

**AutoCAD<sup>®</sup> 2000i**

**System Variables  
Commands & Descriptions**

**Quick Reference Guide**

AutoCAD® stores the values for its operating environment and some of its commands in system variables. Each system variable has an associated type: integer, real, point, switch, or text string. You can examine any system variable and change any writable system variable directly on the command line by entering the system variable name, or by using the SETVAR command or the AutoLISP® getvar and setvar functions. Many system variables are also accessible through dialog box options.

System variables control many AutoCAD functions and many aspects of the design environment. System variables control how many commands work. System variables also turn drawing modes (such as Snap and Grid) on or off, set default scales for hatch patterns, and store information about the current drawing and AutoCAD configuration.

System variables have abbreviated names that are usually 6-10 characters long. Many system variables have simple on or off settings. For example, for GRIDMODE, entering **0**, **off**, or **false** turns the grid off; entering **1**, **on**, or **true** turns the grid on. Other system variables store numbers or text: LIMMAX stores the upper-right drawing limits, for example, 12.0000, 9.000; DWGNAME is a read-only variable that stores the current drawing file name.

You generally change system variables through dialog box entries. However, you can change system variables directly on the command line, which is quicker in certain situations.

The following pages list in alphabetical order the system variables. This guide is for reference only and may include errors or omissions.

System Variable	Value	Description
ACADLSPASDOC	0	Loads <i>acad.lsp</i> into just the first drawing opened in an AutoCAD session.
	1	Loads <i>acad.lsp</i> into every drawing opened.
ACADPREFIX	READ ONLY	Stores the directory path, if any, specified by the ACAD environment variable, with path separators appended if necessary.
ACADVER	READ ONLY	Stores the AutoCAD version number. This variable differs from the DXF file \$ACADVER header variable, which contains the drawing database level number.
ACISOUTVER	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.
AFLAGS		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	0	Sets the base angle to 0 with respect to the current UCS.
ANGDIR		Sets the positive angle direction from angle 0 with respect to the current UCS.
	0	Counterclockwise
	1	Clockwise
APBOX		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

System Variable	Value	Description
APERTURE	10	Sets the display size for the aperture, in pixels. The aperture is the selection tool used in drawing commands. Initial Value is 10
AREA	READ ONLY	AREA is both a command and a system variable. The AREA system variable stores the last area computed by the AREA command. Because entering <b>area</b> at the Command prompt invokes the AREA command, you must use the SETVAR command to access the AREA system variable.
ATTDIA		Controls whether the -INSERT command uses a dialog box for attribute value entry.
	0	Issues prompts on the command line
ATTMODE	1	Uses a dialog box
		Controls display of attributes.
	0	Off: Makes all attributes invisible
ATTREQ	1	Normal: Retains current visibility of each attribute: visible attributes are displayed; invisible attributes are not
	2	On: Makes all attributes visible
		Determines whether the INSERT command uses default attribute settings during insertion of blocks.
AUDITCTL	0	Assumes the defaults for the values of all attributes
	1	Turns on prompts or dialog box for attribute values, as specified by ATTDIA
AUNITS		Controls whether the AUDIT command creates an audit report (ADT) file.
	0	Prevents writing of ADT files
AUNITS	1	Writes ADT files
		Sets units for angles.
	0	Decimal degrees
	1	Degrees/minutes/seconds
	2	Gradians
3	Radians	
4	Surveyor's units	

System Variable	Value	Description
AUPREC		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).
AUTOSNAP		Controls AutoSnap marker, tooltip, and magnet. Also turns on polar and object snap tracking, and controls the display of polar and object snap tracking tooltips. The system variable value is the sum of the following bit values:
	0	Turns off the AutoSnap marker, tooltips, and magnet. Also turns off polar tracking, object snap tracking, and tooltips for polar and object snap tracking.
	1	Turns on the AutoSnap marker
	2	Turns on the AutoSnap tooltips
	4	Turns on the AutoSnap magnet
	8	Turns on polar tracking
	16	Turns on object snap tracking
	32	Turns on tooltips for polar tracking and object snap tracking
BACKZ		Stores the back clipping plane offset from the target plane for the current viewport, in drawing units. Meaningful only if the back clipping bit in VIEWMODE is on. The distance of the back clipping plane from the camera point can be found by subtracting BACKZ from the camera-to-target distance.
BINDTYPE		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		Controls whether marker blips are visible.
	0	Turns off marker blips
	1	Turns on marker blips
CDATE		Sets calendar date and time

System Variable	Value	Description
CECOLOR	Bylayer	Sets the color of new objects
CELTSCALE	1	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	Bylayer	Sets the linetype of new objects
CELWEIGHT		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 hundredths of 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 hundredths of millimeters. (Multiply a value by 2540 to convert values from inches to hundredths of millimeters.)
CHAMFERA		Sets the first chamfer distance.
CHAMFERB		Sets the second chamfer distance.
CHAMFERC		Sets the chamfer length.
CHAMFERD		Sets the chamfer angle.
CHAMMODE		Sets the input method by which AutoCAD creates chamfers.
	0	Requires two chamfer distances
	1	Requires one chamfer distance and an angle
CIRCLERAD		Sets the default circle radius. A zero indicates no default.
CLAYER		Sets the current layer.
CMDACTIVE		Stores the bitcode that indicates whether an ordinary command, transparent command, script, or dialog box is active. The value is the sum of the following:
	1	Ordinary command is active
	2	Ordinary command and a transparent command are active
	4	Script is active

System Variable	Value	Description
	8	Dialog box is active
	16	AutoLISP is active (only visible to an ObjectARX-defined command)
CMDDIA		<p>When you start a new drawing, options are displayed on the command line, instead of in a dialog box. This behavior occurs even though the system variables CMDDIA and FILEDIA are both set to 1.</p> <p>For the NEW command to display a dialog box, rather than command line prompts, you need to select the Show Startup dialog option in the Options dialog box.</p> <ol style="list-style-type: none"> <li>1. From the Tools menu, choose Options.</li> <li>2. In the Options dialog box, choose the System tab.</li> <li>3. In the General Options area, select the Show Startup dialog option.</li> <li>4. Choose OK to apply the changes and close the Options dialog box.</li> </ol>
CMDECHO		Controls whether AutoCAD echoes prompts and input during the AutoLISP command function.
	0	Turns off echoing
	1	Turns on echoing
CMDNAMES		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.
CMLJUST		Specifies multiline justification.
	0	Top
	1	Middle
	2	Bottom
CMLSCALE		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).

<b>System Variable</b>	<b>Value</b>	<b>Description</b>
CMLSTYLE		Sets the multiline style that AutoCAD uses to draw the multiline.
COMPASS		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		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		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
CPROFILE		Displays the name of the current profile. For more information on profiles, see the OPTIONS command.
CTAB		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		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.

System Variable	Value	Description
CVPORT		<p>Sets the identification number of the current viewport. You can change this value, which changes the current viewport, if the following conditions are met:</p> <ol style="list-style-type: none"> <li>1. The identification number you specify is that of an active viewport.</li> <li>2. A command in progress has not locked cursor movement to that viewport.</li> <li>3. Tablet mode is off.</li> </ol>
DATE		<p>Stores the current date and time represented as a Julian date and fraction in a real number: For example, on January 29, 1993, at 2:29:35 in the afternoon, the DATE variable would contain 2446460.603877364. DATE returns a true Julian date only if the system clock is set to UTC/Zulu (Greenwich Mean Time). TDCREATE and TDUPDATE have the same format as DATE, but their values represent the creation time and last update time of the current drawing.</p>
DBMOD	<p>1 4 8 16</p>	<p>Indicates the drawing modification status using bitcode. The value is the sum of the following:</p> <ul style="list-style-type: none"> <li>1 Object database modified</li> <li>4 Database variable modified</li> <li>8 Window modified</li> <li>16 View modified</li> </ul> <p>AutoCAD resets the DBMOD value to 0 when you save the drawing.</p>
DCTCUST		<p>Displays the path and file name of the current custom spelling dictionary.</p>
DCTMAIN		<p>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 <i>support</i> directory.</p> <p>You can specify a default main spelling dictionary using the SETVAR command. When prompted for a new value for DCTMAIN, you can enter one of the keywords below. Depending on the language version of AutoCAD, not all dictionaries may be available.</p>

<b>System Variable</b>	<b>Value</b>	<b>Description</b>
DEFLPLSTYLE		Specifies the default plot style for new layers. If the current drawing you are working in is in color-dependent mode (PSTYLEPOLICY is set to 1), DEFLPLSTYLE is read-only and has a value of "ByColor". If the current drawing you are working in is in named plot styles mode (PSTYLEPOLICY is set to 0), DEFLPLSTYLE is writable and has a default value of "Normal". To convert the current drawing to use named or color-dependent plot styles, use the CONVERTPSTYLES utility provided with the AutoCAD 2000i Migration Assistance.
DEFPLSTYLE		Specifies the default plot style for new objects. If the current drawing you are working in is in color-dependent mode (PSTYLEPOLICY is set to 1), DEFPLSTYLE is read-only and has a value of "ByColor". If the current drawing you are working in is in named plot styles mode (PSTYLEPOLICY is set to 0), DEFPLSTYLE is writable and has a default value of "ByLayer". To convert the current drawing to use named or color-dependent plot styles, use the CONVERTPSTYLES utility provided with the AutoCAD 2000i Migration Assistance.
DELOBJ		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		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.

System Variable	Value	Description
	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.
DIASAT		Stores the exit method of the most recently used dialog box.
	0	Cancel
	1	OK
DIMADEC		Controls the number of precision places displayed in angular dimensions. 0-8 Indicates the number of decimal places to display in angular dimensions.
DIMALT		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		Controls the number of decimal places in alternate units.
DIMALTF		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		Rounds off the alternate dimension units.
DIMALTTD		Sets the number of decimal places for the tolerance values in the alternate units of a dimension.
DIMALTTZ		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

System Variable	Value	Description
DIMALTU		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
DIMALTZ	8	Windows® Desktop (decimal format using Control Panel settings for decimal separator and number grouping symbols)
		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		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		Controls the associativity of dimension objects.
	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.

System Variable	Value	Description
	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.
DIMASZ		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		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		Sets the units format for angular dimensions.
	0	Decimal degrees
	1	Degrees/minutes/seconds
	2	Gradians
	3	Radians DIMAUNIT sets this value when entered on the command line or when set from the Primary Units area in the Annotation dialog box.
DIMAZIN		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)

System Variable	Value	Description
DIMBLK	<p>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.</p> <p>"" 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</p>	
DIMBLK1		<p>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.</p>
DIMBLK2		<p>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</p>
DIMCEN	0	<p>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.</p> <p>No center marks or lines are drawn</p>

System Variable	Value	Description
	<0	Centerlines are drawn
	>0	Center marks are drawn
DIMCLRD		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		Assigns colors to dimension extension lines. The color can be any valid color number. See DIMCLRD.
DIMCLRT		Assigns colors to dimension text. The color can be any valid color number. See DIMCLRD.
DIMDEC		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		Sets the distance the dimension line extends beyond the extension line when oblique strokes are drawn instead of arrowheads.
DIMDLI		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		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		Specifies how far to extend the extension line beyond the dimension line.

System Variable	Value	Description
DIMEXO		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		Obsolete. Has no effect in AutoCAD 2000 and later releases except to preserve the integrity of pre-AutoCAD 2000 scripts and AutoLISP routines. DIMFIT is replaced by DIMATFIT and DIMTMOVE.
DIMFRAC		Sets the fraction format when DIMLUNIT is set to 4 (Architectural) or 5 (Fractional).
	0	Horizontal
	1	Diagonal
	2	Not stacked (for example, 1/2)
DIMGAP		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		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

System Variable	Value	Description
DIMLDRBLK		Specifies the arrow type for leaders. To turn off arrowhead display, enter a single period (.). For a list of arrowhead entries, see DIMBLK.
DIMLFAC		<p>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.</p> <p>DIMLFAC has no effect on angular dimensions, and is not applied to the values held in DIMRND, DIMTM, or DIMTP.</p> <p>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.</p> <p>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.</p> <p>Dim: <b>dimlfac</b></p> <p>Current value &lt;1.0000&gt; New value (Viewport): <b>v</b></p> <p>Select viewport to set scale:</p> <p>AutoCAD calculates the scaling of model space to paper space and assigns the negative of this value to DIMLFAC.</p>
DIMLIM	<p>OFF</p> <p>ON</p>	<p>Generates dimension limits as the default text. Setting DIMLIM to on turns DIMTOL off.</p> <p>Dimension limits are not generated as default text</p> <p>Dimension limits are generated as default text</p>
DIMLUNIT	<p>1</p> <p>2</p> <p>3</p> <p>4</p> <p>5</p> <p>6</p>	<p>Sets units for all dimension types except Angular.</p> <p>Scientific</p> <p>Decimal</p> <p>Engineering</p> <p>Architectural</p> <p>Fractional</p> <p>Windows desktop</p>
DIMLWD		Assigns lineweight to dimension lines. Values are standard lineweights.

System Variable	Value	Description
	-3	BYLAYER
	-2	BYBLOCK
		integer representing 100th of mm
DIMLWE		Assigns lineweight to extension lines. Values are standard lineweights.
	-3	BYLAYER
	-2	BYBLOCK
		integer representing 100th of mm
DIMPOST		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		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		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		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.

System Variable	Value	Description
	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		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		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		Suppresses display of the first extension line.
	OFF	Extension line is not suppressed
	ON	Extension line is suppressed
DIMSE2		Suppresses display of the second extension line.
	OFF	Extension line is not suppressed
	ON	Extension line is suppressed
DIMSHO		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.
DIMSOXD		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.

System Variable	Value	Description
DIMSTYLE		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	0	Controls the vertical position of text in relation to the dimension line. 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		Sets the number of decimal places to display in tolerance values for the primary units in a dimension.
DIMTFAC		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		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		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.

System Variable	Value	Description
	ON	Draws dimension text between the extension lines even if AutoCAD ordinarily places it outside those lines.
DIMTM		<p>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.</p> <p>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).</p>
DIMTMOVE		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		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		Controls the position of dimension text outside the extension lines.
	OFF	Aligns text with the dimension line
	ON	Draws text horizontally
DIMTOL		Appends tolerances to dimension text. Setting DIMTOL to on turns DIMLIM off.

System Variable	Value	Description
DIMTOLJ		Sets the vertical justification for tolerance values relative to the nominal dimension text.
	0	Bottom
	1	Middle
	2	Top
DIMTP		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	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		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		Specifies the text style of the dimension.
DIMTXT		Specifies the height of dimension text, unless the current text style has a fixed height.
DIMTZIN		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

System Variable	Value	Description
	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 0.5)
DIMUNIT		Obsolete. Has no effect in AutoCAD 2000 and later releases except to preserve the integrity of pre-AutoCAD 2000 scripts and AutoLISP routines. DIMUNIT is replaced by DIMLUNIT and DIMFRAC.
DIMUPT		Controls options for user-positioned text.
	OFF	Cursor controls only the dimension line location
	ON	Cursor controls both the text position and the dimension line location
DIMZIN		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 0.5)
		DIMZIN also affects real-to-string conversions performed by the AutoLISP rtos and angtos functions.
DISPSILH		Controls display of silhouette curves of solid objects in Wireframe mode. Also controls whether mesh is drawn or suppressed when a solid object is hidden.
	0	Off
	1	On

System Variable	Value	Description
DISTANCE		Stores the distance computed by the DIST command.
DONUTID		Sets the default for the inside diameter of a donut.
DONUTOD		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		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 <b>drag</b> on the command line after selecting the object to drag
	2	Auto; always displays an outline of the object as you drag it
DRAGP1		Sets the regen-drag input sampling rate.
DRAGP2		Sets the fast-drag input sampling rate.
DWGCHECK		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		Stores the same value as SYSCODEPAGE (for compatibility reasons).
DWGNAME		Stores the drawing name as entered by the user. If the drawing has not been named yet, DWGNAME defaults to "Drawing.dwg." If the user specified a drive/directory prefix, the prefix is stored in DWGPREFIX.
DWGPREFIX		Stores the drive/directory prefix for the drawing.
DWGTITLED		Indicates whether the current drawing has been named.
	0	Drawing has not been named
	1	Drawing has been named
EDGEMODE		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

System Variable	Value	Description
		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		Stores the current elevation relative to the current UCS for the current viewport in the current space.
EXPERT		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 <b>y</b> at the prompt. Setting EXPERT can affect scripts, menu macros, AutoLISP, and the command functions.
EXPLMODE		Controls whether the EXPLODE command supports nonuniformly scaled (NUS) blocks.
	0	Does not explode NUS blocks
	1	Explodes NUS blocks

System Variable	Value	Description
EXTMAX		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		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		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 (and later releases) 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		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 <i>N</i> by 1 mesh for cylindrical and conic ACIS solids
	1	Creates an <i>N</i> by <i>M</i> mesh for cylindrical and conic ACIS solids
FACETRES		Adjusts the smoothness of shaded and rendered objects and objects with hidden lines removed. Valid values are from 0.01 to 10.0.
FILEDIA		Suppresses display of the file dialog boxes.
	0	Does not display dialog boxes. You can still request a file dialog box to appear by entering a tilde (~) in response to the command's prompt. The same is true for AutoLISP and ADS functions.

System Variable	Value	Description
	1	Displays file dialog boxes. However, if a script or AutoLISP/ObjectARX™ program is active, AutoCAD displays an ordinary prompt.
FILLETRAD		Stores the current fillet radius.
FILLMODE		Specifies whether multilines, traces, solids, all hatches (including solid-fill), and wide polylines are filled in.
	0	Objects are not filled
	1	1 Objects are filled
FONTALT		<p>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.</p> <p>The dialog box is displayed in the following cases:</p> <ol style="list-style-type: none"> <li>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.</li> <li>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.</li> <li>3. The Browse button is pressed in the Options dialog box when you specify an alternate font.</li> </ol> <p>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 <i>timebd.ttf</i>). 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.</p>

System Variable	Value	Description
FONTMAP		<p>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</p> <p>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.</p>
FRONTZ		<p>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.</p>
FULLOPEN		<p>Indicates whether the current drawing is partially open.</p>
	0	Indicates a partially open drawing
	1	Indicates a fully open drawing
GRIDMODE		<p>Specifies whether the grid is turned on or off.</p>
	0	Turns the grid off
	1	Turns the grid on
GRIDUNIT		<p>Specifies the grid spacing (X and Y) for the current viewport.</p>
GRIPBLOCK		<p>Controls the assignment of grips in blocks.</p>
	0	Assigns a grip only to the insertion point of the block
	1	Assigns grips to objects within the block
GRIPCOLOR		<p>Controls the color of nonselected grips (drawn as box outlines). The valid range is 1 to 255.</p>

<b>System Variable</b>	<b>Value</b>	<b>Description</b>
GRIPHOT		Controls the color of selected grips (drawn as filled boxes). The valid range is 1 to 255.
GRIPS		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		Sets the size of the grip box in pixels. The valid range is 1 to 255.
HANDLES		Reports whether object handles can be accessed by applications.
HIDEPRECISION		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.
	0	Single precision; uses less memory
	1	Double precision; uses more memory
HIGHLIGHT		Controls object highlighting; does not affect objects selected with grips.
	0	Turns off object selection highlighting
	1	Turns on object selection highlighting
HPANG		Specifies the hatch pattern angle.
HPBOUND		Controls the object type created by the BHATCH and BOUNDARY commands.
	0	Creates a region
	1	Creates a polyline
HPDOUBLE		Specifies hatch pattern doubling for user-defined patterns.
	0	Turns off hatch pattern doubling
	1	Turns on hatch pattern doubling

<b>System Variable</b>	<b>Value</b>	<b>Description</b>
HPNAME		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		Specifies the hatch pattern scale factor, which must be nonzero.
HPSPACE		Specifies the hatch pattern line spacing for user-defined simple patterns, which must be nonzero.
HYPERLINKBASE		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		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		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		Stores the Internet location used by the BROWSER command and the Browse the Web dialog box.
INSBASE		Stores the insertion base point set by BASE, which gets expressed as a UCS coordinate for the current space.
INSNAME		Sets a default block name for the INSERT command. The name must conform to symbol naming conventions. Returns "" if no default is set. Enter a period (.) to set no default.
INSUNITS		Specifies a drawing units value for blocks or images inserted from AutoCAD DesignCenter™. INSUNITS applies to blocks and images only, and does not affect other content, including drawings.
	0	Unspecified (No units)
	1	Inches
	2	Feet

System Variable	Value	Description
	3	Miles
	4	Millimeters
	5	Centimeters
	6	Meters
	7	Kilometers
	8	Microinches
	9	Mils
	10	Yards
	11	Angstroms
	12	Nanometers
	13	Microns
	14	Decimeters
	15	Decameters
	16	Hectometers
	17	Gigameters
	18	Astronomical Units
	19	Light Years
	20	Parsecs
INSUNITSDEFSOURCE		Sets source content units value. Valid range is 0-20.
INSUNITSDEFTARGET		Sets target drawing units value. Valid range is 0-20.
ISAVEBAK		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.

System Variable	Value	Description
ISAVEPERCENT		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.
ISOLINES		Specifies the number of isolines per surface on objects. Valid integer values are from 0 to 2047.
LASTANGLE		Stores the end angle of the last arc entered relative to the XY plane of the current UCS for the current space.
LASTPOINT		Stores the last point entered, expressed as a UCS coordinate for the current space; referenced by the @ symbol (@) during keyboard entry.
LASTPROMPT		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.
LAYOUTREGENCTL		Specifies how the display list is updated in the Model tab and layout tabs. For each tab, the display list is updated either by regenerating the drawing when you switch to that tab or by saving the display list to memory and regenerating only the modified objects when you switch to that tab. Changing the LAYOUTREGENCTL setting can improve performance.
	0	The drawing is regenerated each time you switch tabs.
	1	For the Model tab and the last layout made current, the display list is saved to memory and regenerations are suppressed when you switch between the two tabs. For all other layouts, regenerations still occur when you switch to those tabs.

System Variable	Value	Description
	2	<p>The drawing is regenerated the first time you switch to each tab. For the remainder of the drawing session, the display list is saved to memory and regenerations are suppressed when you switch to those tabs.</p> <p>The performance gain achieved by changing the LAYOUTREGENCTL setting is dependent on several factors, including the drawing size and type, the objects contained in the drawing, the amount of available memory, and the effect of other open drawings or applications. When LAYOUTREGENCTL is set to 1 or 2, the amount of additional memory used is the size of the Model tab's display list multiplied by the number of viewports in each layout for which the display list is saved.</p> <p>If LAYOUTREGENCTL is set to 1 or 2 and performance seems slow in general or when you switch between tabs for which the display list is saved, consider changing to a setting of 0 or 1 to find the optimal balance for your work environment.</p> <p>NOTE Regardless of the LAYOUTREGENCTL setting, if you redefine a block or undo a tab switch, the drawing is regenerated the first time you switch to any tab that contains saved viewports.</p>
LENSLENGTH		Stores the length of the lens (in millimeters) used in perspective viewing for the current viewport.
LIMCHECK		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		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.

<b>System Variable</b>	<b>Value</b>	<b>Description</b>
LIMMIN		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		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
		When SDI is set to 0, AutoLISP always behaves as if LISPINIT were set to 1.
LOCALE	READ ONLY	Displays the International Standards Organization (ISO) language code of the current AutoCAD version you're running.
LOGFILEMODE		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	READ ONLY	Specifies the path and name of the log file for the current drawing. The initial value varies depending on the name of the current drawing and where you installed AutoCAD.
LOGFILEPATH		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.
LOGINAME	READ ONLY	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		Sets the global linetype scale factor. The linetype scale factor cannot equal zero.

<b>System Variable</b>	<b>Value</b>	<b>Description</b>
LUNITS		Sets linear units.
	1	Scientific
	2	Decimal
	3	Engineering
	4	Architectural
	5	Fractional
LUPREC		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		Sets the value for the default lineweight. The default lineweight can be set to any valid lineweight value in hundredths of 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 hundredths of millimeters. (Multiply a value by 2540 to convert values from inches to hundredths of millimeters.)
LWDISPLAY		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		Controls whether lineweight units are displayed in inches or millimeters.
		0 Inches
		1 Millimeters
MAXACTVP		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.

System Variable	Value	Description
MAXACTVP		<p>Occasionally as a drawing contains many viewports in paper space, older viewports will not display or will be blank. The MAXACTVP variable has an effect on controls how many viewports can be displayed at one time.</p> <p>The default setting for the MAXACTVP is 64; this allows 64 active viewports to be displayed at one time.</p> <p><b>Note:</b> Increasing the number of active viewports that are displayed can negatively impact performance.</p>
MAXSORT		<p>Sets the maximum number of symbol names or block names sorted by listing commands. If the total number of items exceeds the value of MAXSORT, no items are sorted. To resolve the problem, the value for MAXSORT must exceed the number items to sort.</p>
MBUTTONPAN		<p>Controls the behavior of the third button or wheel on the pointing device.</p>
	0	<p>Supports the action defined in the AutoCAD menu (.mnu) file.</p>
	1	<p>Supports panning by holding and dragging the button or wheel.</p>
MEASUREINIT		<p>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.</p>
	0	<p>English; AutoCAD uses the hatch pattern file and linetype file designated by the ANSIHatch and ANSILinetype registry settings.</p>
	1	<p>Metric; AutoCAD uses the hatch pattern file and linetype file designated by the ISOHatch and ISOLinetype registry settings.</p>
MEASUREMENT		<p>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.</p>

<b>System Variable</b>	<b>Value</b>	<b>Description</b>
	0	English; AutoCAD uses the hatch pattern file and linetype file designated by the ANSIHatch and ANSILinetype registry settings.
	1	Metric; AutoCAD uses the hatch pattern file and linetype file designated by the ISOHatch and ISOLinetype registry settings.
		The drawing units for new drawings are controlled by MEASUREINIT (MEASUREINIT uses the same values as MEASUREMENT). The MEASUREMENT setting of a drawing always overrides the MEASUREINIT setting.
MENUCTL		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		Sets menu echo and prompt control bits. The value is the sum of the following:
	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	READ ONLY	Stores the menu file name, including the path for the file name.
MIRRTEXT		Controls how the MIRROR command reflects text.
	0	Retains text direction
	1	Mirrors the text
MODEMACRO		Displays a text string on the status line, such as the name of the current drawing, time/date stamp, or special modes.

System Variable	Value	Description
		Use MODEMACRO to display a string of text, or use special text strings written in the DIESEL macro language to have AutoCAD evaluate the macro from time to time and base the status line on user-selected conditions.
MTEXTED		Sets the primary and secondary text editors to use for multiline text objects. The default setting is Internal, which calls the internal MTEXT editor. If the mtext object is fewer than 80 characters you can specify :lisped to use the lisp editor.
NOMUTT		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	Suppresses muttering indefinitely
OFFSETDIST		Sets the default offset distance.
	<0	Offsets an object through a specified point
	>0	Sets the default offset distance
OFFSETGAPTYPE		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		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.

<b>System Variable</b>	<b>Value</b>	<b>Description</b>
OLEQUALITY		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
OLESTARTUP	4	High quality photograph
		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
ORTHOMODE	1	Loads the OLE source application when plotting
		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.
ORTHOMODE	0	Turns off Ortho mode
	1	Turns on Ortho mode
OSMODE		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	PER-
	256	TANgent
	512	512 NEArest
	1024	1024 QUIck
	2058	2048 APParent Intersection
4096	4096 EXTension	
8192	8192 PARallel	

System Variable	Value	Description
		<p>To specify more than one object snap, enter the sum of their values. For example, entering <b>3</b> specifies the Endpoint (bitcode 1) and Midpoint (bitcode 2) object snaps. Entering <b>16383</b> specifies all object snaps.</p> <p>When object snaps are switched off using the Osnap button on the status bar, a bitcode of 16384 (0x4000) is returned, in addition to the normal value of OSMODE. With this additional value developers can write applications for AutoCAD, and distinguish this mode from Object Snap modes that have been turned off from within the Drafting Settings dialog box. Setting this bit toggles running object snaps off. Setting OSMODE to a value with this bit off toggles running object snaps on.</p>
OSNAPCOORD		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		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		Controls how point objects and nodes are displayed. For information about values to enter, see the POINT command.
	0	Display on
	1	Display off
PDSIZE		Sets the display size for point objects.
	0	Creates a point at 5 percent of the drawing area height

System Variable	Value	Description
PELLIPSE	>0	Specifies an absolute size
	<0	Specifies a percentage of the viewport size
		AutoCAD R14, AutoCAD 2000, and AutoCAD 2000i can create true ellipses. Previous releases of AutoCAD created ellipses from polylines. You can set the PELLIPSE system variable to control the type of ellipse that AutoCAD creates: a true ellipse or an ellipse created from polylines.
	0	All ellipses are created as true ellipses (default).
	1	All ellipses are created as polylines.
PERIMETER	READ ONLY	Stores the last perimeter value computed by the AREA, DBLIST, or LIST commands.
PFACEVMAX	READ ONLY	Sets the maximum number of vertices per face.
PICKADD		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		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		Sets the object selection target height, in pixels
PICKDRAG		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.

<b>System Variable</b>	<b>Value</b>	<b>Description</b>
PICKFIRST	1	Draws the selection window using dragging. Click one corner and drag the pointing device; release the button at the other corner.
	0	Turns off PICKFIRST; you select objects after you issue a command
PICKSTYLE	1	Turns on PICKFIRST; you select objects before you issue a command
		Controls the use of group selection and associative hatch selection.
PICKSTYLE	0	No group selection or associative hatch selection
	1	Group selection
	2	Associative hatch selection
	3	Group selection and associative hatch selection
PLATFORM	READ ONLY	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		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
PLINEGEN	1	Generates the linetype in a continuous pattern around the vertices of the polyline
		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.
PLINETYPE		
	0	Polylines in older drawings are not converted when opened; PLINE creates old-format polylines

System Variable	Value	Description
	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
PLINEWID		Stores the default polyline width.
PLOTID		Obsolete. Has no effect in AutoCAD 2000 and later releases except to preserve the integrity of pre-AutoCAD 2000 scripts and LISP routines.
PLOTROTMODE		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		Obsolete. Has no effect in AutoCAD 2000 and later releases except to preserve the integrity of pre-AutoCAD 2000 scripts and LISP routines.
PLQUIET		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

System Variable	Value	Description
POLARADDANG		Contains user-defined polar angles. You can add up to 10 angles. Each angle can be up to 25 characters, separated with semicolons (;). AutoCAD displays angles in the format set in the AUNITS system variable.
POLARANG		Sets the polar angle increment. Values are 90, 45, 30, 22.5, 18, 15, 10, and 5.
POLARDIST		Sets the snap increment when the SNAPSTYL system variable is set to 1 (polar snap).
POLARMODE		Controls settings for polar and object snap tracking. The value is the sum of four bitcodes:
		Polar angle measurements
	0	Measure polar angles based on current UCS (absolute)
	1	Measure polar angles from selected objects (relative)
		Object snap tracking
	0	Track orthogonally only
	2	Use polar tracking settings in object snap tracking
		Use additional polar tracking angles
	0	No
	4	Yes
		Acquire object snap tracking points
	0	Acquire automatically
	8	Press SHIFT to acquire
POLYSIDES		Sets the default number of sides for the POLYGON command. The range is 3 to 1024.
POPUPS		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

<b>System Variable</b>	<b>Value</b>	<b>Description</b>
PROJECTNAME		<p>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.</p> <p>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.</p> <p>If the xref or image is not found at the original path, the project paths associated with the project name are searched. If the xref or image is not found there, the AutoCAD search path is searched.</p>
PROJMODE		<p>Sets the current Projection mode for trimming or extending.</p>
	0	True 3D mode (no projection)
	1	Project to the XY plane of the current UCS
	2	Project to the current view plane
PROXYGRAPHICS		<p>Specifies whether images of proxy objects are saved in the drawing.</p>
	0	Does not save image with the drawing; a bounding box is displayed instead
	1	Saves image with the drawing
PROXYNOTICE		<p>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.</p>
	0	No proxy warning is displayed
	1	Proxy warning is displayed
PROXYSHOW		<p>Controls the display of proxy objects in a drawing.</p>
	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

System Variable	Value	Description
PROXYWEBSEARCH		Specifies how AutoCAD checks for Object Enablers. Object Enablers allow you to display and use custom objects in AutoCAD drawings even when the ObjectARX application that created them is unavailable. PROXYWEBSEARCH is also controlled with the Live Enabler options on the System tab of the Options dialog box.
	0	Prevents AutoCAD from checking for Object Enablers regardless of your settings in the Today window (see TODAY).
	1	AutoCAD checks for Object Enablers only if Autodesk Point A is open in the Today window. It is not necessary for the Today window to be open. However, AutoCAD checks for Object Enablers only if a live Internet connection is present.
PSLTSCALE	2	Specifies the number of times AutoCAD will continue to check for Object Enablers after unsuccessful attempts.
		Controls paper space linetype scaling.
	0	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	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		Unknown

<b>System Variable</b>	<b>Value</b>	<b>Description</b>
PSQUALITY		Unknown
PSTYLEMODE		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		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		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		Stores the name of the UCS that defines the origin and orientation of orthographic UCS settings in paper space only.
QTEXTMODE		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		Controls whether BMP preview images are saved with the drawing.
	0	No preview image is created
	1	Preview image created
REFEDITNAME	READ ONLY	Indicates whether a drawing is in a reference-editing state; also, stores the reference file name.

<b>System Variable</b>	<b>Value</b>	<b>Description</b>
REGENMODE	0	0 Turns off the REGENAUTO command
	1	1 Turns on the REGENAUTO command
REMEMBERFOLDERS	0	Controls the default path for the Look In or Save In option in standard file selection dialog boxes.  This setting restores the legacy behavior of AutoCAD 2000 and previous releases. When you start AutoCAD by double-clicking an AutoCAD 2000i icon, if a Start In path is specified for the icon, that path is used as the default for all standard file selection dialog boxes. (To change the Start In setting, right-click the AutoCAD 2000i icon on the Desktop and choose Properties. In the AutoCAD 2000i Properties dialog box, choose the Shortcut tab and specify a new Start In value.)
	1	The last used paths in each particular standard file selection dialog box are remembered across and within AutoCAD sessions. The Start In folder specified for the AutoCAD 2000i icon is not used.
RTDISPLAY	0	Controls the display of raster images during Realtime ZOOM or PAN.  Displays raster image content
	1	Displays raster image outline only  RTDISPLAY is saved in the current profile.
SAVEFILE	READ ONLY	Stores the current automatic save file name.
SAVEFILEPATH		Sets the path to the folder where all Autosave files are stored for the current AutoCAD session.
SAVENAME	READ ONLY	Stores the file name and directory path of the current drawing after you save it.
SAVETIME	0	Sets the automatic save interval, in minutes.  Turns off automatic saving
	>0	Saves the drawing at intervals specified by the nonzero integer automatically

<b>System Variable</b>	<b>Value</b>	<b>Description</b>
		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	READ ONLY	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	READ ONLY	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	READ ONLY	Stores current viewport size in pixels (X and Y).
SDI		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.

System Variable	Value	Description
SHADEDGE		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		Sets the ratio of diffuse reflective light to ambient light in percentage of diffuse reflective light, when SHADEDGE is set to 0 or 1.
SHORTCUTMENU		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 <b>3</b> enables both Default (1) and Edit (2) mode shortcut menus.
SHPNAME		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		Sets the record increment for the SKETCH command.
SKPOLY		Determines whether the SKETCH command generates lines or polylines.
	0	Generates lines
	1	Generates polylines
SNAPANG		Sets the snap and grid rotation angle for the current viewport. The angle you specify is relative to the current UCS.

System Variable	Value	Description
SNAPBASE		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.
		Sets the snap and grid origin point for the current viewport relative to the current UCS.
SNAPISOPAIR		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.
		Controls the isometric plane for the current viewport.
SNAPMODE	0	Left
	1	Top
	2	Right
SNAPMODE		Turns the Snap mode on and off.
SNAPMODE	0	Snap off
	1	Snap on for the current viewport
SNAPSTYL		Sets the snap style for the current viewport.
SNAPSTYL	0	Standard (rectangular snap)
	1	Isometric snap
SNAPTTYPE		Sets the snap style for the current viewport.
SNAPTTYPE	0	Grid, or standard snap.
	1	Polar snap. Snaps along polar angle increments. Use polar snap with polar and object snap tracking.
SNAPUNIT		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		Turns the solid validation on and off for the current AutoCAD session.
	0	Turns off solid validation
	1	Turns on solid validation

System Variable	Value	Description
SORTENTS		Controls the OPTIONS command (from the User Preferences tab) object sort order operations. SORTENTS uses the following bitcodes:
	0	Disables SORTENTS
	1	Sorts for object selection
	2	Sorts for object snap
	4	Sorts for redraws
	8	Sorts for MSLIDE command slide creation
	16	Sorts for REGEN commands
	32	Sorts for plotting
	64	Sorts for PostScript output
		To select more than one setting, enter the sum of the settings' codes. For example, enter <b>3</b> to specify sorting for both object selection and object snap.
		The initial value of 96 enables sorting for plotting and PostScript output only. Setting additional sorting options can result in slower regeneration and redrawing times.
SPLFRAME		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		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.

System Variable	Value	Description
SPLINETYPE		Sets the type of curve generated by the Spline option of the PEDIT command.
	5	Quadratic B-spline
	6	Cubic B-spline
STARTUPTODAY		Controls whether the Today window or the traditional startup dialog box is displayed when starting AutoCAD or creating a new drawing.
	0	Displays the traditional startup dialog box
	1	Displays the Today window
SURFTAB1		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.
SURFTAB2		Sets the mesh density in the N direction for the REVSURF and EDGESURF commands.
SURFTYPE		Controls the type of surface-fitting to be performed by the Smooth option of the PEDIT command.
	5	Quadratic B-spline surface
	6	Cubic B-spline surface
	8	Bezier surface
SURFU		Sets the surface density for PEDIT Smooth in the M direction.
SURFV		Sets the surface density for PEDIT Smooth in the N direction.
SYSCODEPAGE	READ ONLY	Indicates the system code page specified in the <i>acad.xml</i> file. Codes are as follows: ascii dos860 dos932 iso8859-7 big5 dos861 gb2312 iso8859-8 dos437 dos863 iso8859-1 iso8859-9 dos850 dos864 iso8859-2 johab dos852 dos865 iso8859-3 ksc5601 dos855 dos866 iso8859-4 mac-roman dos857 dos869 iso8859-6
TABMODE		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

<b>System Variable</b>	<b>Value</b>	<b>Description</b>
TARGET	READ ONLY	Stores the location (as a UCS coordinate) of the target point for the current viewport.
TDCREATE	READ ONLY	Stores the local time and date the drawing was created.
TDINDWG	READ ONLY	Stores the total editing time.
TDUCREATE	READ ONLY	Stores the universal time and date the drawing was created.
TDUPDATE	READ ONLY	Stores the local time and date of the last update/save.
TDUSRTIMER	READ ONLY	Stores the user-elapsed timer.
TDUUPDATE	READ ONLY	Stores the universal time and date of the last update/save.
TEMPPREFIX	READ ONLY	Contains the directory name (if any) configured for placement of temporary files, with a path separator appended.
TEXTEVAL		Controls the method of evaluation of text strings entered with the TEXT or TEXT commands.
	0	All responses to prompts for text strings and attribute values are taken literally
	1	All text starting from 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 unless it is executed completely with a script or AutoLISP expression. The TEXT command honors the setting of TEXTEVAL.
TEXTFILL		Controls the filling of TrueType fonts while plotting and rendering.
	0	Outputs text as outlines
	1	Outputs text as filled images

<b>System Variable</b>	<b>Value</b>	<b>Description</b>
TEXTQLTY		Sets the resolution tessellation fineness of text outlines for TrueType fonts while plotting and rendering. 0 represents no effort to refine the smoothness of the text; 100 represents a maximum effort to smooth text characters. Lower values decrease resolution and increase plotting speed. Higher values increase resolution and decrease plotting speed.
TEXTSIZE		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		Sets the name of the current text style.
THICKNESS		Sets the current 3D solid thickness.
TILEMODE		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		Controls the display of tooltips.
	0	Turns off the display of tooltips
	1	Turns on the display of tooltips
TRACEWID		Sets the default trace width.
TRACKPATH		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		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.

System Variable	Value	Description
	>0	Turns on spatial indexing. 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.
TREEMAX		<p>Limits memory consumption during drawing regeneration by limiting the number of nodes in the spatial index (oct-tree).</p> <p>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.</p> <p>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.</p> <p>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.</p> <p>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.</p>
TRIMMODE	0	<p>Controls whether AutoCAD trims selected edges for chamfers and fillets.</p> <p>Leaves selected edges intact</p>

System Variable	Value	Description
	1	Trims selected edges to the endpoints of chamfer lines and fillet arcs
TSPACEFAC		Controls the multiline text line spacing distance measured as a factor of text height. Valid values are 0.25 to 4.0.
TSPACETYPE		Controls the type of line spacing used in multiline text. At Least adjusts line spacing based on tallest characters in a line. Exactly uses the specified line spacing, regardless of individual character sizes.
	1	At Least
	2	Exactly
TSTACKALIGN		Controls the vertical alignment of stacked text.
	0	Bottom aligned
	1	Center aligned
	2	Top aligned
TSTACKSIZE		Controls the percentage of stacked text fraction height relative to selected text's current height. Valid values are from 25 to 125.
UCSAXISANG		Unkown
UCSBASE		Stores the name of the UCS that defines the origin and orientation of orthographic UCS settings. Valid values include any named UCS.
UCSFOLLOW		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

System Variable	Value	Description
UCSICON		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.
		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 <code>ucsicon</code> at the Command prompt invokes the UCSICON command, you must use the SETVAR command to access the UCSICON system variable.
UCSNAME	READ ONLY	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	READ ONLY	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		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		Determines whether the current UCS is saved with a named view.
	0	Does not save current UCS with a named view

<b>System Variable</b>	<b>Value</b>	<b>Description</b>
UCSVVP	1	Saves current UCS whenever a named view is created
		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	READ ONLY	Stores the X direction of the current UCS for the current viewport in the current space.
UCSYDIR	READ ONLY	Stores the Y direction of the current UCS for the current viewport in the current space.
UNDOCTL	READ ONLY	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 on
	8	A group is currently active
UNDOMARKS	READ ONLY	Stores the number of marks placed in the UNDO control stream by the Mark option. The Mark and Back options are not available if a group is currently active.
UNITMODE		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
VIEWCTR	READ ONLY	Stores the center of view in the current viewport. Expressed as a UCS coordinate.
VIEWDIR	READ ONLY	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.

<b>System Variable</b>	<b>Value</b>	<b>Description</b>
VIEWMODE	READ ONLY	Stores 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	READ ONLY	Stores the height of the view in the current viewport. Expressed in drawing units.
VIEWTWIST	READ ONLY	Stores the view twist angle for the current viewport.
VISRETAIN		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	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	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.

<b>System Variable</b>	<b>Value</b>	<b>Description</b>
VSMAX	READ ONLY	Stores the upper-right corner of the current viewport's virtual screen. Expressed as a UCS coordinate.
VSMIN	READ ONLY	Stores the lower-left corner of the current viewport's virtual screen. Expressed as a UCS coordinate.
WHIPARC		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.
WMFBKGND		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 color is transparent. The foreground color will depend on the setting of WMFFOREGND.
	1	The background color is the same as the AutoCAD current background color, whether in model space or in a layout. The foreground color remains unchanged.
WMFFOREGND		Controls the assignment of the foreground color of AutoCAD objects 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 WMFFOREGND applies only when WMFBKGND is set to 0.

System Variable	Value	Description
		The AutoCAD defined values are:
	0	The foreground and background colors are swapped if necessary to ensure that the foreground color is <i>darker</i> than the background color.
	1	The foreground and background colors are swapped if necessary to ensure that the foreground color is <i>lighter</i> than the background color.
WORLDUCS	READ ONLY	Indicates whether the UCS is the same as the WCS.
	0	UCS differs from the WCS
	1	UCS matches the WCS
WORLDVIEW		Determines whether input to the 3DORBIT, DVIEW, and VPOINT commands is relative to the WCS (default) or the current UCS.
	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	READ ONLY	Indicates whether a drawing file is read-only or can be written to. For developers who need to determine write status through AutoLISP.
	0	Can't write to the drawing
	1	Can write to the drawing
XCLIPFRAME		Controls the visibility of xref clipping boundaries.
	0	Clipping boundary is not visible
	1	Clipping boundary is visible
XEDIT		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		Controls the fading intensity for references being edited in-place.
	0	0 percent fading, minimum value
	90	90 percent fading, maximum value

<b>System Variable</b>	<b>Value</b>	<b>Description</b>
XLOADCTL		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		Creates a path for storing temporary copies of demand-loaded xref files. For more information, see XLOADCTL.
XREFCTL		Controls whether AutoCAD writes external reference log (XLG) files.
	0	Does not write log files
	1	Writes log files
ZOOMFACTOR		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.