

# 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	AREA
(Read-only)	DBLIST
Type: Real	LIST
Not saved	

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

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

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

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

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

Type: Integer

Saved in: Drawing

Initial value: 0

Sets units for angles.

#### DDUNITS

UNITS

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

#### AUPREC

Type: Integer

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?*

#### DDUNITS

UNITS

#### AUTOSNAP

Type: Integer

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:

#### DDOSNAP

OSNAP

- 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

(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

#### DVIEW

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

XREF  
REFEDIT

Controls how xref names are handled when binding xrefs or editing xrefs in place.

- 0 Traditional binding behavior ("xref1|one" becomes "xref\$0\$one")
- 1 Insert-like behavior ("xref1|one" becomes "one")

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

DDRMODES  
BLIPMODE

Controls whether marker blips are visible.

- 0 Turns off marker blips
- 1 Turns on marker blips

CDATE  
Type: Real  
Not saved

TIME

Shows current calendar date and time. In the format yyyyymmdd.hhhhhmmss.

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

Type: Real

Not saved

Initial value: 0.0000

Sets the default circle radius. A zero indicates no default.

## CIRCLE

## CLAYER

Type: String

Saved in: Drawing

Initial value: "0"

Sets the current layer.

## DDLMODES

LAYER

STATUS

## CMDACTIVE

(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:

*Any command*

- 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

Type: Integer

Saved in: Registry

Initial value: 1

Controls the display of dialog boxes for the PLOT command and the external database (ASE) commands.

*Environment*

- 0 No dialog boxes
- 1 Shows dialog boxes

## CMDECHO

Type: Integer

Not saved

Initial value: 1

Controls whether AutoCAD echoes prompts and input during the AutoLISP command function.

*Environment*

- 0 Turns off echoing
- 1 Turns on echoing

## CMDNAMES

(Read-only)

*Any command*

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). Multiline 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."

DEFPLSTYLE                      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 single 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
"_ARCTICK"	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,  $\frac{1}{2}$ )

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 DIMITOL 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<sup>th</sup> 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 100<sup>th</sup> 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 (Read-only)

## DISTANCE

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

Type: Integer

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)

## PARTIALLOAD

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 HIDE TEXT 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 specified, 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: " <a href="http://www.autodesk.com/acaduser">www.autodesk.com/acaduser</a> "	
Stores the Internet location used by the BROWSER command and the Browse the Web dialog box.	

INSBASE INSERT

Stores the insertion base point set by BASE, which gets expressed as a UCS coordinate for the current space.

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.

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                |

Sets source content units value. Valid range is 0 to 20.

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

## LENSENGTH 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 ANSIShatch 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 ANSIShatch 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 :lisp 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 Suppress muttering indefinitely

OBSCUREDCOLOR  
(2002)

HIDE  
SHADEMODE

Type: integer

Saved in: drawing

Initial value: 0

Specifies the color of obscured lines.

OBSCUREDLYTYPE  
(2002)

HIDE  
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 sparse 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 an 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

Stores the last perimeter value computed by the AREA, DBLIST, or LIST commands.

Displays on the command line.

AREA

LIST

DBLIST

PFACEVMAX

(Read-only)

Type: Integer

Not saved

Initial value: 4

Sets the maximum number of vertices per face.

PFACE

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

Type: Integer

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

BATCH PLOTTING

SCRIPT

POLARADDANG

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.

POLAR SNAP

POLARANG

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.

POLAR SNAP

POLARDIST

Type: Real

Saved in: Registry

Initial value: 0.0000

Sets the snap increment when the SNAPSTYL system variable is set to 1 (polar snap).

POLAR SNAP

POLARMODE

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 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 orthogonally 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.

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

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

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

55

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

Type: Integer

Saved in: Drawing

Initial value: 0

Determines whether the SKETCH command generates lines or polylines.

0 Generates lines

1 Generates polylines

## SKETCH

## SNAPANG

Type: Real

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.

*All drawing and modifying commands*

DDRMODES

## SNAPBASE

Type: 2D point

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.

*All drawing and modifying commands*

DDRMODES

## SNAPISOPAIR

Type: Integer

Saved in: Drawing

Initial value: 0

Controls the isometric plane for the current viewport.

0 Left Isometric

1 Top Isometric

2 Right Isometric

*All drawing and modifying commands*

DDRMODES

## SNAPMODE

Type: Integer

Saved in: Drawing

Initial value: 0

Turns the Snap mode on and off.

0 Snap off

1 Snap on for the current viewport

*All drawing and modifying commands*

DDRMODES

## SNAPSTYL

Type: Integer

Saved in: Drawing

*All drawing and modifying commands*

DDRMODES

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

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 Display

Type: Integer

EDGE

Saved in: Drawing

Initial value: 0

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

## SPLINE

Type: Integer

Saved in: Drawing

Initial value: 8

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

PEDIT

Type: Integer

Saved in: Drawing

Initial value: 6

Sets the type of curve generated by the Spline option of the PEDIT command.

- |   |                    |
|---|--------------------|
| 5 | Quadratic B-spline |
| 6 | Cubic B-spline     |

STARTUPTODAY

TODAY

(2000i, 2002)

NEW

Type: Integer

OPEN

Saved in: Register

Initial value: 1

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
```



|                                                                                                                                                                                                                                                                                                    |                                         |
|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|-----------------------------------------|
| <p><b>SURFTAB1</b><br/> Type: Integer<br/> Saved in: Drawing<br/> Initial value: 6<br/> 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.</p>         | <p><b>SURFTAB</b></p>                   |
| <p><b>SURFTAB2</b><br/> Type: Integer<br/> Saved in: Drawing<br/> Initial value: 6<br/> Sets the mesh density in the N direction for the REVSURF and EDGESURF commands. Values are 2-32766.</p>                                                                                                    | <p><b>SURFTAB</b></p>                   |
| <p><b>SURFTYPE</b><br/> Type: Integer<br/> Saved in: Drawing<br/> Initial value: 6<br/> Controls the type of surface-fitting to be performed by the Smooth option of the PEDIT command.</p> <p>5      Quadratic B-spline surface<br/> 6      Cubic B-spline surface<br/> 8      Bezier surface</p> | <p><b>PEDIT</b></p>                     |
| <p><b>SURFU</b><br/> Type: Integer<br/> Saved in: Drawing<br/> Initial value: 6<br/> Sets the surface density for PEDIT Smooth in the M direction. Values are 2-200.</p>                                                                                                                           | <p><i>3D/Draw</i><br/> <b>PEDIT</b></p> |
| <p><b>SURFV</b><br/> Type: Integer<br/> Saved in: Drawing<br/> Initial value: 6<br/> Sets the surface density for PEDIT Smooth in the N direction. Values are 2-200.</p>                                                                                                                           | <p><i>3D/Draw</i><br/> <b>PEDIT</b></p> |
| <p><b>SYSCODEPAGE</b><br/> (Read-only)<br/> Type: String<br/> Not saved<br/> Initial value: "ansi_1251"<br/> Indicates the system code page specified in the acad.xmx file.</p>                                                                                                                    | <p><i>Environment</i></p>               |
| <p><b>TABMODE</b><br/> Type: Integer<br/> Not saved</p>                                                                                                                                                                                                                                            | <p><i>Environment</i></p>               |

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  
Type: Integer  
Not saved  
Initial value: 0

TEXT  
DTEXT

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  
Type: Integer  
Saved in: Registry  
Initial value: 1

*Display*

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  
Type: Integer  
Not saved  
Initial value: 50

*Environment*

Sets the resolution tessellation 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  
Type: Real  
Saved in: Drawing  
Initial value: 0.2000

TEXT  
DTEXT

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  
Type: String  
Saved in: Drawing  
Initial value: "STANDARD"

TEXT  
DTEXT

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  
Saved in: Registry  
Initial value: 1  
Controls the display of tooltips.

TOOLBAR

- 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  
OBJECT SNAP  
TRACKING*  
Type: Integer  
Saved in: Registry  
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: Integer

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 they 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, OBSCURED COLOR, and OBSCURED LTYPE.

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

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### Statistics:

Total number of variables documented: 377

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