign, and the value. Example: "Line Style=Dashed". See the **Entering Commands** section for more information.

Keyword-only

A command option that does not require a value. The option is activated by entering the option name alone. Example: "Line Convert". See the **Entering Commands** section for more information.

Label

A alphanumeric string ending with a colon placed at the start of a line in a macro that is used as a location that can be branched to using the Goto command. See the **Macros** chapter for more information.

Literal string

An alphanumeric string enclosed in quotes. See the **Entering Commands** section for more information.

Locate

Usually refers to locating a point. This means that the crosshairs will be moved to the desired location on the screen and the mouse key or a key on the keyboard is pressed. See the **Locating Points** section for more information.

Logical expression

A comparison of relational expressions to determine whether the entire expression is true or false. See the Macros chapter for more information.

Member

One of many primitives that make up a group.

Midpoint

The point halfway along an arc or line segment.

News Area

A window or area of the screen (usually in the upper left corner) that is used to display help and status messages. The News Area can be suppressed with the Prompts option of the Preferences command. You might also want to see the Message command.

Operand

A value upon which an operation is performed - data. See the Calculator command.

Operator

A symbol used to represent a function, such as "+" for mathematical addition. See the Calculator command.

Option

This is the general name for a command parameter. An option could be selectable from a pull-down sub-menu or a dialog. Many options are comprised of a keyword, an equals sign and a value. The value can be entered by the operator or selected from a pop-up menu.

Origin

The location on your drawing that is an absolute coordinate location of zero in the X axis and zero in the Y axis. The drawing origin is set by the Origin command. Groups of objects on your drawing also have as origin that is set by the Group command.

Parameter

See Option.

Path

The list of hierarchical folders or directories that leads to a file. Example: "HardDisk:Drawings:Schematic". Colons (:) are used as folder delimiters on Macintosh computers. See the Path command for more information.

Pick

Usually refers to picking an existing object on your drawing. This is done by locating the crosshairs on an object and pressing the mouse key or a key on the keyboard.

Plot Units

The units that your drawing is plotted in. These are units measured on the plotted paper. The drawing scale determines how many work units will be plotted per plot unit.

Precedence

The order that items are evaluated in an expression. See the Calculator command.

Primitive

A single graphic object created using a TurboCAD draw command. The most basic geometric element in a TurboCAD drawing file. Lines, circles and rectangles are examples of primitives.

Prompt Line

The area where Prompts appear on the Prompts palette.

Quit

To terminate or exit a program.

Relational expression

A comparison of two values to determine how they relate to one another - are they equal, is the first greater than the second, etc. See the **Macros** chapter for more information.

Rubberbanding

Consider a command like "Move Vertex" that allows you to move one endpoint of a line segment. As you move the endpoint the segment stretches to show how the new line would look. This dynamic stretching is known as rubberbanding. Many objects can be rubberbanded as they are being drawn, like rectangles, circles, etc.

GLOSSARY

Select

Objects on your drawing are selected for edit by picking them or defining a box. The box allows multiple objects to be selected simultaneously. See the **Common Options** section for more information on the Select option.

Segment

The segment of a line, doubleline or fillet that is between two vertices. A vector from one point to another.

Shift-Click

Pressing the mouse key while holding down the <Shift> key.

Startpoint

The first point located to create the visible part of a graphic primitive like an arc, line, doubleline, etc. In the case of lines, doublelines and fillets, the startpoint is also the origin location.

Stroked

An object that is defined by vectors. This most commonly refers to stroked Text fonts or Crosshatch patterns.

Toggle

If something is Off, turn it On, and vice versa. This applies to all switch values (True, False, Yes, No, All, None, On, Off).

Unary operator

An operator that requires only one operand. For example, the minus in "-3" is said to be a unary minus. See the Calculator command.

Work Units

The units that you use on your drawing to measure distances. You always work on your drawing in full scale work units, but these units may be scaled when plotted.

Value

This is the actual value or setting of an option. Values are either selected from a pop-up menu or entered through the keyboard.

Value Type

A common type of value that is shown in the documentation in italics. For example, the *number* value type means the value must be a number. See the **Entering Commands** section.

Vector

A straight line connecting two points. One segment of a line is a vector.

Vertex

The endpoint of a segment or arc. Each segment has two vertices. Two segments can share a common vertex.

Workarea

This is the name for the logical area where the drawing currently being worked on is stored. All of the current drawing is in the workarea, even if only part of the drawing is visible on your screen.
