AutoCAD® 2000, 2000i & 2002 System Variables

Highlighted numbers are preferred or standard
Variables pertain to all versions except where noted (200XX)

System Variable — **Command(s) affected**

_PKSER — *Environment*
(Read-only)
Type: Integer
Saved in: Registry
Returns the serial number assigned to AutoCAD

ACADLSPASDOC — TOOLS/OPTIONS
Type: Integer
Saved in: Registry
Initial value: 0
Controls whether AutoCAD loads the acad.lsp file into every drawing or just the first drawing opened in an AutoCAD session.

0 Loads acad.lsp into just the first drawing opened in an AutoCAD session
1 Loads acad.lsp into every drawing opened

ACADPREFIX — TOOLS/OPTIONS
(Read-only)
Type: String
Not saved
Stores the directory path, if any, specified by the ACAD environment variable, with path separators appended if necessary.

ACADVER — *Environment*
(Read-only)
Type: String
Not saved
Returns the AutoCAD version number. This variable differs from the DXF file $ACADVER header variable, which contains the drawing database level number.

ACISOUTVER — ACISOUT
Type: Integer — EXPORT
Not saved
Initial value: 40
Controls the ACIS version of SAT files created using the ACISOUT command. Currently, ACISOUT only supports a value of 15 through 18, 20, 21, 30, and 40.

ADCSTATE — ADCENTER
(undocumented) — ADCCLOSE
(Read-only)
Type: Integer

Not saved
Initial value: 0
Hold the display state of the AutoCAD Design Center

0 Not displayed in the environment
1 Currently displayed in the environment

AFLAGS DDATTDEF
Type: Integer ATTDEF
Not saved
Initial value: 0
Sets attribute flags for ATTDEF bitcode. The value is the sum of the following:

0 No attribute mode selected
1 Invisible
2 Constant
4 Verify
8 Preset

ANGBASE DDUNITS
Type: Real UNITS
Saved in: Drawing
Initial value: 0.0000
Sets the base angle to 0 with respect to the current UCS.

ANGDIR DDUNITS
Type: Integer UNITS
Saved in: Drawing
Initial value: 0
Sets the positive angle direction from angle 0 with respect to the current UCS.

0 Counterclockwise
1 Clockwise

APBOX DDOSNAP
Type: Integer OSNAP
Saved in: Registry
Initial value: 0
Turns the AutoSnap aperture box on or off. The aperture box is displayed in the center of the crosshairs when you snap to an object.

0 Aperture box is not displayed
1 Aperture box is displayed

APERTURE DDOSNAP
Type: Integer OSNAP
Saved in: Registry
Initial value: 10

Sets the display size for the aperture, in pixels. The aperture is the selection tool used in drawing commands. The valid range is 1-50.

AREA
(Read-only)
Type: Real
Not saved

AREA
DBLIST
LIST

AREA is both a command and a system variable. The AREA system variable stores the last area computed by the AREA, LIST or DBLIST commands. Because entering area at the Command prompt invokes the AREA command, you must use the SETVAR command to access the AREA system variable.

ATTDIA
Type: Integer
Saved in: Registry
Initial value: 0

DDATTE

Controls whether the -INSERT command uses a dialog box for attribute value entry. See "INSERT Command Line." Attribute interface for entry:

0 Issues prompts on the command line
1 Uses a dialog box

ATTMODE
Type: Integer
Saved in: Drawing
Initial value: 1

ATTDISP

Controls display of attributes. Display of attributes on screen:

0 Off: Makes all attributes invisible
1 Normal: Retains current visibility of each attribute: visible attributes are displayed; invisible attributes are not
2 On: Makes all attributes visible

ATTREQ
Type: Integer
Saved in: Registry
Initial value: 1

INSERT

Determines whether the INSERT command uses default attribute settings during insertion of blocks. Determines if Attribute defaults are used:

0 Assumes the defaults for the values of all attributes
1 Turns on prompts or dialog box for attribute values, as specified by ATTDIA

AUDITCTL
Type: Integer
Saved in: Registry
Initial value: 0

AUDIT

Controls whether the AUDIT command creates an audit report (ADT) file.

0 Prevents writing of ADT file
1 Audit log (ADT) file is written

AUNITS DDUNITS
Type: Integer UNITS
Saved in: Drawing
Initial value: 0
Sets units for angles.

0 Decimal degrees
1 Degrees/minutes/seconds
2 Grads
3 Radians
4 Surveyor's units

AUPREC DDUNITS
Type: Integer UNITS
Saved in: Drawing
Initial value: 0
Sets the number of decimal places displayed for all read-only angular units, and for all editable angular units whose precision is less than or equal to the current AUPREC value. For editable angular units whose precision is greater than the current AUPREC value, the true precision is displayed. AUPREC does not affect the display precision of dimension text (see DIMSTYLE). *Do you understand this?*

AUTOSNAP DDOSNAP
Type: Integer OSNAP
Saved in: Registry
Initial value: 63
Controls AutoSnap marker, tooltip, and magnet. Also turns on polar and object snap tracking, and controls the display of polar and object snap tracking tooltips. The system variable value is the sum of the following bit values:

0 Turns off the AutoSnap marker, tooltips, and magnet. Also turns off polar tracking, object snap tracking, and tooltips for polar and object snap tracking
1 Turns on the AutoSnap marker
2 Turns on the AutoSnap tooltips
4 Turns on the AutoSnap magnet
8 Turns on polar tracking
16 Turns on object snap tracking
32 Turns on tooltips for polar tracking and object snap tracking

BACKZ DVIEW
(Read-only)
Type: Real
Saved in: Drawing
Initial value: 0.0000
Stores the back clipping plane offset from the target plane for the current viewport, in drawing units. Meaningful only if the back clipping bit in VIEWMODE is on. The distance of

the back clipping plane from the camera point can be found by subtracting BACKZ from the camera-to-target distance.

BINDTYPE
Type: Integer
Not Saved
initial value: 0
Controls how xref names are handled when binding xrefs or editing xrefs in place.

XREF
REFEDIT

0 Traditional binding behavior (“xref1|one” becomes “xref0one”)
1 Insert-like behavior (“xref1|one” becomes “one”)

BLIPMODE
Type: Integer
Saved in: Registry
Initial value: 0
Controls whether marker blips are visible.

DDRMODES
BLIPMODE

0 Turns off marker blips
1 Turns on marker blips

CDATE
Type: Real
Not saved
Shows current calendar date and time. In the format yyyymmdd.hhhhmmss.

TIME

CECOLOR
Type: String
Saved in: Drawing
Initial value: “BYLAYER”
Sets the color of new objects.

DDEMODES
COLOR
STATUS

CELTSCALE
Type: Real
Saved in: Drawing
Initial value: 1.0000

DDLTYPE
DDEMODES
LINETYPE
STATUS

Sets the current object linetype scaling factor. Sets the linetype scaling for new objects relative to the LTSCALE command setting. A line created with CELTSCALE = 2 in a drawing with LTSCALE set to 0.5 would appear the same as a line created with CELTSCALE = 1 in a drawing with LTSCALE = 1.

CELTYPE
Type: String
Saved in: Drawing
Initial value: “ByLayer”
Sets the linetype of new objects.

DDLTYPE
DDEMODES
LINETYPE
STATUS

CELWEIGHT LWEIGHT
Type: Integer
Saved In: Drawing
Initial value: “ByLayer”
Sets the lineweight of new objects.

-1 Sets the lineweight to “ByLayer.”
-2 Sets the lineweight to “ByBlock.”
-3 Sets the lineweight to “Default.” “Default” is controlled by the LWDEFAULT system variable.

Other valid values entered in millimeters include 0, 5, 9, 13, 15, 18, 20, 25, 30, 35, 40, 50, 53, 60, 70, 80, 90, 100, 106, 120, 140, 158, 200, and 211.
All values must be entered in millimeters. (Multiply a value by 25.4 to convert values from inches to millimeters.)

CHAMFERA CHAMFER
Type: Real
Saved in: Drawing
Initial value: 0.5000
Sets the first chamfer distance.

CHAMFERB CHAMFER
Type: Real
Saved in: Drawing
Initial value: 0.5000
Sets the second chamfer distance.

CHAMFERC CHAMFER
Type: Real
Saved in: Drawing
Initial value: 1.0000
Sets the chamfer length.

CHAMFERD CHAMFER
Type: Real
Saved in: Drawing
Initial value: 0.0000
Sets the chamfer angle.

CHAMMODE CHAMFER
Type: Integer
Not saved
Initial value: 0
Sets the input method by which AutoCAD creates chamfers.

0 Requires two chamfer distances
1 Requires one chamfer distance and an angle

CIRCLERAD CIRCLE
Type: Real
Not saved
Initial value: 0.0000
Sets the default circle radius. A zero indicates no default.

CLAYER DDLMODES
Type: String LAYER
Saved in: Drawing STATUS
Initial value: “0”
Sets the current layer.

CMDACTIVE *Any command*
(Read-only)
Type: Integer
Not saved
Stores the bitcode that indicates whether an ordinary command, transparent command, script, or dialog box is active. The value is the sum of the following:

1 Ordinary command is active
2 Ordinary command and a transparent command are active
4 Script is active
8 Dialog box is active
16 AutoLISP is active (only visible to an ObjectARX-defined command)

CMDDIA *Environment*
Type: Integer
Saved in: Registry
Initial value: 1
Controls the display of dialog boxes for the PLOT command and the external database (ASE) commands.

0 No dialog boxes
1 Shows dialog boxes

CMDECHO *Environment*
Type: Integer
Not saved
Initial value: 1
Controls whether AutoCAD echoes prompts and input during the AutoLISP command function.

0 Turns off echoing
1 Turns on echoing

CMDNAMES *Any command*
(Read-only)

Type: String
Initial value: “SETVAR”
Not saved
Displays the names of the active and transparent commands. For example, LINE’ZOOM indicates that the ZOOM command is being used transparently during the LINE command. This variable is designed for use with programming interfaces such as AutoLISP, DIESEL, and ActiveX Automation.

CMLJUST MLINE
Type: Integer
Saved in: Drawing
Initial value: 0
Specifies multiline justification.

0 Top
1 Middle
2 Bottom

CMLSCALE MLINE
Type: Real
Saved in: Drawing
Initial value: 1.0000
Controls the overall width of a multiline. A scale factor of 2.0 produces a multiline twice as wide as the style definition. A zero scale factor collapses the multiline into a single line. A negative scale factor flips the order of the offset lines (that is, the smallest or most negative is placed on top when the multiline is drawn from left to right). Multline scale:

0 Single line
1 Default width
n Distance between lines

CMLSTYLE MLINE
Type: String MLSTYLE
Saved in: Drawing
Initial value: “STANDARD”
Sets the multiline style that AutoCAD uses to draw the multiline.

COMPASS 3DORBIT
Type: Integer
Not saved
Initial value: 0
Controls whether the 3D compass is on or off in the current viewport.

0 Turns off the 3D compass
1 Turns on the 3D compass

COORDS *Environment*
Type: Integer
Saved in: Registry

Initial value: 1
Controls when coordinates are updated on the status line.

0 Coordinate display is updated as you specify points with the pointing device
1 Display of absolute coordinates is updated continuously
2 Display of absolute coordinates is updated continuously, and distance and angle from last point are displayed when a distance or angle is requested

CPLOTSTYLE PLOT
Type: String
Saved in: Drawing
Initial Value: “ByLayer”
Controls the current plot style for new objects. The AutoCAD defined values are:
“ByLayer”
“ByBlock”
“Normal”
“User Defined”

CPROFILE TOOLS/OPTIONS
(Read-only)
Type: String
Saved in: Registry
Initial value: <<Unnamed Profile>>
Displays the name of the current profile. For more information on profiles, see the OPTIONS command.

CTAB *Environment*
(Read-only)
Type: String
Saved in: Drawing
Initial value: “Model”
Returns the name of the current (model or layout) tab in the drawing. Provides a means for the user to determine which tab is active.

CURSORSIZE TOOLS/OPTIONS
Type: Integer
Saved in: Registry
Initial value: 5
Determines the size of the crosshairs as a percentage of the screen size. Valid settings range from 1 to 100 percent. When set to 100, the crosshairs are full-screen and the ends of the crosshairs are never visible. When less than 100, the ends of the crosshairs may be visible when the cursor is moved to one edge of the screen.

CVPORT VPORTS
Type: Integer
Saved in: Drawing
Initial value: 2
Sets the identification number of the current viewport. You can change this value, which changes the current viewport, if the following conditions are met:
The identification number you specify is that of an active viewport.

A command in progress has not locked cursor movement to that viewport.
Tablet mode is off.

DATE *Environment*
(Read-only)
Type: Real
Not saved
Displays current date and time in calendar format: yyyy/mm/dd hh:mm:ss.ss

DBCSTATE DBCONNECT
(undocumented) DBCLOSE
(Read-only)
Type: Integer
Initial value: 0
Display state for the Objects Properties dockable container.

0 Not displayed in the environment
1 Currently displayed in the environment

DBMOD *Used by most commands*
(Read-only)
Type: Integer
Not saved
Initial value: 5
Indicates the drawing modification status using bitcode. The value is the sum of the following:

1 Object database modified
2 Symbol table
4 Database variable modified
8 Window modified
16 View modified

AutoCAD resets the DBMOD value to 0 when you save the drawing.

DCTCUST SPELL
Type: String
Saved in: Registry
Initial value: “”
Displays the path and file name of the current custom spelling dictionary.

DCTMAIN SPELL
Type: String
Saved in: Registry
Initial value: “enu”
Displays the file name of the current main spelling dictionary. The full path is not shown because this file is expected to reside in the support directory. American English is “enu.”

DEFLPLSTYLE PLOT
(Read-only)
Type: String
Saved in: Registry
Initial value: “ByColor”
Specifies the default plot style for new layers.

DEFPLSTYLE PLOT
Type: String
Saved in: Registry
Initial value: “ByLayer”
Specifies the default plot style for new objects

DELOBJ *Modify commands*
Type: Integer
Saved in: Registry
Initial value: 1
Controls whether objects used to create other objects are retained or deleted from the drawing database.

0 Objects are retained
1 Objects are deleted

DEMANDLOAD TOOLS/OPTIONS
Type: Integer
Saved in: Registry
Initial Value: 3
Specifies if and when AutoCAD demand loads a third-party application if a drawing contains custom objects created in that application.

0 Turns off demand loading.
1 Demand loads the source application when you open a drawing that contains custom objects. This setting does not demand load the application when you invoke one of the application’s commands.
2 Demand loads the source application when you invoke one of the application’s commands. This setting does not demand load the application when you open a drawing that contains custom objects.
3 Demand loads the source application when you open a drawing that contains custom objects or when you invoke one of the application’s commands.

DIASTAT DD*****
(Read-only)
Type: Integer
Not saved
Stores the exit method of the most recently used dialog box.

0 Cancel
1 OK

DIMADEC DDIM
Type: Integer
Saved in: Drawing
Initial value: -1
Controls the number of precision places displayed in angular dimensions.

-1 Uses DIMDEC value
0–8 Indicates the number of decimal places to display in angular dimensions.

DIMALT DDIM
Type: Switch
Saved in: Drawing
Initial value: Off
Controls the display of alternate units in dimensions. See also DIMALTD, DIMALTF, DIMALTTD, DIMALTTZ, DIMALTZ, and DIMAPOST.

Off Disables alternate units
On Enables alternate units

DIMALTD DDIM
Type: Integer
Saved in: Drawing
Initial value: 2
Controls the number of decimal places in alternate units.

DIMALTF DDIM
Type: Real
Saved in: Drawing
Initial value: 25.4000
Controls the multiplier for alternate units. If DIMALT is turned on, DIMALTF multiplies linear dimensions by a factor to produce a value in an alternate system of measurement. The initial value represents the number of millimeters in an inch.

DIMALTRND DDIM
Type: Real
Saved in: Drawing
Initial value: 0.00
Rounds off the alternate dimension units.

DIMALTTD DDIM
Type: Integer
Saved in: Drawing
Initial value: 2
Sets the number of decimal places for the tolerance values in the alternate units of a dimension.

DIMALTTZ DDIM
Type: Integer

Saved in: Drawing
Initial value: 0
Toggles suppression of zeros in tolerance values.

0 Suppresses zero feet and precisely zero inches
1 Includes zero feet and precisely zero inches
2 Includes zero feet and suppresses zero inches
3 Includes zero inches and suppresses zero feet

To the preceding values, add:
4 Suppresses leading zeros
8 Suppresses trailing zeros

DIMALTU DDIM
Type: Integer
Saved in: Drawing
Initial value: 2
Sets the units format for alternate units of all dimension style family members except angular.

1 Scientific
2 Decimal
3 Engineering
4 Architectural (stacked)
5 Fractional (stacked)
6 Architectural
7 Fractional
8 Windows® Desktop (decimal format using Control Panel settings for decimal separator and number grouping symbols)

DIMALTZ DDIM
Type: Integer
Saved in: Drawing
Initial value: 0
Controls the suppression of zeros for alternate unit dimension values. DIMALTZ values 0–3 affect feet-and-inch dimensions only.

0 Suppresses zero feet and precisely zero inches
1 Includes zero feet and precisely zero inches
2 Includes zero feet and suppresses zero inches
3 Includes zero inches and suppresses zero feet
4 Suppresses leading zeros in decimal dimensions (for example, 0.5000 becomes .5000)
8 Suppresses trailing zeros in decimal dimensions (for example, 12.5000 becomes 12.5)
12 Suppresses both leading and trailing zeros (for example, 0.5000 becomes .5)

DIMAPOST DDIM
Type: String
Saved in: Drawing
Initial value: “”

Specifies a text prefix or suffix (or both) to the alternate dimension measurement for all types of dimensions except angular.
For instance, if the current units are Architectural, DIMALT is on, DIMALTF is 25.4 (the number of millimeters per inch), DIMALTD is 2, and DIMAPOST is set to “mm,” a distance of 10 units would be displayed as 10”[254.00mm].
To turn off an established prefix or suffix (or both), set it to a single period (.).

DIMASO DDIM
Type: Switch
Saved in: Drawing
Initial value: On
Controls the associativity of dimension objects. Also see DIMASSOC.

Off Creates no association between the various elements of the dimension. The lines, arcs, arrowheads, and text of a dimension are drawn as separate objects.
On Creates an association between the elements of the dimension. The elements are formed into a single object. If the definition point on the object moves, the dimension value is updated.

DIMASO is not stored in a dimension style.

DIMASSOC DDIM
(2002)
Type: Integer
Saved in: Drawing
Initial value: 2

This variable is complicated in that it functions (or doesn’t function) differently when opening pre-2002 drawings. When opening pre-2002 drawings, DIMASSOC is not utilized: AutoCAD uses the current setting of DIMASO in the drawing (as if DIMASSOC is set to 1 or 0 – DIMASO ON or OFF.) When opening a 2002 drawing in 2000i or earlier, the value of DIMASSOC is stored, and then restored when the drawing comes back in 2002.
DIMASSOC replaces the DIMASO variable used in earlier versions although it is backwards compatible.

0 There is no association between objects and their dimensions, and AutoCAD’s dimensioning commands do not fully recognize Model Space objects from Paper Space objects. Also, each dimension is a collection of individual objects – lines, arcs, text and so on.
1 There is no association between objects and their dimensions. However, the components of each dimension are interrelated and comprise a singe AutoCAD object. If you stretch a dimension line, for example, the dimension value changes to reflect the new dimension line’s length. While AutoCAD’s dimensioning commands do recognize Model Space objects in Paper Space, they do not compensate for viewport zoom levels. Thus, the dimension values for objects in a viewport having a zoom level of 0.5XP will be one half the correct value.
2 Objects are associated with their dimensions. Therefore, if you modify the length of a line or the radius of an arc, the dimension components and the dimension value automatically update to match the modification. Also, AutoCAD’s dimensioning commands fully recognize Model Space objects from Paper space and they automatically compensate for differences in viewport zoom levels to assign correct values to dimensions.

DIMASZ DDIM
Type: Real
Saved in: Drawing
Initial value: 0.1800
Controls the size of dimension line and leader line arrowheads. Also controls the size of hook lines. Multiples of the arrowhead size determine whether dimension lines and text should fit between the extension lines. DIMASZ is also used to scale arrowhead blocks if set by DIMBLK. DIMASZ has no effect when DIMTSZ is other than zero.

DIMATFIT DDIM
Type: Integer
Saved in: Drawing
Initial value: 3
Determines how dimension text and arrows are arranged when space is not sufficient to place both within the extension lines.

0 Places both text and arrows outside extension lines
1 Moves arrows first, then text
2 Moves text first, then arrows
3 Moves either text or arrows, whichever fits best

AutoCAD adds a leader to moved dimension text when DIMTMOVE is set to 1.

DIMAUNIT DDIM
Type: Integer
Saved in: Drawing
Initial value: 0
Sets the units format for angular dimensions.

0 Decimal degrees
1 Degrees/minutes/seconds
2 Grads
3 Radians
4 Surveyor’s units

DIMAUNIT sets this value when entered on the command line or when set from the Primary Units area in the Annotation dialog box.

DIMAZIN DDIM
Type: Integer
Saved in: Drawing
Initial value: 0
Suppresses zeros for angular dimensions.

0 Displays all leading and trailing zeros
1 Suppresses leading zeros in decimal dimensions (for example, 0.5000 becomes .5000)
2 Suppresses trailing zeros in decimal dimensions (for example, 12.5000 becomes 12.5)

3 Suppresses leading and trailing zeros (for example, 0.5000 becomes .5)

DIMBLK DDIM
Type: String
Saved in: Drawing
Initial value: “”
Sets the arrowhead block displayed at the ends of dimension lines or leader lines. To turn off arrowheads, enter a single period (.). Arrowhead block entries and the names used to select them in the New, Modify and Override Dimension Style dialog boxes are shown below. You can also enter the names of user-defined arrowhead blocks.

“”	closed filled
“_DOT”	dot
“_DOTSMALL”	dot small
“_DOTBLANK”	dot blank
“_ORIGIN”	origin indicator
“_ORIGIN2”	origin indicator 2
“_OPEN”	open
“_OPEN90”	right angle
“_OPEN30”	open 30
“_CLOSED”	closed
“_SMALL”	dot small blank
“_NONE”	none
“_OBLIQUE”	oblique
“_BOXFILLED”	box filled
“_BOXBLANK”	box
“_CLOSEDBLANK”	closed blank
“_DATUMFILLED”	datum triangle filled
“_DATUMBLANK”	datum triangle
“_INTEGRAL”	integral
“_ARCHTICK”	architectural tick

DIMBLK1 DDIM
Type: String
Saved in: Drawing
Initial value: “”
Sets the arrowhead for the first end of the dimension line when DIMSAH is on. To turn off arrowheads, enter a single period (.). For a list of arrowheads, see DIMBLK.

DIMBLK2 DDIM
Type: String
Saved in: Drawing
Initial value: “”
Sets the arrowhead for the second end of the dimension line when DIMSAH is on. To turn off arrowheads, enter a single period (.). For a list of arrowhead entries, see DIMBLK.

DIMCEN DDIM
Type: Real
Saved in: Drawing
Initial value: 0.0900

Controls drawing of circle or arc center marks and centerlines by the DIMCENTER, DIMDIAMETER, and DIMRADIUS commands. For DIMDIAMETER and DIMRADIUS, the center mark is drawn only if you place the dimension line outside the circle or arc.

0 No center marks or lines are drawn
<0 Centerlines are drawn
>0 Center marks are drawn

The absolute value specifies the size of the center mark.

DIMCLRD DDIM
Type: Integer
Saved in: Drawing
Initial value: 0
Assigns colors to dimension lines, arrowheads, and dimension leader lines. Also controls the color of leader lines created with the LEADER command. The color can be any valid color number. Integer equivalents for the BYBLOCK and BYLAYER properties are 0 and 256, respectively.

DIMCLRE DDIM
Type: Integer
Saved in: Drawing
Initial value: 0
Assigns colors to dimension extension lines. The color can be any valid color number. See DIMCLRD.

DIMCLRT DDIM
Type: Integer
Saved in: Drawing
Initial value: 0
Assigns colors to dimension text. The color can be any valid color number. See DIMCLRD.

DIMDEC DDIM
Type: Integer
Saved in: Drawing
Initial value: 4
Sets the number of decimal places displayed for the primary units of a dimension. The precision is based on the units or angle format you have selected.

DIMDLE DDIM
Type: Real
Saved in: Drawing
Initial value: 0.0000
Sets the distance the dimension line extends beyond the extension line when oblique strokes are drawn instead of arrowheads.

DIMDLI DDIM
Type: Real

Saved in: Drawing
Initial value: 0.3800
Controls the spacing of the dimension lines in baseline dimensions. Each dimension line is offset from the previous one by this amount, if necessary, to avoid drawing over it. Changes made with DIMDLI are not applied to existing dimensions.

DIMDSEP DDIM
Type: Single character
Saved in: Drawing
Initial value: Decimal point (“.”)
Specifies a single-character decimal separator to use when creating dimensions whose unit format is decimal.
When prompted, enter a single character at the command line. If dimension units is set to Decimal, the DIMDSEP character is used instead of the default decimal point. If DIMDSEP is set to NULL (default value, reset by entering a period), AutoCAD uses the decimal point as the dimension separator.

DIMEXE DDIM
Type: Real
Saved in: Drawing
Initial value: 0.1800
Specifies how far to extend the extension line beyond the dimension line.

DIMEXO DDIM
Type: Real
Saved in: Drawing
Initial value: 0.0625
Specifies how far extension lines are offset from origin points. If you point directly at the corners of an object to be dimensioned, the extension lines do not touch the object.

DIMFIT DDIM
Type: Integer
Saved in: Drawing
Initial value: 3
Obsolete. Has no effect in AutoCAD 2000 except to preserve the integrity of pre-AutoCAD 2000 scripts and AutoLISP routines. In AutoCAD 2000, DIMFIT is replaced by DIMATFIT and DIMTMOVE. Displays at the command prompt.

0 Both text and arrowheads are placed on the outside extension lines
1 Text is placed between the extension lines and arrowheads are placed on the outside
2 Arrowheads are placed between the extension lines and text is on the outside
3 Text is placed between the extension lines and places the arrowheads on the outside
4 Same as 3 except leaders also connect the text to the dimension line unlike no. 3
5 Both arrowheads and text are placed on the inside of the extension lines, unless not enough space is between the extension lines

DIMFRAC DDIM

Type: Integer
Saved in: Drawing
Initial value: 0
Sets the fraction format when DIMLUNIT is set to 4 (Architectural) or 5 (Fractional).

0 Horizontal
1 Diagonal
2 Not stacked (for example, ½)

DIMGAP DDIM
Type: Real
Saved in: Drawing
Initial value: 0.0900
Sets the distance around the dimension text when the dimension line breaks to accommodate dimension text. Also sets the gap between annotation and a hook line created with the LEADER command. If you enter a negative DIMGAP value, AutoCAD places a box around the dimension text.
AutoCAD also uses DIMGAP as the minimum length for pieces of the dimension line. When calculating the default position for the dimension text, it positions the text inside the extension lines only if doing so breaks the dimension lines into two segments at least as long as DIMGAP. Text placed above or below the dimension line is moved inside only if there is room for the arrowheads, dimension text, and a margin between them at least as large as DIMGAP: 2 * (DIMASZ + DIMGAP).
DIMGAP also sets the gap between a tolerance symbol and its feature control frame.

DIMJUST DDIM
Type: Integer
Saved in: Drawing
Initial value: 0
Controls the horizontal positioning of dimension text.

0 Centered along the dimension line between the extension lines
1 Next to the first extension line
2 Next to the second extension line
3 Above and aligned with the first extension line
4 Above and aligned with the second extension line

DIMLDRBLK DDIM
Type: String
Saved in: Drawing
Initial value: “”
Specifies the arrow type for leaders. To turn off arrowhead display, enter a single period (.). For a list of arrowhead entries, see DIMBLK.

DIMLFAC DDIM
Type: Real
Saved in: Drawing
Initial value: 1.0000

Sets a scale factor for linear dimension measurements. All linear dimension distances, including radii, diameters, and coordinates, are multiplied by DIMLFAC before being converted to dimension text.
DIMLFAC has no effect on angular dimensions, and is not applied to the values held in DIMRND, DIMTM, or DIMTP.
If you create a dimension in paper space and DIMLFAC is not set to zero, AutoCAD multiplies the distance measured by the absolute value of DIMLFAC. In model space, negative values for DIMLFAC are ignored, and the value 1.0 is used instead.
AutoCAD computes a value for DIMLFAC if you try to change DIMLFAC from the Dim prompt while in paper space and you select the Viewport option.
Dim: dimlfac
Current value <1.0000> New value (Viewport): v
Select viewport to set scale:
AutoCAD calculates the scaling of model space to paper space and assigns the negative of this value to DIMLFAC.

DIMLIM DDIM
Type: Switch
Saved in: Drawing
Initial value: Off
Generates dimension limits as the default text. Setting DIMLIM to on turns DIMTOL off.

Off Dimension limits are not generated as default text
On Dimension limits are generated as default text

DIMLUNIT DDIM
Type: Integer
Saved in: Drawing
Initial value: 2
Sets units for all dimension types except Angular.

1 Scientific
2 Decimal
3 Engineering
4 Architectural
5 Fractional
6 Windows desktop

DIMLWD DDIM
Type: Enum
Saved in: Drawing
Initial value: -2
Assigns lineweight to dimension lines. Values are standard lineweight (BYLAYER, BYBLOCK, integer representing 100^{th} of mm).

-1 Sets the lineweight to “ByLayer.”
-2 Sets the lineweight to “ByBlock.”
-3 Sets the lineweight to “Default.” “Default” is controlled by the LWDEFAULT system variable.

Other valid values entered in millimeters include 0, 5, 9, 13, 15, 18, 20, 25, 30, 35, 40, 50, 53, 60, 70, 80, 90, 100, 106, 120, 140, 158, 200, and 211.
All values must be entered in millimeters. (Multiply a value by 25.4 to convert values from inches to millimeters.)

DIMLWE DDIM
Type: Enum
Saved in: Drawing
Initial value: -2
Assigns lineweight to extension lines. Values are standard lineweight (BYLAYER, BYBLOCK, integer representing 100th of mm).

-1 Sets the lineweight to “ByLayer.”
-2 Sets the lineweight to “ByBlock.”
-3 Sets the lineweight to “Default.” “Default” is controlled by the LWDEFAULT system variable.

Other valid values entered in millimeters include 0, 5, 9, 13, 15, 18, 20, 25, 30, 35, 40, 50, 53, 60, 70, 80, 90, 100, 106, 120, 140, 158, 200, and 211.
All values must be entered in millimeters. (Multiply a value by 25.4 to convert values from inches to millimeters.)

DIMPOST DDIM
Type: String
Saved in: Drawing
Initial value: “”
Specifies a text prefix or suffix (or both) to the dimension measurement. For example, to establish a suffix for millimeters, set DIMPOST to mm; a distance of 19.2 units would be displayed as 19.2 mm.
If tolerances are turned on, the suffix is applied to the tolerances as well as to the main dimension.
Use <> to indicate placement of the text in relation to the dimension value. For example, enter <>mm to display a 5.0 millimeter radial dimension as “5.0mm.” If you entered mm <>, the dimension would be displayed as “mm 5.0.” Use the <> mechanism for angular dimensions.

DIMRND DDIM
Type: Real
Saved in: Drawing
Initial value: 0.0000
Rounds all dimensioning distances to the specified value. For instance, if DIMRND is set to 0.25, all distances round to the nearest 0.25 unit. If you set DIMRND to 1.0, all distances round to the nearest integer. Note that the number of digits edited after the decimal point depends on the precision set by DIMDEC. DIMRND does not apply to angular dimensions.

DIMSAH DDIM
Type: Switch
Saved in: Drawing
Initial value: Off
Controls the display of dimension line arrowhead blocks.

Off Use arrowhead blocks set by DIMBLK
On Use arrowhead blocks set by DIMBLK1 and DIMBLK2

DIMSCALE DDIM
Type: Real
Saved in: Drawing
Initial value: 1.0000
Sets the overall scale factor applied to dimensioning variables that specify sizes, distances, or offsets. Also affects the scale of leader objects created with the LEADER command.

0.0 AutoCAD computes a reasonable default value based on the scaling between the current model space viewport and paper space. If you are in paper space or model space and not using the paper space feature, the scale factor is 1.0.
>0 AutoCAD computes a scale factor that leads text sizes, arrowhead sizes, and other scaled distances to plot at their face values.

DIMSCALE does not affect tolerances or measured lengths, coordinates, or angles.

DIMSD1 DDIM
Type: Switch
Saved in: Drawing
Initial value: Off
Controls suppression of the first dimension line.

When turned on, suppresses the display of the dimension line and arrowhead between the first extension line and the text.

DIMSD2 DDIM
Type: Switch
Saved in: Drawing
Initial value: Off
Controls suppression of the second dimension line.

When turned on, suppresses the display of the dimension line and arrowhead between the second extension line and the text.

DIMSE1 DDIM
Type: Switch
Saved in: Drawing
Initial value: Off
Suppresses display of the first extension line.

Off Extension line is not suppressed
On Extension line is suppressed

DIMSE2 DDIM
Type: Switch
Saved in: Drawing

Initial value: Off
Suppresses display of the second extension line.

Off Extension line is not suppressed
On Extension line is suppressed

DIMSHO DDIM
Type: Switch
Saved in: Drawing
Initial value: On
Controls redefinition of dimension objects while dragging. Associative dimensions recompute dynamically as they are dragged. On some computers, dynamic dragging can be very slow, so, set DIMSHO to off to drag the original image instead. DIMSHO is not stored in a dimension style.

Off Update off
On Update on

DIMSOXD DDIM
Type: Switch
Saved in: Drawing
Initial value: Off
Suppresses drawing of dimension lines outside the extension lines.

Off Dimension lines are not suppressed
On Dimension lines are suppressed

If the dimension lines would be outside the extension lines and DIMTIX is on, setting DIMSOXD to on suppresses the dimension line. If DIMTIX is off, DIMSOXD has no effect.

DIMSTYLE DDIM
(Read-only)
Type: String
Saved in: Drawing
Initial value: “STANDARD”
DIMSTYLE is both a command and a system variable. The DIMSTYLE system variable shows the current dimension style. To display the DIMSTYLE system variable, use the SETVAR command. The DIMSTYLE system variable is read-only; you cannot change its value on the command line. To change the current dimension style, use the DIMSTYLE command.

DIMTAD DDIM
Type: Integer
Saved in: Drawing
Initial value: 0
Controls the vertical position of text in relation to the dimension line.

0 Centers the dimension text between the extension lines.
1 Places the dimension text above the dimension line except when the dimension line is not horizontal and text inside the extension lines is forced horizontal (DIMTIH = 1).

The distance from the dimension line to the baseline of the lowest line of text is the current DIMGAP value.

2 Places the dimension text on the side of the dimension line farthest away from the defining points.

3 Places the dimension text to conform to Japanese Industrial Standards (JIS).

DIMTDEC DDIM
Type: Integer
Saved in: Drawing
Initial value: 4
Sets the number of decimal places to display in tolerance values for the primary units in a dimension.

DIMTFAC DDIM
Type: Real
Saved in: Drawing
Initial value: 1.0000
Sets a scale factor used to calculate the height of text for dimension fractions and tolerances. AutoCAD multiplies DIMTXT by DIMTFAC to set the fractional or tolerance text height.

DIMTIH DDIM
Type: Switch
Saved in: Drawing
Initial value: On
Controls the position of dimension text inside the extension lines for all dimension types except ordinate.

Off Aligns text with the dimension line

On Draws text horizontally

DIMTIX DDIM
Type: Switch
Saved in: Drawing
Initial value: Off
Draws text between extension lines.

Off Varies with the type of dimension. For linear and angular dimensions, AutoCAD places text inside the extension lines if there is sufficient room. For radius and diameter dimensions that don't fit inside the circle or arc, DIMTIX has no effect and always forces the text outside the circle or arc.

On Draws dimension text between the extension lines even if AutoCAD ordinarily places it outside those lines.

DIMTM DDIM
Type: Real
Saved in: Drawing
Initial value: 0.0000

Sets the minimum (or lower) tolerance limit for dimension text when DIMTOL or DIMLIM is on. AutoCAD accepts signed values for DIMTM. If DIMTOL is on and DIMTP and DIMTM are set to the same value, AutoCAD draws a tolerance value.
If DIMTM and DIMTP values differ, the upper tolerance is drawn above the lower, and a plus sign is added to the DIMTP value if it is positive.
For DIMTM, AutoCAD uses the negative of the value you enter (adding a minus sign if you specify a positive number and a plus sign if you specify a negative number).

DIMTMOVE DDIM
Type: Integer
Saved in: Drawing
Initial value: 0
Sets dimension text movement rules.

0 Moves the dimension line with dimension text
1 Adds a leader when dimension text is moved
2 Allows text to be moved freely without a leader

DIMTOFL DDIM
Type: Switch
Saved in: Drawing
Initial value: Off
Controls whether a dimension line is drawn between the extension lines even when the text is placed outside. For radius and diameter dimensions (when DIMTIX is off), draws a dimension line inside the circle or arc and places the text, arrowheads, and leader outside.

Off Does not draw dimension lines between the measured points when arrowheads are placed outside the measured points
On Draws dimension lines between the measured points even when arrowheads are placed outside the measured points

DIMTOH DDIM
Type: Switch
Saved in: Drawing
Initial value: On
Controls the position of dimension text outside the extension lines.

Off Aligns text with the dimension line
On Draws text horizontally

DIMTOL DDIM
Type: Switch
Saved in: Drawing
Initial value: Off
Appends tolerances to dimension text. Setting DIMTOL to on turns DIMLIM off.

Off No tolerance is added or used
On Tolerance is added

DIMTOLJ DDIM
Type: Integer
Saved in: Drawing
Initial value: 1
Sets the vertical justification for tolerance values relative to the nominal dimension text.

0 Bottom
1 Middle
2 Top

DIMTP DDIM
Type: Real
Saved in: Drawing
Initial value: 0.0000
Sets the maximum (or upper) tolerance limit for dimension text when DIMTOL or DIMLIM is on. AutoCAD accepts signed values for DIMTP. If DIMTOL is on and DIMTP and DIMTM are set to the same value, AutoCAD draws a tolerance value.
If DIMTM and DIMTP values differ, the upper tolerance is drawn above the lower and a plus sign is added to the DIMTP value if it is positive.

DIMTSZ DDIM
Type: Real
Saved in: Drawing
Initial value: 0.0000
Specifies the size of oblique strokes drawn instead of arrowheads for linear, radius, and diameter dimensioning.

0 Draws arrowheads.
>0 Draws oblique strokes instead of arrowheads. The size of the oblique strokes is determined by this value multiplied by the DIMSCALE value.

DIMTVP DDIM
Type: Real
Saved in: Drawing
Initial value: 0.0000
Controls the vertical position of dimension text above or below the dimension line. AutoCAD uses the DIMTVP value when DIMTAD is off. The magnitude of the vertical offset of text is the product of the text height and DIMTVP. Setting DIMTVP to 1.0 is equivalent to setting DIMTAD to on. AutoCAD splits the dimension line to accommodate the text only if the absolute value of DIMTVP is less than 0.7.

DIMTXSTY DDIM
Type: String
Saved in: Drawing
Initial value: “STANDARD”
Specifies the text style of the dimension.

DIMTXT DDIM
Type: Real

Saved in: Drawing
Initial value: 0.1800
Specifies the height of dimension text, unless the current text style has a fixed height.

DIMTZIN DDIM
Type: Integer
Saved in: Drawing
Initial value: 0
Controls the suppression of zeros in tolerance values. DIMTZIN stores this value when you enter it on the command line or set it under Primary Units in the Annotation dialog box. DIMTZIN values 0–3 affect feet-and-inch dimensions only.

0 Suppresses zero feet and precisely zero inches
1 Includes zero feet and precisely zero inches
2 Includes zero feet and suppresses zero inches
3 Includes zero inches and suppresses zero feet
4 Suppresses leading zeros in decimal dimensions (for example, 0.5000 becomes .5000)
8 Suppresses trailing zeros in decimal dimensions (for example, 12.5000 becomes 12.5)
12 Suppresses both leading and trailing zeros (for example, 0.5000 becomes .5)

DIMUNIT DDIM
Type: Integer
Saved in: Drawing
Initial value: 2
Obsolete. Has no effect in AutoCAD 2000 except to preserve the integrity of pre-AutoCAD 2000 scripts and AutoLISP routines. In AutoCAD 2000, DIMUNIT is replaced by DIMLUNIT and DIMFRAC. Displays on the command line.

DIMUPT DDIM
Type: Switch
Saved in: Drawing
Initial value: Off
Controls options for user-positioned text.

Off Cursor controls only the dimension line location; text goes to it's default location
On Cursor controls both the text position and the dimension line location

DIMZIN DDIM
Type: Integer
Saved in: Drawing
Initial value: 0
Controls the suppression of zeros in the primary unit value. DIMZIN stores this value when you enter it on the command line or set it under Primary Units in the Annotation dialog box. DIMZIN values 0–3 affect feet-and-inch dimensions only.

0 Suppresses zero feet and precisely zero inches
1 Includes zero feet and precisely zero inches
2 Includes zero feet and suppresses zero inches

3 Includes zero inches and suppresses zero feet
4 Suppresses leading zeros in decimal dimensions (for example, 0.5000 becomes .5000)
8 Suppresses trailing zeros in decimal dimensions (for example, 12.5000 becomes 12.5)
12 Suppresses both leading and trailing zeros (for example, 0.5000 becomes .5)

DIMZIN also affects real-to-string conversions performed by the AutoLISP rtos and angtos functions.

DISPSILH TOOLS/OPTIONS
Type: Integer
Saved in: Drawing
Initial value: 0
Controls display of silhouette curves of solid objects in Wireframe mode. Also controls whether mesh is drawn (0) or suppressed (1) when a solid object is hidden.

0 Off
1 On

DISTANCE DISTANCE
(Read-only)
Type: Real
Not saved
Stores the distance computed by the DIST command. Displays on the command line.

DONUTID DONUT
Type: Real
Not saved
Initial value: 0.5000
Sets the default for the inside diameter of a donut.

DONUTOD DONUT
Type: Real
Not saved
Initial value: 1.0000
Sets the default for the outside diameter of a donut. The value must be nonzero. If DONUTID is larger than DONUTOD, the two values are swapped by the next command.

DRAGMODE TOOLS/OPTIONS
Type: Integer
Saved in: Registry
Initial value: 2
Controls the display of objects being dragged.

0 Does not display an outline of the object as you drag it
1 Displays the outline of the object as you drag it only if you enter drag on the command line after selecting the object to drag
2 Auto; always displays an outline of the object as you drag it

DRAGP1 TOOLS/OPTIONS
Type: Integer
Saved in: Registry
Initial value: 10
Sets the regen-drag input sampling rate.

DRAGP2 TOOLS/OPTIONS
Type: Integer
Saved in: Registry
Initial value: 25
Sets the fast-drag input sampling rate.

DWGCHECK *Environment*
Type: Integer
Saved in: Registry
Initial Value: 0
Determines whether a drawing was last edited by a product other than AutoCAD. At zero, the dialog box display is suppressed. Changing the value to 1 will display the dialog box, if warranted.

DWGCODEPAGE *Environment*
(Read-only)
Type: String
Saved in: Drawing
Stores the same value as SYSCODEPAGE (for compatibility reasons).

DWGNAME *Environment*
(Read-only)
Type: String
Not saved
Initial value: “Drawing1.dwg”
Stores the drawing name as entered by the user. If the drawing has not been named yet, DWGNAME defaults to “Drawing1.dwg.” If the user specified a drive/directory prefix, the prefix is stored in DWGPREFIX.

DWGPREFIX *Environment*
(Read-only)
Type: String
Not saved
Stores the drive/directory prefix for the drawing.

DWGTITLED *Environment*
(Read-only)
Type: Integer
Not saved
Initial value: 0

Indicates whether the current drawing has been named.

0 Drawing has not been named
1 Drawing has been named

EDGEMODE TRIM
Type: Integer EXTEND
Saved in: Registry
Initial value: 0
Controls how the TRIM and EXTEND commands determine cutting and boundary edges.

0 Uses the selected edge without an extension
1 Extends or trims the selected object to an imaginary extension of the cutting or boundary edge

Lines, arcs, elliptical arcs, rays, and polylines are objects eligible for natural extension. The natural extension of a line or ray is an unbounded line (xline), an arc is a circle, and an elliptical arc is an ellipse. A polyline is broken down into its line and arc components, which are extended to their natural boundaries.

ELEVATION *Drawing and modify commands*
Type: Real
Saved in: Drawing (viewport specific)
Initial value: 0.0000
Stores the current elevation relative to the current UCS for the current viewport in the current space.

EXPERT *Environment*
Type: Integer
Not saved
Initial value: 0
Controls whether certain prompts are issued.

0 Issues all prompts normally.
1 Suppresses “About to regen, proceed?” and “Really want to turn the current layer off?”
2 Suppresses the preceding prompts and “Block already defined. Redefine it?” (BLOCK) and “A drawing with this name already exists. Overwrite it?” (SAVE or WBLOCK).
3 Suppresses the preceding prompts and those issued by the LINETYPE command if you try to load a linetype that’s already loaded or create a new linetype in a file that already defines that linetype.
4 Suppresses the preceding prompts and those issued by UCS Save and VPORTS Save if the name you supply already exists.
5 Suppresses the prompt, “That name is already in Use, redefine it?” issued by the -DIMSTYLE Save option when you supply the name of an existing dimension style.

When a prompt is suppressed by EXPERT, the operation in question is performed as though you entered (Y) at the prompt. Setting EXPERT can affect scripts, menu macros, AutoLISP, and the command functions.

EXPLMODE EXPLODE
Type: Integer
Not saved
Initial value: 1
Controls whether the EXPLODE command supports nonuniformly scaled (NUS) blocks.

0 Does not explode NUS blocks
1 Explodes NUS blocks

EXTMAX *Display*
(Read-only)
Type: 3D Point
Saved in: Drawing
Stores the upper-right point of the drawing extents. Expands outward as new objects are drawn; shrinks only with ZOOM All or ZOOM Extents. Reported in world coordinates for the current space.

EXTMIN *Display*
(Read-only)
Type: 3D Point
Saved in: Drawing
Stores the lower-left point of the drawing extents. Expands outward as new objects are drawn; shrinks only with ZOOM All or ZOOM Extents. Reported in world coordinates for the current space.

EXTNAMES *Drawing database*
Type: Integer
Saved in: Drawing
Initial value: 1
Sets the parameters for named object names (such as linetypes and layers) stored in symbol tables.

0 Uses Release 14 parameters, which limit names to 31 characters in length. Names can include the letters A to Z, the numerals 0 to 9, and the special characters, dollar sign ($), underscore (_), and hyphen (-).
1 Uses AutoCAD 2000 parameters. Names can be up to 255 characters in length, and can include the letters A to Z, the numerals 0 to 9, spaces, and any special characters not used by Microsoft Windows and AutoCAD for other purposes.

FACETRATIO *3D/Display*
Type: Integer
Not saved
Initial Value: 0
Controls the aspect ratio of faceting for cylindrical and conic ACIS solids. A setting of 1 increases the density of the mesh to improve the quality of rendered and shaded models.
0 Creates an N by 1 mesh for cylindrical and conic ACIS solids
1 Creates an N by M mesh for cylindrical and conic ACIS solids

FACETRES *3D/Display*
Type: Real
Saved in: Drawing
Initial value: 0.5
Adjusts the smoothness of shaded and rendered objects and objects with hidden lines removed. Valid values are from 0.01 to 10.0 (smoothest). FACETRES controls the smoothness of shaded and rendered curved solids. It is linked to the value set by VIEWRES (a command, not a variable): when FACETRES is set to 1, there is a one-to-one correspondence between the viewing resolution of circles, arcs, and ellipses and the tessellation of solid objects. For example, when FACETRES is set to 2, the tessellation will be twice the tessellation set by VIEWRES. VIEWRES can be set from 1 to 20000 (the smoothest).
When you raise and lower the value of VIEWRES, objects controlled by both VIEWRES and FACETRES are affected. When you raise and lower the value of FACETRES, only solid objects are affected.

FILEDIA *Environment*
Type: Integer
Saved in: Register
Initial value: 1
Controls the display of dialog boxes that read and write files. If you prefer to enter file names on the command line, set FILEDIA to off (0). If FILEDIA is set to off and you want to use the dialog box, enter a ~ (tilde) at the file name prompt.
FILEDIA controls the display of file listing dialog boxes. Set FILEDIA to 1 to enable these dialog boxes.

0 Command line
1 Dialog Box

FILLETRAD FILLET
Type: Real
Saved in: Drawing
Initial value: 0.5000
Stores the current fillet radius.

FILLMODE DDRMODES
Type: Integer
Saved in: Drawing
Initial value: 1
Specifies whether multilines, traces, solids, all hatches (including solid-fill), and wide polylines are filled in.

0 Objects are not filled
1 Objects are filled

FONTALT TOOLS/OPTIONS
Type: String
Saved in: Registry
Initial value: “simplex.shx”

Specifies the alternate font to be used when the specified font file cannot be located. If an alternate font is not specified, AutoCAD displays the Alternate Font dialog box. The dialog box is displayed in the following cases:

1. A Release 13 drawing is opened; FONTALT is not set or not found; and a TrueType®, SHX, or PostScript font is not found for a defined text style.
2. A Release 14 drawing is opened, FONTALT is not set or not found, and an SHX or PostScript font is not found for a defined text style. For missing TrueType fonts in Release 14 drawings, AutoCAD automatically substitutes the closest TrueType font available.
3. The Browse button is pressed in the Options dialog box when you specify an alternate font.

AutoCAD validates the alternate font specified for FONTALT. If the font name or font file name is not found, the message “Font not found” is displayed. Enter either a TrueType font name (for example, Times New Roman Bold) or a TrueType file name (for example timebd.ttf). When a TrueType file name is entered for FONTALT, AutoCAD returns the font name in place of the file name if the font is registered with the operating system.

FONTMAP TOOLS/OPTIONS
Type: String
Saved in: Registry
Initial value: “acad.fmp”
Specifies the font mapping file to be used. A font mapping file contains one font mapping per line; the original font used in the drawing and the font to be substituted for it are separated by a semicolon (;). For example, to substitute the Times TrueType font for the Roman font, the line in the mapping file would read as follows:
romanc.shx;times.ttf
If FONTMAP does not point to a font mapping file, if the FMP file is not found, or if the font file name specified in the FMP file is not found, AutoCAD uses the font defined in the style. If the font in the style is not found, AutoCAD substitutes the font according to substitution rules.

FRONTZ DDVIEW
(Read-only)
Type: Real
Saved in: Drawing
Stores the front clipping plane offset from the target plane for the current viewport, in drawing units. Meaningful only if the front clipping bit in VIEWMODE is on and the front-clip-not-at-eye bit is also on. The distance of the front clipping plane from the camera point is found by subtracting FRONTZ from the camera-to-target distance.

FULLOPEN OPEN
(Read-only) PARTIALOAD
Type: Integer PARTIALOPEN
Not saved
Initial value: 1
Indicates whether the current drawing is partially open.

0 Indicates a partially open drawing
1 Indicates a fully open drawing

GRIDMODE DDRMODES
Type: Integer
Saved in: Drawing
Initial value: 0
Specifies whether the grid is turned on or off.

0 Turns the grid off
1 Turns the grid on

GRIDUNIT DDRMODES
Type: 2D point
Saved in: Drawing
Initial value: 0.5000,0.5000
Specifies the grid spacing (X and Y) for the current viewport.

GRIPBLOCK DDGRIPS
Type: Integer
Saved in: Registry
Initial value: 1
Controls the assignment of grips in blocks.

0 Assigns a grip only to the insertion point of the block
1 Assigns grips to objects within the block

GRIPCOLOR DDGRIPS
Type: Integer
Saved in: Registry
Initial value: 5
Controls the color of nonselected grips (drawn as box outlines). The valid range is 1 to 255.

1 Minimum (Red)
5 Default (Blue)
255 Maximum

GRIPHOT DDGRIPS
Type: Integer
Saved in: Registry
Initial value: 1
Controls the color of selected grips (drawn as filled boxes). The valid range is 1 to 255.

1 Default (Red)
255 Maximum

GRIPS DDGRIPS
Type: Integer
Saved in: Registry
Initial value: 1

Controls the use of selection set grips for the Stretch, Move, Rotate, Scale, and Mirror Grip modes.

0 Turns off grips
1 Turns on grips

To adjust the size of the grips and the effective selection area used by the cursor when you snap to a grip, use GRIPSIZE.

GRIPSIZE DDGRIPS
Type: Integer
Saved in: Registry
Initial value: 10
Sets the size of the grip box in pixels. The valid range is 1 to 255.

HALOGAP HIDE
(2002) SHADEMODE
Type: Real
Not saved
Specifies the distance to shorten a displayed line.

HANDLES *Environment*
(Read-only)
Type: Integer
Saved in: Drawing
Initial value: On
Reports whether object handles can be accessed by applications.

0 Off
1 On

HIDEPRECISION *Environment*
(undocumented)
Type: Integer
Not saved
Initial Value: 0
Controls the accuracy of hides and shades. Hides can be calculated in double precision or single precision. Setting HIDEPRECISION to 1 produces more accurate hides by using double precision, but this setting also uses more memory and can affect performance, especially when hiding solids. Useful when nothing else seems to work.

0 Single precision; uses less memory
1 Double precision; uses more memory

HIDETEXT HIDE
(2002)
Type: Switch
Saved in: Drawing
Initial value: On

The setting of this variable determines whether text objects created with the TEXT, DTEXT or MTEXT commands are processed during a HIDE. With pre-2002 text objects, it acts as if HIDETEXT was set to Off.

Off Disables Hide processing of text objects. Text is not hidden and does not hide other objects unless the text has a thickness assigned to it.
On Enables Hide processing of text objects. Text is hidden and does hide objects.

HIGHLIGHT DDRMODES
Type: Integer
Not saved
Initial value: 1
Controls object highlighting; does not affect objects selected with grips.

0 Off
1 On

HPANG HATCH
Type: Real
Not saved
Initial value: 0
Specifies the hatch pattern angle.

HPBOUND HATCH
Type: Integer
Not saved
Initial value: 1
Controls the object type created by the BHATCH and BOUNDARY commands.

0 Creates a region
1 Creates a polyline

HPDOUBLE HATCH
Type: Integer
Not saved
Initial value: 0
Specifies hatch pattern doubling for user-defined patterns.

0 Turns off hatch pattern doubling
1 Turns on hatch pattern doubling

HPNAME HATCH
Type: String
Not saved
Initial value: “ANSI31”
Sets a default hatch pattern name of up to 34 characters without spaces. Returns “” if there is no default. Enter a period (.) to set no default.

HPSCALE HATCH
Type: Real
Not saved
Initial value: 1.0000
Specifies the hatch pattern scale factor, which must be nonzero.

HPSPACE HATCH
Type: Real
Not saved
Initial value: 1.0000
Specifies the hatch pattern line spacing for user-defined simple patterns, which must be nonzero.

HYPERLINKBASE HYPERLINK
Type: String
Saved in: Drawing
Initial value: “”
Specifies the path used for all relative hyperlinks in the drawing. If no value is specifed, the drawing path is used for all relative hyperlinks.

IMAGEHLT IMAGE
Type: Integer
Saved in: Registry
Initial value: 0
Controls whether the entire raster image or only the raster image frame is highlighted.

0 Highlights only the raster image frame
1 Highlights the entire raster image

INDEXCTL *Environment*
Type: Integer
Saved in: Drawing
Initial value: 0
Controls whether layer and spatial indexes are created and saved in drawing files.

0 No indexes are created
1 Layer index is created
2 Spatial index is created
3 Layer and spatial indexes are created

INETLOCATION BROWSER
Type: String
Saved in: Registry
Initial value: “www.autodesk.com/acaduser”
Stores the Internet location used by the BROWSER command and the Browse the Web dialog box.

INSBASE INSERT

Type: 3D point
Saved in: Drawing
Initial value: 0.0000,0.0000,0.0000
Stores the insertion base point set by BASE, which gets expressed as a UCS coordinate for the current space.

INSNAME INSERT
Type: String
Not saved
Initial value: “”
Sets a default block name for the INSERT command. The name must conform to symbol naming conventions. Returns “” if no default is set. Enter a period (.) to set no default.

INSUNITS *AutoCAD Design Center (ADC)*
Type: Integer
Saved in: Drawing
Initial value: 0
When you drag a block or image from AutoCAD Design Center, specifies a drawing units value as follows:

0 Unspecified (No units)
1 Inches
2 Feet
3 Miles
4 Millimeters
5 Centimeters
6 Meters
7 Kilometers
8 Microinches
9 Mils
10 Yards
11 Angstroms
12 Nanometers
13 Microns
14 Decimeters
15 Decameters
16 Hectometers
17 Gigameters
18 Astronomical Units
19 Light Years
20 Parsecs

INSUNITSDEFSOURCE *AutoCAD Design Center (ADC)*
Type: Integer
Saved in: Registry
Initial value: 0
Sets source content units value. Valid range is 0 to 20.

INSUNITSDEFTARGET *AutoCAD Design Center (ADC)*
Type: Integer

Saved in: Registry
Initial value: 0
Sets target drawing units value. Valid range is 0 to 20.

ISAVEBAK TOOLS/OPTIONS
Type: Integer
Saved in: Registry
Initial value: 1
Improves the speed of incremental saves, especially for large drawings. ISAVEBAK controls the creation of a backup file (BAK). In Windows, copying the file data to create a BAK file for large drawings takes a major portion of the incremental save time.
0 No BAK file is created (even for a full save)
1 A BAK file is created

WARNING: In some cases (such as a power failure in the middle of a save), it's possible that drawing data can be lost.

ISAVEPERCENT TOOLS/OPTIONS
Type: Integer
Saved in: Registry
Initial value: 50
Determines the amount of wasted space tolerated in a drawing file. The value of ISAVEPERCENT is an integer between 0 and 100. The default value of 50 means that the estimate of wasted space within the file does not exceed 50 percent of the total file size. Wasted space is eliminated by periodic full saves. When the estimate exceeds 50 percent, the next save will be a full save. This resets the wasted space estimate to 0. If ISAVEPERCENT is set to 0, every save is a full save.

0 Minimum
100 Maximum

ISOLINES *Display*
Type: Integer
Saved in: Drawing
Initial value: 4
Specifies the number of isolines per surface on objects. Valid integer values are from 0 to 2047.

0 Minimum
4 Default
2047 Maximum

LASTANGLE ARC
(Read-only)
Type: Real
Not saved
Stores the end angle of the last arc entered relative to the XY plane of the current UCS for the current space.

LASTPOINT *Most commands*
Type: 3D point
Not saved
Initial value: 0.0000,0.0000,0.0000
Stores the last point entered, expressed as a UCS coordinate for the current space; referenced by the at symbol (@) during keyboard entry.

LASTPROMPT *Most commands*
(Read-only)
Type: String
Not saved
Initial value: “”
Stores the last string echoed to the command line. This string is identical to the last line seen at the command line and includes any user input.

LAYERPMODE
Type: Switch
Saved in: Drawing
Initial value: On
Controls the tracking of layer changes (used by the LAYERP command.)

On Tracks changes in layer states (on, off, frozen, locked)
Off Does not track changes in layer states or stops tracking if switched from on to off

LAYOUTREGENCTL LAYOUT
(2002)
Type: Integer
Saved in: Drawing
Initial value: 2

Controls the display behavior of how a layout acts when switching to it.

0 Each time you switch to a layout tab a regeneration occurs
1 Model and the last layout tab is chached into memory
2 Initial display of the tab is the only time that a regeration occurs and any other time that the tab is activated it is read from cache

LENSLENGTH DVIEW
(Read-only)
Type: Real
Saved in: Drawing
Stores the length of the lens (in millimeters) used in perspective viewing for the current viewport.

LIMCHECK *Most commands*
Type: Integer
Saved in: Drawing
Initial value: 0
Controls the creation of objects outside the drawing limits.

0 Objects can be created outside the limits
1 Objects cannot be created outside the limits

LIMMAX LIMITS
Type: 2D point
Saved in: Drawing
Initial value: 12.0000,9.0000
Stores the upper-right drawing limits for the current space, expressed as a world coordinate. LIMMAX is read-only when paper space is active and the paper background or paper margins are displayed.

LIMMIN LIMITS
Type: 2D point
Saved in: Drawing
Initial value: 0.0000,0.0000
Stores the lower-left drawing limits for the current space, expressed as a world coordinate. LIMMIN is read-only when paper space is active and the paper background or paper margins are displayed.

LISPINIT TOOLS/OPTIONS
Type: Integer
Saved in: Registry
Initial value: 1
When single-document interface is enabled, specifies whether AutoLISP-defined functions and variables are preserved when you open a new drawing or whether they are valid in the current drawing session only.

0 AutoLISP functions and variables are preserved from drawing to drawing
1 AutoLISP functions and variables are valid in the current drawing only

LOCALE *Environment*
(Read-only)
Type: String
Not saved
Initial value: “enu”
Displays the International Standards Organization (ISO) language code of the current AutoCAD version you’re running.

LOGFILEMODE *Environment*
Type: Integer
Saved in: Registry
Initial value: 0
Specifies whether the contents of the text window are written to a log file.

0 Log file is not maintained
1 Log file is maintained

LOGFILENAME *Environment*
(Read-only)
Type: String
Saved in: Drawing
Initial value: “C:\ACAD2000\acad.log”
Specifies the path and name of the log file for the current drawing. The initial value varies depending on where you installed AutoCAD.

LOGFILEPATH *Environment*
Type: String
Saved in: Registry
Initial value: “C:\ACAD2000\acad.log”
Specifies the path for the log files for all drawings in a session. You can also specify the path by using the OPTIONS command. The initial value varies depending on where you installed AutoCAD.

LOGINNAME *Environment*
(Read-only)
Type: String
Not saved
Displays the user’s name as configured or as input when AutoCAD is loaded. The maximum length for a login name is 30 characters.

LTSCALE *Display*
Type: Real
Saved in: Drawing
Initial value: 1.0000
Sets the global linetype scale factor. The linetype scale factor must be greater than zero.

LUNITS DDUNITS
Type: Integer
Saved in: Drawing
Initial value: 2
Sets linear units.

1 Scientific
2 Decimal
3 Engineering
4 Architectural
5 Fractional

LUPREC DDUNITS
Type: Integer
Saved in: Drawing
Initial value: 4
Sets the number of decimal places displayed for all read-only linear units, and for all editable linear units whose precision is less than or equal to the current LUPREC value. For editable linear units whose precision is greater than the current LUPREC value, the true precision is

displayed. LUPREC does not affect the display precision of dimension text (see DIMSTYLE).

LWDEFAULT LWEIGHT
Type: Enum
Saved in: Registry
Initial value: 25

Sets the value for the default lineweight. The default lineweight can be set to any valid lineweight value in millimeters, including: 0, 5, 9, 13, 15, 18, 20, 25, 30, 35, 40, 50, 53, 60, 70, 80, 90, 100, 106, 120, 140, 158, 200, and 211.
All values must be entered in millimeters. (Multiply a value by 25.4 to convert values from inches to millimeters.)

LWDISPLAY LWEIGHT
Type: Integer
Saved in: Drawing
Initial value: 0
Controls whether the lineweight is displayed on the Model or Layout tab. The setting is saved with each tab in the drawing.

0 Lineweight is not displayed
1 Lineweight is displayed

LWUNITS LWEIGHT
Type: Integer
Saved In: Registry
Initial value: 1
Controls whether lineweight units are displayed in inches or millimeters.

0 Inches
1 Millimeters

MACROTRACE *Environment*
Type: Integer
Saved in: Drawing
Initial value: 0
A debugging tool for DIESEL expressions.

0 MACROTRACE disabled
1 Displays and evaluation of all DIESEL expressions in the command line

MAXACTVP MVIEW
Type: Integer
Saved in: Drawing
Initial value: 64
Sets the maximum number of viewports that can be active at one time in the display. MAXACTVP has no effect on the number of viewports that are plotted.

0 Minimum
64 Maximum

MAXSORT *Environment*
Type: Integer
Initial value: 200
Controls AutoCAD's alphabetizing within dialog boxes. The system variable by default is set to 200 and may be adjust at any time by the user. Determines how many file names, layer names, blocks, linetypes, and so on, are sorted alphabetically in AutoCAD dialog boxes. Setting MAXSORT to 0 will show layers, etc. sorted by creation time (newest first.)

0 Minimum
32767 Maximum

MBUTTONPAN *Environment*
Type: Integer
Saved in: Registry
Initial Value: 1
Controls the behavior of the third button or wheel on the pointing device.

0 Supports the action defined in the AutoCAD menu (.mnu) file.
1 Supports panning by holding and dragging the button or wheel.

MEASUREINIT *Environment*
Type: Integer
Saved in: Registry
Initial value: 0
Sets the initial drawing units as English or metric. Specifically, MEASUREINIT controls which hatch pattern and linetype files an existing drawing uses when it's opened. It also controls which template is used. Uses the same vales as the system variable MEASUREMENT.

0 English; AutoCAD uses the hatch pattern file and linetype file designated by the ANSIHatch and ANSILinetype registry settings.
1 Metric; AutoCAD uses the hatch pattern file and linetype file designated by the ISOHatch and ISOLinetype registry settings.

MEASUREMENT *Environment*
Type: Integer
Saved in: Drawing
Initial value: 0
Sets drawing units as English or metric for the current drawing only. Specifically, MEASUREMENT controls which hatch pattern and linetype files an existing drawing uses when it is opened.

0 English; AutoCAD uses the hatch pattern file and linetype file designated by the ANSIHatch and ANSILinetype registry settings.
1 Metric; AutoCAD uses the hatch pattern file and linetype file designated by the ISOHatch and ISOLinetype registry settings.

The drawing units for new drawings are controlled by MEASUREINIT (MEASUREINIT uses the same values as MEASUREMENT). The MEASUREMENT setting of a drawing always overrides the MEASUREINIT setting.

MENUCTL *Environment*
Type: Integer
Saved in: Registry
Initial value: 1
Controls the page switching of the screen menu.

0 Screen menu does not switch pages in response to keyboard command entry
1 Screen menu does switch pages in response to keyboard command entry

MENUECHO *Environment*
Type: Integer
Not saved
Initial value: 0
Sets menu echo and prompt control bits. The value is the sum of the following:
0 Normal
1 Suppresses echo of menu items (^P in a menu item toggles echoing)
2 Suppresses display of system prompts during menu
4 Disables ^P toggle of menu echoing
8 Displays input/output strings; debugging aid for DIESEL macros

MENUNAME MENU
(Read-only)
Type: String
Saved in: Registry
Stores the menu file name, including the path for the file name.

MIRRTEXT MIRROR
Type: Integer
Saved in: Drawing
Initial value: 1
Controls how the MIRROR command reflects text.

0 Retains text direction
1 Mirrors the text

MODEMACRO *Environment*
Type: String
Not saved
Initial value: “”
Displays a text string on the status line, such as the name of the current drawing, time/date stamp, or special modes.

MTEXTED MTEXT
Type: String

Saved in: Registry
Initial value: “Internal”
Sets the primary and secondary text editors to use for multiline text objects. The default setting is Internal, which calls the internal MTEXT editor. If the mtext object is fewer than 80 characters you can specify :lisped to use the lisp editor.

NOMUTT *Environment*
Type: Integer
Not Saved
Initial Value: 0
Suppresses the message display (muttering) when it wouldn’t normally be suppressed. Displaying messages is the normal mode of AutoCAD, but message display is suppressed during scripts, AutoLISP routines, and so on.

0 Resumes normal muttering behavior
1 Supress muttering indefinitely

OBSCUREDCOLOR HIDE
(2002) SHADEMODE
Type: integer
Saved in: drawing
Initial value: 0
Specifies the color of obscured lines.

OBSCUREDLTYPE HIDE
(2002) SHADEMODE
Type: Integer
Saved in: Drawing
Initial value: 0
Specifies the linetype of obscured lines.

0 Off
1 Solid
2 Dashed
3 Dotted
4 Short dash
5 Medium dash
6 Long dash
7 Double short dash
8 Double medium dash
9 Double long dash
10 Medium long dash
11 sparce dot

OFFSETDIST OFFSET
Type: Real
Not saved
Initial value: 1.0000
Sets the default offset distance.

<0 Offsets an object through a specified point
>0 Sets the default offset distance

OFFSETGAPTYPE OFFSET
Type: Integer
Saved in: Registry
Initial value: 0
Controls how to offset polylines when a gap is created as a result of offsetting the individual polyline segments.

0 Extends the segments to fill the gap
1 Fills the gaps with a filleted arc segment (the radius of the arc segment is equal to the offset distance)
2 Fills the gaps with a chamfered line segment

OLEHIDE *Display*
Type: Integer
Saved in: Registry
Initial value: 0
Controls the display of OLE objects in AutoCAD.

0 All OLE objects are visible
1 OLE objects are visible in paper space only
2 OLE objects are visible in model space only
3 No OLE objects are visible

OLEHIDE affects both screen display and printing.

OLEQUALITY *Display*
Type: Integer
Saved in: Registry
Initial value: 1
Controls the default quality level for embedded OLE objects.

0 Line art quality, such as an embedded spreadsheet
1 Text quality, such as an embedded Word document
2 Graphics quality, such as an embedded pie chart
3 Photograph quality
4 High quality photograph

OLESTARTUP *Display*
Type: Integer
Saved in: Drawing
Initial value: 0
Controls whether the source application of an embedded OLE object loads when plotting. Loading the OLE source application may improve the plot quality.
0 Does not load the OLE source application
1 Loads the OLE source application when plotting

OPMSTATE PROPERTIES
(2000i, 2000)
(undocumented)
(Read-only)
Type: Integer
Not saved
Initial value: 0
Display state for the Objects Properties dockable container.

0 Not displayed in the environment
1 Currently displayed in the environment

ORTHOMODE *Environment*
Type: Integer
Saved in: Drawing
Initial value: 0
Constrains cursor movement to the perpendicular. When ORTHOMODE is turned on, the cursor can move only horizontally or vertically relative to the UCS and the current grid rotation angle.

0 Turns off Ortho mode
1 Turns on Ortho mode

OSMODE *All drawing and modify commands*
Type: Integer
Saved in: Registry
Initial value: 4133
Sets running Object Snap modes using the following bitcodes.

0 NONe
1 ENDpoint
2 MIDpoint
4 CENter
8 NODe
16 QUAdrant
32 INTersection
64 INSertion
128 PERpendicular
256 TANgent
512 NEArest
1024 QUIck
2048 APParent Intersection
4096 EXTension (2000 only)
8192 PARallel (2000 only)
To specify more than one object snap, enter the sum of their values. For example, entering 3 specifies the Endpoint (bitcode 1) and Midpoint (bitcode 2) object snaps. Entering 16383 specifies all object snaps.

OSNAPCOORD *Environment*
Type: Integer

Saved in: Registry
Initial value: 0
Controls whether coordinates entered on the command line override running object snaps.

0 Running object snap settings override keyboard coordinate entry
1 Keyboard entry overrides object snap settings
2 Keyboard entry overrides object snap settings except in scripts

PAPERUPDATE PLOT
Type: Integer
Saved in: Registry
Initial Value: 0
Controls the display of a warning dialog when attempting to print a layout with a paper size different from the paper size specified by the default for the plotter configuration file.

0 Displays a warning dialog box if the paper size specified in the layout is not supported by the plotter
1 Sets paper size to the configured paper size of the plotter configuration file

PDMODE DDTYPE
Type: Integer
Saved in: Drawing
Initial value: 0
Controls how point objects are displayed

0 Dot (minimum)
1 No display

(see POINT command for additional options)

PDSIZE DDPTYPE
Type: Real
Saved in: Drawing
Initial value: 0.0000
Sets the display size for point objects.

0 Creates a point at 5 percent of the drawing area height
>0 Specifies an absolute size
<0 Specifies a percentage of the viewport size

PELLIPSE ELLIPSE
(undocumented)
Type: Integer
Saved in: Drawing
Initial value: 0
Controls whether or not you create at ellipse object as a set of Polylines or as a true ellipse object. When the value of PELLIPSE is set to 0, a true Ellipse object is created versus one created with Polyline segments if the value was set to 1. One of the things that you will notice is that when working with a true Ellipse and you offset it, it will create the new ellipse as a Spline. True ellipses have grips and can be exploded whereas Splines do not and

cannot be exploded. This is untrue if you create the ellipse with PELLIPSE set to 1. Pre-2000 versions did not create a true ellipse but approximated it with segmented polylines.

0 True ellipse
1 Polyline

PERIMETER
(Read-only)
Type: Real
Not saved

AREA
LIST
DBLIST

Stores the last perimeter value computed by the AREA, DBLIST, or LIST commands. Displays on the command line.

PFACEVMAX
(Read-only)
Type: Integer
Not saved
Initial value: 4

PFACE

Sets the maximum number of vertices per face.

PICKADD

All modifying commands

Type: Integer
Saved in: Registry
Initial value: 1
Controls whether subsequent selections replace the current selection set or add to it.

0 Turns off PICKADD. The objects most recently selected become the selection set. Previously selected objects are removed from the selection set. Add more objects to the selection set by pressing SHIFT while selecting.
1 Turns on PICKADD. Each object selected, either individually or by windowing, is added to the current selection set. To remove objects from the set, press SHIFT while selecting.

PICKAUTO

All modifying commands

Type: Integer
Saved in: Registry
Initial value: 1
Controls automatic windowing at the Select Objects prompt.

0 Turns off PICKAUTO
1 Draws a selection window (for either a window or a crossing selection) automatically at the Select Objects prompt

PICKBOX

DDSELECT

Type: Integer
Saved in: Registry
Initial value: 3
Sets the object selection target height, in pixels. Valid range is 0-50.

PICKDRAG *All modifying commands*
Type: Integer
Saved in: Registry
Initial value: 0
Controls the method of drawing a selection window.

0 Draws the selection window using two points. Click the pointing device at one corner then click to select another corner.
1 Draws the selection window using dragging. Click one corner and drag the pointing device; release the button at the other corner.

PICKFIRST *All modifying commands*
Type: Integer
Saved in: Registry
Initial value: 1
Controls whether you select objects before (noun-verb selection) or after you issue a command.

0 Turns off PICKFIRST; you select objects after you issue a command
1 Turns on PICKFIRST; you select objects before you issue a command

PICKSTYLE DDGROUP
Type: Integer HATCH
Saved in: Registry
Initial value: 1
Controls the use of group selection and associative hatch selection.

0 No group selection or associative hatch selection
1 Group selection
2 Associative hatch selection
3 Group selection and associative hatch selection

PLATFORM *Environment*
(Read-only)
Type: String
Not saved
Indicates which AutoCAD platform is in use. One of the following strings may appear:
“Microsoft Windows NT Version 4.00 (x86)” “Microsoft Windows Version 4.00 (x86)”

PLINEGEN POLYLINE
Type: Integer
Saved in: Drawing
Initial value: 0
Sets how linetype patterns generate around the vertices of a 2D polyline. Does not apply to polylines with tapered segments.

0 Generates polylines to start and end with a dash at each vertex
1 Generates the linetype in a continuous pattern around the vertices of the polyline

PLINETYPE POLYLINE
Type: Integer
Saved in: Registry
Initial value: 2
Specifies whether AutoCAD uses optimized 2D polylines. PLINETYPE controls both the creation of new polylines with the PLINE command and the conversion of existing polylines in drawings from previous releases.

0 Polylines in older drawings are not converted when opened; PLINE creates old-format polylines
1 Polylines in older drawings are not converted when opened; PLINE creates optimized polylines
2 Polylines in older drawings are converted when opened; PLINE creates optimized polylines

For more information on the two formats, see the CONVERT command.
PLINETYPE also controls the polyline type created with the following commands: BOUNDARY (when object type is set to Polyline), DONUT, PEDIT (when selecting a line or arc), POLYGON, and SKETCH (when SKPOLY is set to 1).

PLINEWID POLYLINE
Type: Real
Saved in: Drawing
Initial value: 0.0000
Stores the default polyline width.

PLOTID PLOT
Type: String
Saved in: Registry
Initial value: “”
Obsolete. Has no effect in AutoCAD 2000 except to preserve the integrity of pre-AutoCAD 2000 scripts and LISP routines.

PLOTROTMODE PLOT
Type: Integer
Saved in: Registry
Initial value: 1
Controls the orientation of plots.

0 Rotates the effective plotting area so the corner with the Rotation icon aligns with the paper at the lower left for 0, top left for 90, top right for 180, and lower right for 270. X and Y origin offsets are calculated relative to the lower-left corner.
1 Aligns the lower-left corner of the effective plotting area with the lower-left corner of the paper.
2 Works the same as 0 value except that the X and Y origin offsets are calculated relative to the rotated origin position.

PLOTTER PLOT
Type: Integer

Saved in: Registry
Initial value: 0
Obsolete. Has no effect in AutoCAD 2000 except to preserve the integrity of pre-AutoCAD 2000 scripts and LISP routines. Displays on the command line. 0=first plotter, 28 is maximum (29 total).

PLQUIET BATCH PLOTTING
Type: Integer SCRIPT
Saved in: Registry
Initial Value: 0
Controls the display of optional dialog boxes and nonfatal errors for batch plotting and scripts.
0 Displays plot dialog boxes and nonfatal errors
1 Logs nonfatal errors and doesn't display plot-related dialog boxes

POLARADDANG POLAR SNAP
Type: String
Saved in: Registry
Initial value: null
Contains user-defined polar angles. You can add up to 10 angles. Each angle can be up to 25 characters, separated with semicolons (;). AutoCAD displays angles in the format set in the AUNITS system variable.

POLARANG POLAR SNAP
Type: Real
Saved in: Registry
Initial value: 90
Sets the polar angle increment. Values are 90, 45, 30, 22.5, 18, 15,10, and 5.

POLARDIST POLAR SNAP
Type: Real
Saved in: Registry
Initial value: 0.0000
Sets the snap increment when the SNAPSTYL system variable is set to 1 (polar snap).

POLARMODE POLAR SNAP
Type: Real
Saved in: Registry
Initial value: 0.0000
Sets the snap increment when the SNAPSTYL system variable is set to 1 (polar snap). Controls the setting for object snap tracking and polar snap.

Polar angle of measurement:
0 Polar angles are measured from the current UCS
1 Polar angles are measured from selected objects

Object snap tracking:
0 Track orthagonally only]
2 Utilize polar tracking setting in object snap tracking mode

Use additionally supplied polar angles:
0 No
4 Yes

Acquire object snap tracking points:
0 Acquire automatically
8 SHIFT must be pressed to acquire point

POLYSIDES POLYGON
Type: Integer
Not saved
Initial value: 4
Sets the default number of sides for the POLYGON command. The range is 3 to 1024.

POPUPS *Environment*
(Read-only)
Type: Integer
Not saved
Displays the status of the currently configured display driver.

0 Does not support dialog boxes, the menu bar, and icon menus
1 Supports these features

PRODUCT *Environment*
(2000i, 2002)
(Read-only)
Type: String
Default: “AutoCAD”
Returns the product name.

PROGRAM *Environment*
(2000i, 2002)
(Read-only)
Type: String
Default: “acad”
Returns the program name.

PROJECTNAME *Drawing*
Type: String
Saved in: Drawing
Initial value: “”
Assigns a project name to the current drawing. Used when an xref or image is not found in its original path. The project name points to a section in the registry that can contain one or more search paths for each project name defined. Project names and their search directories are created from the Files tab of the Options dialog box.
Project names make it easier for users to manage xrefs and images when drawings are exchanged between customers, or if users have different drive mappings to the same location on a server.

If the xref or image is not found at the original path, the project paths associated with the project name are searched. If the xref or image is not found there, the AutoCAD search path is searched.

PROJMODE TRIM
Type: Integer EXTEND
Saved in: Registry
Initial value: 1
Sets the current Projection mode for trimming or extending.

0 True 3D mode (no projection)
1 Project to the XY plane of the current UCS
2 Project to the current view plane

PROXYGRAPHICS OPEN
Type: Integer
Saved in: Drawing
Initial value: 1
Specifies whether images of proxy objects are saved in the drawing.

0 Does not save image with the drawing; a bounding box is displayed instead
1 Saves image with the drawing

PROXYNOTICE OPEN
Type: Integer
Saved in: Registry
Initial value: 1
Displays a notice when a proxy is created. A proxy is created when you open a drawing containing custom objects created by an application that is not present. A proxy is also created when you issue a command that unloads a custom object's parent application.

0 No proxy warning is displayed
1 Proxy warning is displayed

PROXYSHOW OPEN
Type: Integer
Saved in: Registry
Initial value: 1
Controls the display of proxy objects in a drawing.

0 Proxy objects are not displayed
1 Graphic images are displayed for all proxy objects
2 Only the bounding box is displayed for all proxy objects

PROXYWEBSEARCH TODAY
(2000i, 2002) *Enablers*
Type: Integer
Initial Value: 1
Saved in: Registry

Specifies how AutoCAD goes about checking for Object Enablers.

0 Prevents from checking for Object Enablers
1 AutoCAD will only look for Object Enablers at the Autodesk Point A portal web site. Live connection to the internet must be established.
2 Specifies the number of times AutoCAD will try to check for the Object Enabler

PSLTSCALE MVIEW
Type: Integer
Saved in: Drawing
Initial value: 1
Controls paper space linetype scaling.

0 No special linetype scaling. Linetype dash lengths are based on the drawing units of the space (model or paper) in which the objects were created. Scaled by the global LTSCALE factor.
1 Viewport scaling governs linetype scaling. If TILEMODE is set to 0, dash lengths are based on paper space drawing units, even for objects in model space. In this mode, viewports can have varying magnifications, yet display linetypes identically. For a specific linetype, the dash lengths of a line in a viewport are the same as the dash lengths of a line in paper space. You can still control the dash lengths with LTSCALE.

When you change PSLTSCALE or use a command such as ZOOM with PSLTSCALE set to 1, objects in viewports are not automatically regenerated with the new linetype scale. Use the REGEN or REGENALL commands to update the linetype scales in each viewport.

PSPROLOG PSOUT
Type: String
Saved in: Registry
Initial value: “”
Assigns a name for a prolog section to be read from the acad.psf file when you use the PSOUT command.

PSQUALITY *Drawing*
Type: Integer
Saved in: Registry
Initial value: 75
Controls the rendering quality of PostScript images and whether they are drawn as filled objects or as outlines.

0 Turns off PostScript image generation.
<0 Sets the number of pixels per AutoCAD drawing unit for the PostScript resolution.
>0 Sets the number of pixels per drawing unit but uses the absolute value. Causes AutoCAD to show the PostScript paths as outlines and does not fill them.

PSTYLEMODE PLOT
Read Only
Saved in: Drawing
Initial value: 0

Indicates whether the current drawing is in a Color-Dependent or Named Plot Style mode.

0 Uses named plot style tables in the current drawing
1 Uses color-dependent plot style tables in the current drawing

PSTYLEPOLICY PLOT
Type: Integer
Saved in: Registry
Initial value: 1
Controls whether an object's color property is associated with its plot style. The new value you assign affects only newly created drawings and pre-AutoCAD 2000 drawings.

0 No association is made between color and plot style. The plot style for new objects is set to the default defined in DEFPLSTYLE. The plot style for new layers is set to the default defined in DEFLPLSTYLE.
1 An object's plot style is associated with its color.

PSVPSCALE *Display*
Type: Real
Not Saved
Initial Value: 0.000000000
Sets the view scale factor for all newly created viewports. The view scale factor is defined by comparing the ratio of units in paper space to the units in newly created model space viewports. The view scale factor you set is used with the VPORTS command. A value of 0 means the scale factor is Scaled to Fit. A scale must be a positive real value.

PUCSBASE *Display*
Type: String
Saved in: Drawing
Initial value: ""
Stores the name of the UCS that defines the origin and orientation of orthographic UCS settings in paper space only.

QTEXTMODE *2D/Display*
Type: Integer
Saved in: Drawing
Initial value: 0
Controls how text is displayed.

0 Turns off Quick Text mode; displays characters
1 Turns on Quick Text mode; displays a box in place of text

RASTERPREVIEW *Drawing*
Type: Integer
Saved in: Registry
Initial value: 1
Controls whether BMP preview images are saved with the drawing.

0 No preview image is created

1 Preview image created

REFEDITNAME REFEDIT
(Read-only)
Type: String
Not Saved
Initial value: “”
Indicates whether a drawing is in a reference-editing state; also, stores the reference file name.

REGENMODE *Display*
Type: Integer
Saved in: Drawing
Initial value: 1
Controls automatic regeneration of the drawing.

0 Turns off the REGENAUTO command
1 Turns on the REGENAUTO command

RE-INIT *Environment*
Type: Integer
Not saved
Initial value: 0
Reinitializes the digitizer, digitizer port, and acad.pgp file by using the following bitcodes:

1 Digitizer input/output port reinitialized
2 Plotter port reinitialized
4 Digitizer reinitialized
8 Display reinitialized
16 PGP file reinitialized (reload)

To specify more than one reinitialization, enter the sum of the bitcode values. For example, enter 5 to specify both digitizer port (1) and digitizer reinitialization (4.)

REMEMBERFOLDERS *Environment*
(2000i, 2002)
Type: Integer
Saved in: Registry
Initial value: 1
Controls the path that is used to Look IN or Save In as the default/

0 Uses the Start In value of the shortcut icon on the desktop. This setting is used to restore the behavior of AutoCAD 2000 and previous releases
1 Uses the last path that was called up from each particular dialog box. The Start In value is ignored for AutoCAD 2000i after the session has been started and changes have been made under each dialog box. The values are only held while the session is active.

RTDISPLAY *Display* – RTZOOM and RTPAN

Type: Integer
Saved in: Registry
Initial value: 1
Controls the display of raster images during Realtime ZOOM or PAN.

0 Displays raster image content
1 Displays raster image outline only

RTDISPLAY is saved in the current profile.

SAVEFILE *Environment*
(Read-only)
Type: String
Saved in: Registry
Initial value: “”
Stores the current automatic save file name.

SAVEFILEPATH *Environment*
Type: String
Saved in: Registry
Initial value: “C:\TEMP\”
Specifies the path to the directory for all automatic save files for the AutoCAD session. You can also change the path on the Files tab in the Options dialog box.

SAVENAME *Environment*
(Read-only)
Type: String
Not saved
Initial value: “”
Stores the file name and directory path of the current drawing after you save it.

SAVETIME *Environment*
Type: Integer
Saved in: Registry
Initial value: 120
Sets the automatic save interval, in minutes. Temp file of drawing is saved to the name supplied in the SAVEFILE variable.

0 Turns off automatic saving
>0 Saves the drawing at intervals specified by the nonzero integer automatically

The SAVETIME timer starts as soon as you make a change to a drawing. It is reset and restarted by a manual QSAVE, SAVE, or SAVEAS. The current drawing is saved to the path specified by the SAVEFILEPATH system variable. The file name is stored in the SAVEFILE system variable.

SCREENBOXES *Display*

(Read-only)
Type: Integer
Not saved
Initial value: 0
Stores the number of boxes in the screen menu area of the drawing area. If the screen menu is turned off, SCREENBOXES is zero. On platforms that permit the drawing area to be resized or the screen menu to be reconfigured during an editing session, the value of this variable might change during the editing session.

SCREENMODE *Display*
(Read-only)
Type: Integer
Not saved
Initial value: 3
Stores a bitcode indicating the graphics/text state of the AutoCAD display. It is the sum of the following bit values:

0 Text screen is displayed
1 Drawing area is displayed
2 Dual-screen display is configured

SCREENSIZE *Environment*
(Read-only)
Type: 2D point
Not saved
Stores current viewport size in pixels (X and Y).

SDI *Environment*
Type: Integer
Saved in: Registry
Initial value: 0
Controls whether AutoCAD runs in single- or multiple-document interface. Helps third-party developers update applications to work smoothly with the AutoCAD multiple-drawing mode.

0 Turns on multiple-drawing interface.
1 Turns off multiple-drawing interface.
2 (Read-only) Multiple-drawing interface is disabled because AutoCAD has loaded an application that does not support multiple drawings. SDI setting 2 is not saved.
3 (Read-only) Multiple-drawing interface is disabled because the user has set SDI to 1 and AutoCAD has loaded an application that does not support multiple drawings. (SDI was set to 1 before the application was loaded.) SDI setting 3 is not saved.

If SDI is set to 3, AutoCAD switches it back to 1 when the application that doesn't support multiple drawings is unloaded.

SHADEDGE SHADE
Type: Integer
Saved in: Drawing
Initial value: 3
Controls the shading of edges in rendering.

0 Faces shaded, edges not highlighted
1 Faces shaded, edges drawn in background color
2 Faces not filled, edges in object color
3 Faces in object color, edges in background color

SHADEDIF SHADE
Type: Integer
Saved in: Drawing
Initial value: 70
Sets the ratio of diffuse reflective light to ambient light in percentage of diffuse reflective light, when SHADEDGE is set to 0 or 1.
0 Minimum
70 Default
100 Maximum

SHORTCUTMENU *Environment*
Type: Integer
Saved in: Registry
Initial value: 11
Controls whether Default, Edit, and Command mode shortcut menus are available in the drawing area. SHORTCUTMENU uses the following bitcodes:

0 Disables all Default, Edit, and Command mode shortcut menus, restoring R14 legacy behavior.
1 Enables Default mode shortcut menus.
2 Enables Edit mode shortcut menus.
4 Enables Command mode shortcut menus. In this case, the Command mode shortcut menu is available whenever a command is active.
8 Enables Command mode shortcut menus only when command options are currently available from the command line.

To enable more than one type of shortcut menu at once, enter the sum of their values. For example, entering 3 enables both Default (1) and Edit (2) mode shortcut menus.

SHPNAME SHAPE
Type: String
Not saved
Initial value: “”
Sets a default shape name that must conform to symbol naming conventions. If no default is set, it returns “”. Enter a period (.) to set no default.

SKETCHINC SKETCH
Type: Real
Saved in: Drawing
Initial value: 0.1000
Sets the record increment for the SKETCH command. It can be any value greater than zero but a very small number will use a lot of memory.

SKPOLY SKETCH
Type: Integer
Saved in: Drawing
Initial value: 0
Determines whether the SKETCH command generates lines or polylines.

0 Generates lines
1 Generates polylines

SNAPANG *All drawing and modifying commands*
Type: Real DDRMODES
Saved in: Drawing
Initial value: 0
Sets the snap and grid rotation angle for the current viewport. The angle you specify is relative to the current UCS.
Changes to this variable are not reflected in the grid until the display is refreshed. AutoCAD does not redraw automatically when system variable settings are changed.

SNAPBASE *All drawing and modifying commands*
Type: 2D point DDRMODES
Saved in: Drawing
Initial value: 0.0000,0.0000
Sets the snap and grid origin point for the current viewport relative to the current UCS.

Changes to this variable are not reflected in the grid until the display is refreshed. AutoCAD does not redraw automatically when system variable settings are changed.

SNAPISOPAIR *All drawing and modifying commands*
Type: Integer DDRMODES
Saved in: Drawing
Initial value: 0
Controls the isometric plane for the current viewport.

0 Left Isometric
1 Top Isometric
2 Right Isometric

SNAPMODE *All drawing and modifying commands*
Type: Integer DDRMODES
Saved in: Drawing
Initial value: 0
Turns the Snap mode on and off.

0 Snap off
1 Snap on for the current viewport

SNAPSTYL *All drawing and modifying commands*
Type: Integer DDRMODES
Saved in: Drawing

Initial value: 0
Sets the snap style for the current viewport.

0 Standard (rectangular snap)
1 Isometric snap

SNAPTYPE *All drawing and modifying commands*
Type: Integer DDRMODES
Saved in: Registry
Initial Value: 0
Sets the snap style for the current viewport.

0 Grid, or standard snap.
1 Polar snap. Snaps along polar angle increments. Use polar snap with polar and object snap tracking.

SNAPUNIT *All drawing and modifying commands*
Type: 2D point DDRMODES
Saved in: Drawing
Initial value: 0.5000,0.5000
Sets the snap spacing for the current viewport. If SNAPSTYL is set to 1, AutoCAD adjusts the X value of SNAPUNIT automatically to accommodate the isometric snap.
Changes to this system variable are not reflected in the grid until the display is refreshed. AutoCAD does not redraw automatically when system variable settings are changed.

SOLIDCHECK *Environment*
Type: Integer
Saved in: Not Saved
Initial value: 1
Turns the solid validation on and off for the current AutoCAD session.

0 Turns off solid validation
1 Turns on solid validation

SORTENTS DDSELECT
Type: Integer
Saved in: Drawing
Initial value: 96
Controls the OPTIONS command (from the Selection tab) object sort order operations. SORTENTS uses the following bitcodes:

0 Disables SORTENTS
1 Sorts for object selection
2 Sorts for object snap
4 Sorts for redraws
8 Sorts for MSLIDE command slide creation
16 Sorts for REGEN commands
32 Sorts for plotting
64 Sorts for PostScript output

To select more than one setting, enter the sum of the settings' codes. For example, enter 3 to specify sorting for both object selection and object snap.
The initial value of 96 enables sorting for plotting and PostScript output only. Setting additional sorting options can result in slower regeneration and redrawing times.

SPLFRAME
Type: Integer
Saved in: Drawing
Initial value: 0
Display
EDGE
Controls the display of splines and spline-fit polylines.

0 Does not display the control polygon for splines and spline-fit polylines. Displays the fit surface of a polygon mesh, not the defining mesh. Does not display the invisible edges of 3D faces or polyface meshes.
1 Displays the control polygon for splines and spline-fit polylines. Only the defining mesh of a surface-fit polygon mesh is displayed (not the fit surface). Invisible edges of 3D faces or polyface meshes are displayed.

SPLINESEGS
Type: Integer
Saved in: Drawing
Initial value: 8
SPLINE
Sets the number of line segments to be generated for each spline-fit polyline generated by the Spline option of the PEDIT command.
Valid settings range from –32768 to 32767. If you set SPLINESEGS to a negative value, AutoCAD generates segments using the absolute value of the setting, and then applies a fit-type curve to those segments. Fit-type curves use arcs as the approximating segments. Using arcs yields a smoother generated curve when few segments are specified, but the curve can take longer to generate.

SPLINETYPE
Type: Integer
Saved in: Drawing
Initial value: 6
PEDIT
Sets the type of curve generated by the Spline option of the PEDIT command.

5 Quadratic B-spline
6 Cubic B-spline

STARTUPTODAY
(2000i, 2002)
Type: Integer
Saved in: Register
Initial value: 1
TODAY
NEW
OPEN
Controls whether or not the Today window is used or the Traditional startup dialog box.

0 Displays Traditional startup dialog box
1 Displays Today window

SURFTAB1 SURFTAB
Type: Integer
Saved in: Drawing
Initial value: 6
Sets the number of tabulations to be generated for the RULESURF and TABSURF commands. Also sets the mesh density in the M direction for the REVSURF and EDGESURF commands. Values are 2-32766.

SURFTAB2 SURFTAB
Type: Integer
Saved in: Drawing
Initial value: 6
Sets the mesh density in the N direction for the REVSURF and EDGESURF commands. Values are 2-32766.

SURFTYPE PEDIT
Type: Integer
Saved in: Drawing
Initial value: 6
Controls the type of surface-fitting to be performed by the Smooth option of the PEDIT command.

5 Quadratic B-spline surface
6 Cubic B-spline surface
8 Bezier surface

SURFU *3D/Draw*
Type: Integer PEDIT
Saved in: Drawing
Initial value: 6
Sets the surface density for PEDIT Smooth in the M direction. Values are 2-200.

SURFV *3D/Draw*
Type: Integer PEDIT
Saved in: Drawing
Initial value: 6
Sets the surface density for PEDIT Smooth in the N direction. Values are 2-200.

SYSCODEPAGE *Environment*
(Read-only)
Type: String
Not saved
Initial value: “ansi_1251”
Indicates the system code page specified in the acad.xmx file.

TABMODE *Environment*
Type: Integer
Not saved

Initial value: 0
Controls the use of the tablet. For more information on using and configuring a tablet, see the command TABLET.

0 Turns off Tablet mode
1 Turns on Tablet mode

TARGET DDVIEW
(Read-only)
Type: 3D point
Saved in: Drawing
Stores the location (as a UCS coordinate) of the target point for the current viewport.

TDCREATE *Drawing*
(Read-only)
Type: Real
Saved in: Drawing
Stores the local time and date the drawing was created in Julian format.

TDINDWG *Drawing*
(Read-only)
Type: Real
Saved in: Drawing
Stores the total editing time in Julian format.

TDUPDATE Drawing
(Read-only)
Type: Real
Saved in: Drawing
Stores the local time and date of the last update/save in Julian format.

TDUSRTIMER *Drawing*
(Read-only)
Type: Real
Saved in: Drawing
Stores the user-elapsed timer in Julian format.

TDUUPDATE *Drawing*
(Read-only)
Type: Real
Saved in: Drawing
Stores the universal time and date of the last update/save in Julian format.

TEMPPREFIX *Environment*
(Read-only) TOOLS/OPTIONS
Type: String
Not saved

Contains the directory name (if any) configured for placement of temporary files, with a path separator appended.

TEXTEVAL TEXT
Type: Integer DTEXT
Not saved
Initial value: 0
Controls the method of evaluation of text strings.

0 All responses to prompts for text strings and attribute values are taken literally
1 Text starting with an opening parenthesis [( ] or an exclamation mark (!) is evaluated as an AutoLISP expression, as for nontextual input

The TEXT command takes all input literally regardless of the setting of TEXTEVAL.

TEXTFILL *Display*
Type: Integer
Saved in: Registry
Initial value: 1
Controls the filling of TrueType fonts while plotting, exporting with the PSOUT command, and rendering.

0 Outputs text as outlines
1 Outputs text as filled images

TEXTQLTY *Environment*
Type: Integer
Not saved
Initial value: 50
Sets the resolution tesselation fineness of text outlines for TrueType fonts while plotting, exporting with the PSOUT command, and rendering. 0 represents no effort to refine the smoothness of the text; 100 represents a maximum effort to smooth text characters. Values represent dots-per-inch. Lower values decrease resolution and increase plotting speed. Higher values increase resolution and decrease plotting speed.

TEXTSIZE TEXT
Type: Real DTEXT
Saved in: Drawing
Initial value: 0.2000
Sets the default height for new text objects drawn with the current text style (has no effect if the style has a fixed height).

TEXTSTYLE TEXT
Type: String DTEXT
Saved in: Drawing
Initial value: “STANDARD”
Sets the name of the current text style.

THICKNESS *All drawing commands except 3D commands*
Type: Real
Saved in: Drawing
Initial value: 0.0000
Sets the current 3D solid thickness.

TILEMODE *Drawing*
Type: Integer
Saved in: Drawing
Initial value: 1
Makes the Model tab or the last layout tab current.

0 Makes the last active layout tab (paper space) active
1 Makes the Model tab active

TOOLTIPS *Environment*
Type: Integer TOOLBAR
Saved in: Registry
Initial value: 1
Controls the display of tooltips.

0 Turns off the display of tooltips
1 Turns on the display of tooltips

TRACEWID TRACE
Type: Real
Saved in: Drawing
Initial value: 0.0500
Sets the default trace width.

TRACKPATH POLAR SNAP
Type: Integer OBJECT SNAP
Saved in: Registry TRACKING
Initial value: 0
Controls the display of polar and object snap tracking alignment paths.

0 Displays full screen object snap tracking path
1 Displays object snap tracking path only between the alignment point and From point to cursor location
2 Does not display polar tracking path
3 Does not display polar or object snap tracking paths

TREEDEPTH *Environment*
Type: Integer
Saved in: Drawing
Initial value: 3020
Specifies the maximum depth, that is, the number of times the tree-structured spatial index can divide into branches.

0 Suppresses the spatial index entirely, eliminating the performance improvements it provides in working with large drawings. This setting assures that objects are always processed in database order, making it unnecessary ever to set SORTENTS.

>0 Turns on TREEDEPTH. An integer of up to four digits is valid. The first two digits refer to model space, and the second two digits refer to paper space.

<0 Treats model space objects as 2D (Z coordinates are ignored), as is always the case with paper space objects. Such a setting is appropriate for 2D drawings and makes more efficient use of memory without loss of performance.

NOTE: You cannot use TREEDEPTH transparently.

TREEMAX *Environment*
Type: Integer
Saved in: Registry
Initial value: 10000000
Limits memory consumption during drawing regeneration by limiting the number of nodes in the spatial index (oct-tree).
By imposing a fixed limit with TREEMAX, you can load drawings created on systems with more memory than your system and with a larger TREEDEPTH than your system can handle. These drawings, if left unchecked, have an oct-tree large enough to eventually consume more memory than is available to your computer. TREEMAX also provides a safeguard against experimentation with inappropriately high TREEDEPTH values.
The initial default for TREEMAX is 10000000 (10 million), a value high enough to effectively disable TREEMAX as a control for TREEDEPTH. The value to which you should set TREEMAX depends on your system's available RAM. You get about 15,000 oct-tree nodes per megabyte of RAM.
If you want an oct-tree to use up to, but no more than, 2 megabytes of RAM, set TREEMAX to 30000 (2 x 15,000). If AutoCAD runs out of memory allocating oct-tree nodes, restart AutoCAD, set TREEMAX to a smaller number, and try loading the drawing again.
AutoCAD might occasionally run into the limit you set with TREEMAX. Follow the resulting prompt instructions. Your ability to increase TREEMAX depends on your computer's available memory.

TRIMMODE CHAMFER
Type: Integer FILLET
Saved in: Registry
Initial value: 1
Controls whether AutoCAD trims selected edges for chamfers and fillets.

0 Leaves selected edges intact
1 Trims selected edges to the endpoints of chamfer lines and fillet arcs

TSPACEFAC MTEXT
Type: Real
Not saved
Initial value: 1.000
Controls the multiline text line spacing distance measured as a factor of text height. Valid values are 0.25 to 4.0.

TSPACETYPE MTEXT

Type: Integer
Not saved
Initial value: 1
Controls the type of line spacing used in multiline text.

1 At Least adjusts the line spacing based on the tallest characters that are found in a line
2 Exactly uses the value for line spacing, rather than individual character sizes

TSTACKALIGN MTEXT
Type: Integer
Saved in: Drawing
Initial value: 1
Controls the vertical alignment of stacked text.

0 Bottom aligned
1 Center aligned
2 Top aligned

TSTACKSIZE MTEXT
Type: Integer
Saved in: Drawing
Initial value: 70
Controls the percentage of stacked text fraction height relative to selected text's current height. Valid range is 1 to 127.

UCSAXISANG UCS
Type: Integer UCSMAIN
Saved in: Registry
Initial value: 90
Stores the default angle when rotating the UCS around one of its axes using the X, Y, or Z options of the UCS command. Its value must be entered as an angle in degrees (valid values are: 5, 10, 15, 18, 22.5, 30, 45, 90, 180).

UCSBASE UCS
Type: String UCSMAIN
Saved in: Drawing
Initial value: "World"
Stores the name of the UCS that defines the origin and orientation of orthographic UCS settings. Valid values include any named UCS.

UCSFOLLOW *Display*
Type: Integer
Saved in: Drawing (viewport specific)
Initial value: 0
Generates a plan view whenever you change from one UCS to another. Set UCSFOLLOW separately for each viewport. If UCSFOLLOW is on for a particular viewport, AutoCAD generates a plan view in that viewport whenever you change coordinate systems. Once the new UCS has been established, you can use DVIEW, PLAN, VIEW, or VPOINT to change

the view of the drawing. It will change to a plan view again the next time you change coordinate systems.

0 UCS does not affect the view
1 Any UCS change causes a change to the plan view of the new UCS in the current viewport

The setting of UCSFOLLOW is maintained separately for paper space and model space and can be accessed in either, but the setting is ignored while in paper space (it is always treated as if set to 0). Although you can define a non-world UCS in paper space, the view remains in plan view to the world coordinate system.

UCSICON *Display*
Type: Integer
Saved in: Drawing (viewport specific)
Initial value: 3
Displays the UCS icon for the current viewport using bitcode. UCSICON is both a command and a system variable. It is the sum of the following:

0 No icon displayed
1 On; icon is displayed
2 Origin; if icon is displayed, the icon floats to the UCS origin if possible
3 On and displayed at origin.

The UCSICON command controls the visibility and placement of the UCS icon. Because entering ucsicon at the Command prompt invokes the UCSICON command, you must use the SETVAR command to access the UCSICON system variable.

UCSNAME *Display*
(Read-only)
Type: String
Initial value: “”
Saved in: Drawing (viewport specific)
Stores the name of the current coordinate system for the current viewport in the current space. Returns a null string if the current UCS is unnamed.

UCSORG *Display*
(Read-only)
Type: 3D point
Saved in: Drawing (viewport specific)
Stores the origin point of the current coordinate system for the current viewport in the current space. This value is always stored as a world coordinate.

UCSORTHO *Display*
Type: Integer
Saved in: Registry
Initial value: 1
Determines whether the related orthographic UCS setting is restored automatically when an orthographic view is restored.

0 Specifies that the UCS setting remains unchanged when an orthographic view is restored
1 Specifies that the related orthographic UCS setting is restored automatically when an orthographic view is restored

UCSVIEW *Display*
Type: Integer
Saved in: Registry
Initial value: 1
Determines whether the current UCS is saved with a named view.

0 Does not save current UCS with a named view
1 Saves current UCS whenever a named view is created

UCSVP *Display*
Type: Integer
Saved in: Drawing (viewport specific)
Initial value: 1
Determines whether the UCS in active viewports remains fixed or changes to reflect the UCS of the currently active viewport.

0 Unlocked; UCS reflects the UCS of the current viewport
1 Locked; UCS stored in viewport, and is independent of the UCS of the current viewport

UCSXDIR *Display*
(Read-only)
Type: 3D point
Saved in: Drawing (viewport specific)
Stores the X direction of the current UCS for the current viewport in the current space.

UCSYDIR *Display*
(Read-only)
Type: 3D point
Saved in: Drawing (viewport specific)
Stores the Y direction of the current UCS for the current viewport in the current space.

UNDOCTL *Drawing*
(Read-only)
Type: Integer
Not saved
Initial value: 13
Stores a bitcode indicating the state of the Auto and Control options of the UNDO command. It's the sum of the following values:

0 UNDO is turned off
1 UNDO is turned on
2 Only one command can be undone
4 Auto is turned off

8 A group is currently active

UNDOMARKS *Drawing*
(Read-only)
Type: Integer
Not saved
Initial value: 1
Stores a bitcode indicating the state of the Auto and Control options of the UNDO command. It's the sum of the following values:

0 UNDO is turned off
1 UNDO is turned on
2 Only one command can be undone
4 Auto is turned off
8 A group is currently active

UNITMODE *Environment*
Type: Integer
Saved in: Drawing
Initial value: 0
Controls the display format for units.

0 Displays fractional, feet-and-inches, and surveyor's angles as previously set
1 Displays fractional, feet-and-inches, and surveyor's angles in input format

USERI1-5 *Drawing*
Type: Integer
Saved in: Drawing
Initial value: 0
USERI1, USERI2, USERI3, USERI4, and USERI5 are used for storage and retrieval of integer values.

USERR1-5 *Drawing*
Type: Real
Saved in: Drawing
Initial value: 0.0000
USERR1, USERR2, USERR3, USERR4, and USERR5 are used for storage and retrieval of real numbers.

USERS1-5 *Drawing*
Type: String
Not saved
Initial Value: ""
USERS1, USERS2, USERS3, USERS4, and USERS5 are used for storage and retrieval of text string data.

VIEWCTR *Display*
(Read-only)

Type: 3D point
Saved in: Drawing
Stores the center of view in the current viewport. Expressed as a UCS coordinate.

VIEWDIR *Display*
(Read-only)
Type: 3D vector
Saved in: Drawing
Stores the viewing direction in the current viewport expressed in UCS coordinates. This describes the camera point as a 3D offset from the target point.

VIEWMODE *Display*
(Read-only)
Type: Integer
Saved in: Drawing
Controls the View mode for the current viewport using bitcode. The value is the sum of the following:

0 Turned off.
1 Perspective view active.
2 Front clipping on.
4 Back clipping on.
8 UCS Follow mode on.
16 Front clip not at eye. If on, the front clip distance (FRONTZ) determines the front clipping plane. If off, FRONTZ is ignored, and the front clipping plane is set to pass through the camera point (vectors behind the camera are not displayed). This flag is ignored if the front clipping bit (2) is off.

VIEWSIZE *Display*
(Read-only)
Type: Real
Saved in: Drawing
Stores the height of the view in the current viewport. Expressed in drawing units.

VIEWTWIST DVIEW
(Read-only)
Type: Real
Saved in: Drawing
Stores the view twist angle for the current viewport.

VISRETAIN XREF
Type: Integer
Saved in: Drawing
Initial value: 1
Controls the visibility, color, linetype, lineweight, and plot styles (if PSTYLEPOLICY is set to 0) of xref-dependent layers and specifies whether nested xref path changes are saved.

0 The layer table, as stored in the reference drawing (xref) takes precedence. Changes made to xref-dependent layers in the current drawing are valid in the current session

only and are not saved with the drawing. When the current drawing is reopened, the layer table is reloaded from the reference drawing and the current drawing reflects those settings. The layer settings affected are On, Off, Freeze, Thaw, Color, Ltype, LWeight, and PStyle (if PSTYLEPOLICY is set to 0). This setting also specifies that changes made to the paths of nested xrefs are for the current session only and are not saved with the drawing.

1 Xref-dependent layer changes made in the current drawing take precedence. Layer settings are saved with the current drawing's layer table and persist from session to session. Nested xref path changes are saved with the current drawing and persist from session to session.

VSMAX *Display*
(Read-only)
Type: 3D point
Saved in: Drawing
Stores the upper-right corner of the current viewport's virtual screen. Expressed as a UCS coordinate.

VSMIN *Display*
(Read-only)
Type: 3D point
Saved in: Drawing
Stores the lower-left corner of the current viewport's virtual screen. Expressed as a UCS coordinate.

WEBPROXYSEARCH *Environment*
(2000i, 2002)
Type: Integer
Saved in: Drawing
Initial value: 1
Specifies how AutoCAD goes about checking for Object Enablers.

0 Prevents AutoCAD from checking for Object Enablers
1 AutoCAD will only look for Object Enablers at the Autodesk Point A portal web site. Live connection to the internet must be established
2 Specifies the number of times AutoCAD will try to check for the Object Enabler

WHIPARC *Display*
Type: Integer
Saved in: Registry
Initial value: 0
Controls whether the display of circles and arcs is smooth.

0 Circles and arcs are not smooth, but rather are displayed as a series of vectors.
1 Circles and arcs are smooth, displayed as true circles and arcs.

WHIPTHREAD *Display*
(2000i, 2002)

Type: Integer
Saved in: Registry
Initial value: 3
Specifies how AutoCAD will use additional processors on the machine. This system variable has no effect on machines with single processors.

0 No multithreading is used
1 Regeneration only is carried out by multiple processors
2 Redraw only is carried out by multiple processors
3 Regeneration and Redraw are carried out by multiple processors

WMFBKGND EXPORT
Type: Integer
Not saved
Initial value: 1
Controls whether the background display of AutoCAD objects is transparent in other applications when these objects are output to a Windows metafile using the WMFOUT command.
Copied to the Clipboard in AutoCAD and pasted as a Windows metafile.
Dragged and dropped from AutoCAD as a Windows metafile
The AutoCAD defined values are:

0 The background is transparent.
1 The background color is the same as the AutoCAD current background color.

WMFFFOREGND EXPORT
(2000i, 2002)
Type: Integer
Not saved
Initial value: 0
Controls how the foreground color assignment to AutocAD objects of a WMF file or copyclip is created and displayed in other applications.

0 Foreground and background colors are swapped in order to ensure that the foreground color is *darker* than the background color
1 Foreground and background color are swapped in order to ensure that the foreground color is *lighter* than the background color

WORLDUCS *Display*
(Read-only)
Type: Integer
Not saved
Initial value: 1
Indicates whether the UCS is the same as the WCS.

0 UCS differs from the WCS
1 UCS matches the WCS

WORLDVIEW DVIEW
Type: Integer VPOINT

Saved in: Drawing
Initial value: 1
Determines whether input to the 3DORBIT, DVIEW, and VPOINT commands is relative to the WCS (default), the current UCS, or the UCS specified by the UCSBASE system variable.

0 UCS remains unchanged
1 UCS changes to the WCS for the duration of the DVIEW or VPOINT commands; the DVIEW and VPOINT command input is relative to the current UCS

WRITESTAT *Drawing*
(Read-only)
Not saved
Initial value: 1
Indicates whether a drawing file is read-only or can be written to. For developers who need to determine write status through AutoLISP.

0 Read only
1 Read and write

XCLIPFRAME XREF
Type: Integer
Saved in: Drawing
Initial value: 0
Controls the visibility of xref clipping boundaries.

0 Clipping boundary is not visible
1 Clipping boundary is visible

XEDIT REFEDIT
Type: Integer
Saved in: Drawing
Initial value: 1
Controls whether the current drawing can be edited in-place when being referenced by another drawing.

0 Can’t use in-place reference editing
1 Can use in-place reference editing

XFADECTL REFEDIT
Type: Integer
Saved in: Registry
Initial value: 50
Controls the fading intensity for references being edited in-place.

0 0 percent fading, minimum value
90 90 percent fading, maximum value

XLOADCTL XREF

Type: Integer
Saved in: Registry
Initial value: 1
Turns xref demand loading on and off and controls whether it opens the original drawing or a copy.

0 Turns off demand loading; entire drawing is loaded
1 Turns on demand loading, reference file is kept open
2 Turns on demand loading; a copy of the reference file is opened

When XLOADCTL is set to 2, the reference file copy is stored in the AutoCAD temporary files directory (defined by the OPTIONS command) or in a user-specified directory.

XLOADPATH *Environment*
Type: String XREF
Saved in: Registry
Initial value: “”
Creates a path for storing temporary copies of demand-loaded xref files. For more information, see XLOADCTL.

XREFCTL *Environment*
Type: Integer XREF
Saved in: Registry
Initial value: 0
Controls whether AutoCAD writes external reference log (XLG) files.

0 Does not write Xref log files
1 Writes Xref log files

ZOOMFACTOR *Display – Intellimouse support*
Type: Interger
Saved in: Registry
Initial value: 10
Accepts an integer between 3 and 100 as valid values. The higher the number, the more incremental the change applied by each mouse-wheel’s forward/backward movement.

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NOTE: There are many variables that, even though then can be called from the command line, no longer function or are read-only. Some of them are:

_SERVER, _VERNUM, AUXSTAT, AXISMODE, AXISUNIT, DBGLISTALL, ENTEXTS, ENTMODS, FORCE_PAGING, GLOBCHECK, LAZYLOAD, NODENAME, PHANDLE, and QAFLAGS

To save some time nosing around in this document, there are only 5 variables associated only with 2002: DIMASSOC, HALOGAP, HIDETEXT, OBSCUREDCOLOR, and OBSCUREDLTYPE.

And there are only 4 associated only with 2000i and 2002: LAYOUTREGENCTL, PROXYWEBSEARCH, REMEMBERFOLDERS, and STARTUPTODAY.

Compiled from various sources by:
R. B. “Andy” Anderson
2744 Botts Landing Road, #1103
Deland, Florida 32720-8953
andyanderson@bigfoot.com

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Statistics:

Total number of variables documented: 377

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