

# Appendix C

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## AutoCAD System Variables

### **ACADLSPASDOC**

Integer

Controls whether AutoCAD loads the LISP file into every drawing or just the first drawing opened in an AutoCAD session.

### **ACADPREFIX**

String

The ACADPREFIX variable contains the direction path for support files specified by the ACAD environment variable. Path separators are attached if needed. This is a read-only variable.

### **ACADVER**

String

The ACADVER variable contains the AutoCAD version number, which can have values. This variable is different from the DXF file \$ACADVER header variable, which stores the drawing database level number. This is a read-only variable.

### **ACISOUTVER**

Integer

The ACISOUTVER variable controls the ACIS version of SAT files created using the ACISOUT command. Initial value is 40.

### **AFLAGS**

Integer

The AFLAGS variable establishes the attribute flags for the **ATTDEF** command bit-code. The initial value for this variable is 0. Basic-

cally the value of this variable is the addition of the following: 0 - No attribute mode selected, 1 - Invisible, 2 - Constant, 4 - Verify, 8 - Preset

### **ANGBASE**

Real

The ANGBASE variable establishes the base angle 0 in relation to the current UCS. This variable is saved in the drawing and has an initial value of 0.0000.

### **ANGDIR**

Integer

The ANGDIR variable establishes the angle from angle 0 in relation to the prevailing UCS. This variable is saved in the drawing and has an initial value of 0. 0 - Direction is counter-clockwise, 1 - Direction is clockwise.

### **APBOX**

Integer

The APBOX variable turns the AutoSnap aperture box on or off. Initial value is 1. 0 - AutoSnap aperture box is not displayed, 1 - AutoSnap aperture box is displayed.

### **APERTURE**

Integer

The APERTURE variable defines the object snap target height in pixels. This variable is saved in registry and has an initial value of 10.

**AREA** Real

The most recently calculated area with commands such as AREA, LIST, or DBLIST is stored in this variable. This is a read-only variable. You can examine this variable through the **SETVAR** command.

**ATTDIA** Integer

With the ATTDIA variable you can specify whether you want to enter the attribute value through the INSERT dialog box or from the command line. This variable is saved in the drawing and has an initial value of 0.

0 - Attribute values can be specified on the command line, 1 - Attribute values can be specified in the dialog box.

**ATTMODE** Integer

The ATTMODE variable controls the Attribute Display mode and is saved in the drawing and its initial value is 1. 0 - Attribute Display mode is off, 1 - Normal, 2 - On

**ATTREQ** Integer

The value contained in the ATTREQ variable determines whether **INSERT** command uses the default attribute settings when the blocks are being inserted. 0 - The default values for all the attributes are used, 1 - This is also the initial value and enables prompts or dialog box for attribute values (depending on the value of ATTDIA variable).

**AUDITCTL** Integer

The AUDITCTL variable determines whether an .adt file (audit report file) will be created by AutoCAD. This variable is saved in registry. 0 - Does not allow writing of .adt files. This is also the initial value, 1 - Allows writing of .adt files.

**AUNITS** Integer

The AUNITS variable establishes the Angular Units mode and is saved in the drawing. 0 - Decimal degrees (initial value), 1 - Degrees/minutes/ seconds, 2 - Gradians, 3 - Radians,

4 - Surveyor's units

**AUPREC** Integer

The AUPREC variable establishes the angular units decimal places. This variable is saved in the drawing and has an initial value of 0.

**AUTOSNAP** Integer

The AUTOSNAP variable controls the display of AutoSnap marker, and Snap Tips and turns the AutoSnap magnet on or off. Initial value is 7. 0 - Turns off marker, Snap Tip and magnet, 1 - Turns on marker, 2 - Turns on Snap tooltips, 4 - Turns on magnet

**BACKZ** Real

The BACKZ variable contains the back clipping plane offset (in current drawing units) from the target plane for the current viewport. You can determine the distance between the back clipping plane and the camera point by subtracting the BACKZ value from the camera to target distance. This variable is saved in the drawing.

**BINDTYPE** Integer

Controls how xref names are handled when binding xrefs or editing xrefs in place. The value is not saved and its initial value is 0. 0 - Traditional binding behavior, 1 - Insert- like behavior.

**BLIPMODE** Integer

The visibility of the blip marks is controlled by BLIPMODE variable. This variable is saved in the drawing and its initial value is 0. 0 - Blip marks are not visible, 1 - Blip marks are visible.

**CDATE** Real

The calendar date and time is stored in this variable. This is a read-only variable.

**CECOLOR** String  
The CECOLOR variable defines the color of new objects. This variable is saved in the drawing and its initial value is “BYLAYER” (256)

**CELTSCALE** Real  
The CELTSCALE variable defines the current global linetype scale factor for objects. This variable is saved in the drawing and its initial value is 1.0000.

**CELTYPE** String  
The CELTYPE variable defines the linetype that will be used in the new objects. This variable is saved in the drawing and its initial value is “BYLAYER.”

**CELWEIGHT** Integer  
The CELWEIGHT variable is stored in the drawing. It set the lineweight of the new objects. Its initial value is 1. 1 - Sets the lineweight to “ByLayer”, 2 - Sets the lineweight to “ByBlock”, 3 - Sets the lineweight to “Default” (controlled by LWDEFAULT system variable).

**CHAMFERA** Real  
The CHAMFERA variable defines the first chamfer distance. This variable is saved in the drawing and its initial value is 0.5000.

**CHAMFERB** Real  
The CHAMFERB variable defines the second chamfer distance. This variable is saved in the drawing and its initial value is 0.5000.

**CHAMFERC** Real  
The CHAMFERC variable sets the chamfer length. This variable is saved in the drawing and its initial value is 1.0000.

**CHAMFERD** Real  
The CHAMFERD variable sets the chamfer angle. This variable is saved in the drawing and its initial value is 0.0000.

**CHAMMODE** Integer  
With the CHAMMODE variable you can specify the method that will be used to create chamfers. 0 - This is the initial value and in this case two chamfer distances are required, 1 - One chamfer length and an angle are required.

**CIRCLERAD** Real  
The CIRCLERAD variable defines the default circle radius. The initial value of this variable is 0.0000.

**CLAYER** String  
The CLAYER variable sets the current layer. This variable is saved in the drawing and its initial value is “0.”

**CMDACTIVE** Integer  
The CMDACTIVE variable contains the bit-code that signifies whether an ordinary command, transparent command, dialog box, or script is active. Basically the value of this variable is the addition of the following: 1 - Only ordinary command is active, 2 - Ordinary command as well as transparent command are active, 4 - Script is active, 8 - If this bit is set then Dialog box is active, 16 - AutoLISP is active.

**CMDECHO** Integer  
The CMDECHO variable determines whether the prompts and input of a AutoLISP (command) function are echoed. This variable is saved in registry and its initial value is 1. 0 - Echoing is disabled, 1 - Echoing enabled.

**CMDNAMES** String  
The CMDNAMES variable displays the name of the presently active command and transparent command. This variable is read-only.

**CMLJUST** Integer  
The CMLJUST variable determines the justification of a multiline. This variable is saved

in registry and its initial value is 1. 0 - Sets Top justification, 1 - Sets Middle justification, 2 - Sets Bottom justification.

**CMLSCALE** Real

The CMLSCALE variable determines the width of a multiline. For example, a scale factor of 3.0 generates a multiline that is thrice as wide as specified in the style definition. If the value is set to 0, the multiline takes the form of a single line. By specifying a negative scale factor, the order of offset lines is flipped. This variable is saved in registry and has an initial value of 1.0000.

**CMLSTYLE** String

The CMLSTYLE variable sets the name of the multiline style that is used to draw multilines. This variable is saved in registry and has an initial value "STANDARD".

**COMPASS** Integer

The COMPASS variable determines 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** Integer

The COORDS variable determines when the coordinates are updated. This variable is saved in the drawing and its initial value is 1. 0 - Coordinates are updated only upon picking points, 1 - Absolute coordinates are continuously updated, 2 - Absolute continuously plus, when a distance or angle are requested, then the distance and angle from the last point are displayed.

**CPLOTSTYLE** String

Controls the current plot style for new objects. The AutoCAD defined values are ByLayer, ByBlock, Normal, User Defined.

**CPROFILE** String

The CPROFILE variable displays the name of the current profile. The value is saved in

registry and the initial value is <<Unnamed Profile>>. This is a read only variable.

**CTAB** String

The CTAB variable returns the name of the current (model or layout) tab in the drawing. The value is saved in drawing and it is a read only variable.

**CURSORSIZE** Integer

This variable determines the size of the crosshairs as a percentage of the screen size. Initial value is 5.

**CVPORT** Integer

The CVPORT variable establishes the identification number of the current viewport. When this value is changed, the current viewport is also changed in case the following conditions hold good: 1 - The specified identification number belongs to an active viewport, 2 - The cursor movement to the specified viewport is not locked by the command being executed, 3 - Tablet mode is off. The variable is saved in the drawing and its initial value is 2.

**DATE** Real

The DATE variable contains the current date and time as a Julian date and fraction in a real number. This variable is readonly.

**DBMOD** Integer

The DBMOD variable expresses the drawing modification status using bit-code. This variable is read-only. Basically the value of this variable is the addition of the following: 0 - No changes, 1 - The object database is changed, 2 - The symbol table is changed, 4 - The database variable is changed, 8 - The window is changed, 16 - The view is changed.

**DCTCUST** String

The DCTCUST variable shows the current custom spelling dictionary path and filename.

This variable is saved in registry and its initial value is "".

**DCTMAIN** String

The DCTMAIN variable shows the current main spelling dictionary filename. Normally this file is located in the \support directory. The default main spelling dictionary can be specified using the **SETVAR** command. This variable is saved in registry and its initial value is "".

**DEFLPSTYLE** String

The DEFLPSTYLE variable specifies the default plot style for new layers. This variable is saved in registry and its initial value is "".

**DEFPLSTYLE** String

The DEFPLSTYLE variable specifies the default plot style for new objects. This variable is saved in registry and its initial value is "By Layer".

**DELOBJ** Integer

The DELOBJ variable determines whether objects used to draw other objects are kept or deleted from the drawing database. This variable is saved in the drawing and its initial value is 1. 1 - Objects are deleted from the drawing database, 0 - Objects are kept in the drawing database.

**DEMANDLOAD** Integer

This variable specifies if and when AutoCAD demand loads a third-party application if a drawing contains custom objects created in that application. Initial value is 3.

**DIASAT** Integer

The method of exiting from the most recently used dialog box is held in the DIASAT variable. This variable is readonly. 0 - Cancel, 1 - OK.

**DIMADEC** Integer

This variable is used to controls the number

of precision places displayed in the angular dimensions. Initial value is 1. -1 Angular dimension is drawn using the number of decimal places corresponding to the DIMDEC setting. 0-8 Angular dimension is drawn using the number of decimal places corresponding to the DIMADEC setting.

**DIMALT** Switch

The DIMALT variable controls the dimensioning in alternate units system. If the DIMALT variable is on (1), alternate unit dimensioning is facilitated. This variable is saved in the drawing and its initial value is Off (0).

**DIMALTD** Integer

The DIMALTD (DIMension ALternate units Decimal places) variable controls the number of decimal places (decimal precision) of the dimension text in the alternate units if DIMALT variable is on. This variable is saved in the drawing and its initial value is 2.

**DIMALTF** Real

The DIMALTF variable (DIMension ALternate units scale Factor) controls alternate units scale factor. In case DIMALT variable is enabled, all the linear dimensions will be multiplied with this factor to generate a value in an alternate units system. The initial value for DIMALTF is 25.4. This variable is saved in the drawing.

**DIMALTRND** Real

The DIMALTRND variable rounds off the alternate dimension units. The value is saved in drawing and initial value is 0.00.

**DIMALTTD** Integer

The DIMALTTD variable establishes the number of decimal places for the tolerance values of an alternate units dimension. This variable is saved in the drawing and has an initial value of 2.

**DIMALTTZ** Integer

The DIMALTTZ variable controls the suppression of zeros for alternate tolerance values. With this variable, the real-to-string transformation carried out by AutoLISP functions `rtos` and `angtos` is also influenced. This variable is saved in the drawing and has an initial value of 0. 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. Value in the range of 0 and 3 influence only the feet and inches dimensions. However, you can add 4 to the above values to omit the leading zeroes in all decimal dimensions. If you add 8, the trailing zeroes are omitted. If 12 (both 4 and 8) is added, the leading and the trailing zeroes are omitted.

**DIMALTU** Integer

The DIMALTU variable establishes the units format for alternate units of all dimensions except angular. This variable is saved in the drawing and has an initial value of 2. 1 - Scientific, 2 - Decimal, 3 - Engineering, 4 - Architectural(Stacked), 5 - Fractional (Stacked), 6 - Architectural, 7 - Fractional, 8 - Windows® Desktop settings.

**DIMALTZ** Integer

The DIMALTZ variable controls the suppression of zeros for alternate units dimension values. With this variable, the real-to-string transformation carried out by AutoLISP functions `rtos` and `angtos` is also influenced. This variable is saved in the drawing and has an initial value of 0. 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. Value in the range of 0 and 3 influence only the feet and inches dimensions. However, you can add 4 to the above values to omit the leading zeroes in all decimal dimensions. If you add

8, the trailing zeroes are omitted. If 12 (both 4 and 8) is added, the leading and the trailing zeroes are omitted.

**DIMAPOST** String

With the help of DIMAPOST variable, you position in alternate units system. If the DIMALT variable is on (1), alternate unit dimensioning is facilitated. This variable is saved in the drawing and its initial value is Off (0).

**DIMALTD** Integer

The DIMALTD (DIMension ALternate units Decimal places) variable controls the number of decimal places (decimal precision) of the dimension text in the alternate units if DIMALT variable is on. This variable is saved in the drawing and its initial value is 2.

**DIMALTF** Real

The DIMALTF variable (DIMension ALternate units scale Factor) controls alternate units scale factor. In case DIMALT variable is enabled, all the linear dimensions will be multiplied with this factor to generate a value in an alternate units system. The initial value for DIMALTF is 25.4. This variable is saved in the drawing.

**DIMALTRND** Real

The DIMALTRND variable rounds off the alternate dimension units. The value is saved in drawing and initial value is 0.00.

**DIMALTTD** Integer

The DIMALTTD variable establishes the number of decimal places for the tolerance values of an alternate units dimension. This variable is saved in the drawing and has an initial value of 2.

**DIMALTTZ** Integer

The DIMALTTZ variable controls the suppression of zeros for alternate tolerance values. With this variable, the real-to-string

transformation carried out by AutoLISP functions `rtos` and `angtos` is also influenced. This variable is saved in the drawing and has an initial value of 0. 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. Value in the range of 0 and 3 influence only the feet and inches dimensions. However, you can add 4 to the above values to omit the leading zeroes in all decimal dimensions. If you add 8, the trailing zeroes are omitted. If 12 (both 4 and 8) is added, the leading and the trailing zeroes are omitted.

**DIMALTU** Integer

The `DIMALTU` variable establishes the units format for alternate units of all dimensions except angular. This variable is saved in the drawing and has an initial value of 2. 1 - Scientific, 2 - Decimal, 3 - Engineering, 4 - Architectural(Stacked), 5 - Fractional (Stacked), 6 - Architectural, 7 - Fractional, 8 - Windows® Desktop settings.

**DIMALTZ** Integer

The `DIMALTZ` variable controls the suppression of zeros for alternate units dimension values. With this variable, the real-to-string transformation carried out by AutoLISP functions `rtos` and `angtos` is also influenced. This variable is saved in the drawing and has an initial value of 0. 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. Value in the range of 0 and 3 influence only the feet and inches dimensions. However, you can add 4 to the above values to omit the leading zeroes in all decimal dimensions. If you add 8, the trailing zeroes are omitted. If 12 (both 4 and 8) is added, the leading and the trailing zeroes are omitted.

**DIMAPOST** String

With the help of `DIMAPOST` variable, you can append a text prefix, suffix, or both to an alternate dimensioning measurement. This can be done in case of all the dimensions except angular dimensions. The variable is saved in the drawing and has an initial value of "". In order to disable an existing suffix or prefix, set the value of this variable to a single period.

**DIMASO** Switch

The `DIMASO` variable governs the creation of associative dimensions. This variable is saved in the drawing (not in the dimension style) and its initial value is set to on. Off (0) - The dimension created are not associative in nature and hence in such dimensions no association exists between the dimension and the points on the object. All the dimensioning entities such as arrowheads, dimension lines, extension lines, dimension text, etc. are drawn as separate entities, On (1) - The dimension created are associative in nature and hence in such dimensions there exists an association between the dimension and the definition points. If you edit the object, (Editing like trimming or stretching) the dimensions associated with that object also change. Also, the appearance of associative dimensions can be preserved when they are edited by commands such as `STRETCH` or `TEDIT`. For example, a vertical associative dimension is retained as a vertical dimension even after an editing operation. The associative dimension is always generated with the same dimension variable settings as defined in the dimension style.

**DIMASSOC** Integer

This variable is used to control whether the new dimension will be associative in nature or not. Initial value is 2. 0 - Dimensions are exploded, 1 - The dimensions created are non-associative, 2 - The dimensions are associative in nature.



**DIMASZ** Real

The DIMASZ (Dimension arrowhead size) variable specifies the size of dimension line and leader line arrowheads when DIMTSZ is set to zero. The size of arrowhead blocks set by DIMBLK is also controlled by DIMASZ variable. Multiples of this variable determine whether the dimension line and text will be located between the extension lines. This variable is saved in the drawing and has an initial value of 0.18 units.

**DIMATFIT** Integer

The DIMATFIT variable determines how dimension text and arrows are arranged when space is not sufficient to place both within the extension lines. The initial value is 3 and is saved in drawing. 0 - Places both text and arrows outside extension lines, 1 - Moves arrows outside extension lines, 2 - Moves text first, then arrows, 3 - Moves either text or arrows, whichever fits best.

**DIMAUNIT** Integer

The DIMAUNIT variable establishes the angle format for angular dimensions. This variable is saved in the drawing and its initial value is 0. 0 - Decimal degrees format, 1 - Degrees/minutes/seconds format, 2 - Gradians format, 3 - Radians format, 4 - Surveyor's units format.

**DIMAZIN** Integer

The DIMAZIN variable suppresses zeros for angular dimensions. The initial value is 0 and saved in drawing. 0 - Displays all leading and trailing zeros, 1 - Suppresses leading zeros in decimal dimensions, 2 - Suppresses trailing zeros in decimal dimensions, 3 - Suppresses leading and trailing zeros.

**DIMBLK** String

DIMBLK variable replaces the default arrowheads at the end of the dimension lines with a user defined block. The user defined block that may replace the standard arrowhead can

be a custom designed arrow or some other symbol. DIMBLK (DIMension BLocK) takes the name of the block as its string value. This variable is saved in the drawing and its initial value is no block (""). To discard an existing block name, set its value to a single period (.).

**DIMBLK1** String

DIMBLK1 variable designates user defined arrow block for the first end of the dimension line. This option can be used only if the DIMSAH (DIMension Separate Arrow blocks) variable is on. The value of this variable is the name of earlier formulated block as in the case of DIMBLK. You can discard an existing block name by setting its value to a single period (.). This variable is saved in the drawing and its initial value is no block ("").

**DIMBLK2** String

DIMBLK2 variable designates a user defined arrow block for the second end of the dimension line. This option can be used only if DIMSAH (DIMension Separate Arrow blocks) variable is on. The value of this variable is the name of earlier formulated block as in the case of DIMBLK. You can discard an existing block name, by setting its value to a single period (.). This variable is saved in the drawing and its initial value is no block ("").

**DIMCEN** Real

The DIMCEN (DIMension CENTER) variable governs the drawing of center marks and the center lines of circles and the arcs by the DIMCENTER, DIMDIAMETER, and DIMRADIUS commands. DIMCEN takes a distance as its argument. The value of the DIMCEN variable determines the result. This variable is saved in the drawing and its initial value is 0.0900. 0 - Center marks or center lines are not drawn, >0 - Center marks are drawn and their size is governed by the value of the DIMCEN. For example, a value of 0.250 displays center dashes which are 0.2500 units



long, <0 - Center lines in addition to center marks are drawn and again the size of the mark portion is governed by the absolute value of the DIMCEN. The center lines extend beyond the circle or arc by the value entered. For example a value of -0.2500 for DIMCEN variable will draw a center dashes 0.25 units long and also the center lines will be extended beyond the circle/arc by a distance of 0.25 units. With the DIMRADIUS and DIMDIAMETER commands, center mark or center line is generated only when the dimension line is located outside the circle or arc.

**DIMCLRD** Integer

The DIMCLRD variable is used to assign colors to dimension lines, arrowheads, and the dimension leader lines. This variable can take any permissible color number or the special color labels BYBLOCK (0) or BYLAYER (256) as its value. If you use the SETVAR command, then you have to enter the integer number of the color you want to assign to the DIMCLRD variable. This variable is saved in the drawing and its initial value is 0.

**DIMCLRE** Integer

DIMCLRE variable is used to assign color to the dimension extension lines. Just as DIMCLRD, DIMCLRE (DIMension CoLoR Extension) can take any permissible color number or the special color labels BYBLOCK or BYLAYER. This variable is saved in the drawing and its initial value is 0.

**DIMCLRT** Integer

The DIMCLRT (DIMension CoLoR Text) variable is used to assign a color to the dimension text. DIMCLRT can take any permissible color number or the special color labels BYBLOCK (0) or BYLAYER (256). This variable is saved in the drawing and its initial value is 0.

**DIMDEC** Integer

The DIMDEC variable establishes the number for decimal places of a primary units dimension. This variable is saved in the drawing and its initial value is 4.

**DIMDLE** Real

By default the dimension lines meet the extension lines. But if you want that the dimension line to continue past the extension lines, DIMDLE (Dimension Line Extension) variable can be used for this function. DIMDLE is used only when DIMTSZ variable is non-zero (When DIMTSZ variable is nonzero, ticks are drawn instead of arrows). The dimension line will extend past the extension line by the value of DIMDLE. This variable is saved in the drawing and its initial value is 0.0000.

**DIMDLI** Real

The DIMDLI variable governs the spacing between the successive dimension lines when dimensions are created with the DIMCONTINUE and DIMBASELINE commands. Successive dimension lines are offset by the DIMDLI value, if needed, to avert drawing over the previous dimension. This variable is saved in the drawing and its initial value is 0.38 units.

**DIMDSEP** Single character

The DIMDSEP variable Specifies a singlecharacter decimal separator to use when creating dimensions whose unit format is decimal. The initial value is a Decimal point and it is saved in drawing.

**DIMEXE** Real

The extension of the extension line past the dimension line is governed by the DIMEXE (Dimension EXtension line Extension) variable. This variable is saved in the drawing and has an initial value of 0.18 units.

**DIMEXO**

Real

There exists a small space between the origin points you specify and the start of the extension lines. The size of this gap is controlled by the DIMEXO (DIMension EXtension line Offset) variable. The offset distance is equal to the value of the DIMEXO variable. This variable is saved in the drawing and has an initial value of 0.0625 units.

**DIMFIT**

Integer

Obsolete. Has no effect in AutoCAD 2004 except to preserve the integrity of pre AutoCAD 2000 scripts and AutoLISP routines. This variable is saved in the drawing and its initial value is 3. DIMFIT is replaced by DIMATFIT and DIMTMOVE.

**DIMFRAC**

Integer

The DIMFRAC variable controls the fraction format when DIMLUNIT is set to 4 (Architectural) or 5 (Fractional). It is saved in drawing and initial value is 0. 0 - Horizontal, 1 - Diagonal, 2 - Not stacked (for example, 3/5).

**DIMGAP**

Real

The DIMGAP variable controls the space between the dimension line and the dimension text (distance maintained around the dimension text), when the dimension line is split into two for the placement of dimension text. The gap between the leader and annotation created with the LEADER command is also governed by DIMGAP variable. This variable is saved in the drawing and its initial value for DIMGAP is 0.0900 units. By entering a negative DIMGAP value, you can create a reference dimension, in which case you get the dimension text with a box drawn around it. DIMGAP value is also used by AutoCAD as the measure of minimum length needed for the segments of the dimension line. AutoCAD places the dimension text inside the extension lines only if the dimension line is split into two segments each of which is at least as long as DIMGAP. In case the text is positioned

over or under the dimension line, it is placed inside the dimension line only if there is space for the arrows, dimension text, and a margin between them has a minimum value at least as much as  $\text{DIMGAP} \cdot 2 \cdot (\text{DIMGAP} + \text{IMASZ})$ .

**DIMJUST**

Integer

The DIMJUST variable governs the horizontal dimension text position. This variable is saved in the drawing and its initial value is 0. 0 - The text is center justified between the extension lines, 1 - The text is placed next to the first extension line, 2 - The text is placed next to the second extension line, 3 - The text is placed above and aligned with the first extension line, 4 - The text is placed above and aligned with the second extension line.

**DIMLDRBLK**

String

The DIMLDRBLK variable controls the arrow type for leaders. To turn off arrowhead display, enter a single period (.).

**DIMLFAC**

Real

The DIMLFAC (DIMension Length FACtor) variable acts as a global scale factor for all linear dimensioning measurements. The linear distances measured by dimensioning include coordinates, diameter, and radii. These linear distances are multiplied by the prevailing DIMLFAC value before they are projected as dimension text. In this manner DIMLFAC scales the contents of the default text. The angular dimensions are not scaled. Also DIMLFAC does not apply to the values held in DIMTM, DIMTP, or DIMRND. For example, if you want to scale the default dimension measurement by a value of 2, set the value of DIMLFAC to 2. When dimensioning in the paper space, if the value of DIMLFAC variable is not zero, then the distance measured is multiplied by the absolute value of DIMLFAC. In case of dimensioning in the model space, values less than zero are neglected, instead the value of DIMLFAC is taken as 1.0. If in paper space you select the

viewport option and try to change DIMLFAC from the Dim: prompt, AutoCAD will compute a value for the DIMLFAC for you. This is illustrated as follows: Dim: DIMLFAC, Current value <1.0000> New value (Viewport): V Select viewport to set scale: The scaling of model space to paper space is computed by AutoCAD and the negative of the computed value is assigned to DIMLFAC. This variable is saved in the drawing and its initial value is 1.0000.

**DIMLIM** Switch

The DIMLIM (DIMension LIMits) variable acts as a switch and creates the dimension limits as the default text if it is on (1). Also DIMITOL is forced to be off. This variable is saved in the drawing and its initial value is off.

**DIMLUNIT** Integer

The DIMLUNIT variable controls units for all dimension types except Angular. The initial value is Off (0) and it is saved in drawing. 1 - Scientific, 2 - Decimal, 3 - Engineering, 4 - Architectural, 5 - Fractional, 6 - Windows desktop settings. DIMLWD Enum The DIMLWD variable assigns lineweight to dimension lines. Values are standard lineweights. The initial value is "By Block" and it is saved in drawing.

**DIMLWE** Enum

The DIMLWD variable assigns lineweight to extension lines. Values are standard lineweights. The initial value is "By Block" and it is saved in drawing.

**DIMPOST** String

The DIMPOST variable is used to define prefix or suffix to the dimension measurement. The variable is saved in the drawing and has an initial value "" (empty string). DIMPOST takes a string value as its argument. For example if you want to have a suffix for centimeters, set DIMPOST to "cm". A distance of

4.0 units will be displayed as 4.0cm. In case tolerances are enabled, the suffix you have defined gets applied to the tolerances as well as to the main dimension. To establish a prefix to a dimension text, type "<>" and then the prefix at the same prompt.

**DIMRND** Real

The DIMRND (DIMension RouND) variable is used for rounding all the dimension measurements to the specified value. For example if the DIMRND is set to 0.10, then all the measurements are rounded to the nearest 0.10 unit. Like wise a value of 1 for this variable will result in the rounding of all the measurements to the nearest integer. The angular measurements cannot be rounded. The variable is saved in the drawing and has an initial value of 0.0000.

**DIMSAH** Switch

The DIMSAH (DIMension Separate custom Arrow Head) variable governs the placement of user-defined arrow blocks instead of the standard arrows at the end of the dimension line. As explained before, DIMBLK1 variable places a user defined arrow block at the first end of the dimension line and DIMBLK2 places a user defined arrow block at the other end of the dimension line. This variable is saved in the drawing and its initial value is off. On - DIMBLK1 and DIMBLK2 specify different user-defined arrow blocks to be drawn at the two ends of the dimension line, Off - Ordinary arrowheads or userdefined arrowhead block defined by the DIMBLK variable is used.

**DIMSCALE** Real

The DIMSCALE variable controls the scale factor for all the size-related dimension variables such as those that affect text size, center mark size, arrow size, leader objects, etc. The DIMSCALE is not applied to the measured lengths, coordinates, angles, or tolerances. The default value for this variable is

1.0000; and in this case the dimensioning variables assume their preset values and the drawing is plotted at full scale. If the drawing is to be plotted at half the size, then the scale factor is the reciprocal of the drawing size. Hence the scale factor or the DIMSCALE value will be reciprocal of 1/2 which is  $2/1 = 2$ . 0.0 - A default value based on the scaling between the current model space viewport and paper space is calculated. In case you are not using the paper space feature, then the scale factor is 1.0, >0 - A scale factor is computed that makes the text sizes, arrowhead sizes, and scaled distances to plot at their face value.

**DIMSD1** Switch

The DIMSD1 (DIMension Suppress Dimension line 1) variable suppresses the drawing of the first dimension line when it is on. This variable is saved in the drawing and its initial value is off.

**DIMSD2** Switch

The DIMSD1 (DIMension Suppress Dimension line 2) variable suppresses the drawing of second dimension line when it is on. This variable is saved in the drawing and its initial value is off.

**DIMSE1** Switch

The DIMSE1 variable is used to suppress drawing of the first extension line. When DIMSE1 (DIMension Suppress Extension line 1) is on, the first extension line is not drawn. This variable is saved in the drawing and its initial value is off.

**DIMSE2** Switch

The DIMSE2 variable is used to suppress drawing of the second extension line. When DIMSE2 (DIMension Suppress Extension line 2) is on, the second extension line is not drawn. This variable is saved in the drawing and its initial value is off.

**DIMSHO** Switch

DIMSHO variable governs the redefinition of dimension entities while dragging into some position. If DIMSHO (DIMension SHOW dragged dimensions) is on, associative dimensions will be computed dynamically as they are dragged. The DIMSHO value is saved in the drawing (not in a dimension style) and its initial value is on (1). Dynamic dragging reduces the speed of some computers and hence in such situations DIMSHO should be set off (0). However, when you are using the pointing device to specify the length of the leader in Radius and Diameter dimensioning, the DIMSHO setting is neglected and dynamic dragging is used.

**DIMSOXD** Switch

If you want to place text inside the extension lines, you will have to set the DIMITX variable on. And if you want to suppress the dimension lines and the arrow heads you will have to set the DIMDSOXD (DIMension Suppress Outside eXtension Dimension lines) variable on. DIMSOXD suppresses the drawing of dimension lines and the arrow heads when they are placed outside the extension lines. If DIMITX is on and DIMSOXD is off and there is not enough space inside the extension lines for drawing the dimension lines, then dimension lines will be drawn outside the extension lines. In such a situation, if both DIMITX and DIMSOXD are on, then the dimension line will be totally suppressed. DIMSOXD works only when DIMITX is on. The DIMSOXD variable is saved in the drawing and its initial value is off.

**DIMSTYLE** String

DIMSTYLE variable is used for displaying the name of the present dimension style. DIMSTYLE is a read-only variable and is saved in the drawing. You can change the dimension style using the DDIM or DIMSTYLE command.

**DIMTAD**

Integer

The DIMTAD (DIMension Text Above Dimension line) variable governs the vertical placement of the dimension text with respect to the dimension line. DIMTAD gets actuated when dimension text is drawn between the extension lines and is aligned with the dimension line, or when the dimension text is placed outside the extension lines. This variable is saved in the drawing and its initial value is 0. 0 - For this value the dimension text is placed at the center between the extension lines, 1 - The dimension text is placed above the dimension line and a single (unsplit) dimension line is drawn under it spanning between the extension lines. The exceptions to this arise when the dimension line is not horizontal and text inside the extension line is forced to be horizontal by making DIMTIH = 1. The space between the dimension line and the baseline of the lowest line of text is nothing but the prevailing DIMGAP value, 2 - The dimension text is placed on the side of the dimension line most remote from the defining points, 3 - The dimension text is placed to tune to a JIS representation.

**DIMTDEC**

Integer

The DIMTDEC variable establishes the number of decimal places for the tolerance values for the primary units dimension. This variable is saved in the drawing and its initial value is 4.

**DIMTFAC**

Real

With the DIMTFAC (DIMension Tolerance scale FACtor) variable you can control the scaling factor of the text height of the tolerance values in relation to the dimension text height set by DIMTXT. Suppose DIMTFAC is set to 1.0 (the default value for DIMTFAC variable), then the text height of the tolerance text will be equal to the dimension text height. If DIMTFAC is set to a value of 0.50, the text height of the tolerance is half of the dimension text height. This variable is saved in the

drawing and its initial value is 1.0000. It is important to remember that the scaling of tolerance text to any requirement is possible only when DIMTOL is on and DIMTM and DIMTP variable values are not identical, or when DIMLIM is on.

**DIMTIH**

Switch

The DIMTIH (DIMension Text Inside Horizontal) variable controls the placement of the dimension text inside the extension lines for Linear, Radius, Angular, and Diameter dimensioning. DIMTIH is effective only when the dimension text fits between the extension lines. On - If DIMTIH is on (the default setting), it forces the dimension text inside the extension lines to be placed horizontally, rather than aligned, Off - In case DIMTIH is off, the dimension text is aligned with the dimension line.

**DIMTIX**

Switch

The DIMTIX variable draws the text between the extension lines. This variable is saved in the drawing and its initial value is Off. On - When DIMTIX is set to on, the dimension text is placed amidst the extension lines even if it would normally be placed outside the extension lines, Off - If DIMTIX is off, the placement of the dimension text depends on the type of dimension. For example, if the dimensions are Linear or Angular, the text will be placed inside the extension lines by AutoCAD if there is enough space available. While as for the Radius and Diameter dimensions, the text is placed outside the object being dimensioned.

**DIMTM**

Real

The DIMTM variable establishes the lower (minimum) tolerance limit for the dimension text. Tolerance is defined as the total amount by which a particular dimension is permitted to vary. The tolerance or limit values are drawn only if DIMTOL or DIMLIM variable is on. DIMTM (DIMension Tolerance Minus)

identifies the lower tolerance and DIMTP (DIMension Tolerance Plus) identifies the upper tolerance. You can specify signed values for DIMTM and DIMTP variables. If DIMTOL is on and both DIMTM and DIMTP have same value, AutoCAD draws the “ñ” symbol followed by the tolerance value. If DIMTM and DIMTP hold different values, the upper tolerance is drawn above the lower tolerance. Also a positive (+) sign is appended to the DIMTP value if it is positive. For minus tolerance value (DIMTM), the negative of the value you enter (negative sign if you enter positive value and positive sign if you enter negative value) is displayed. Signs are not appended with zero. This variable is saved in the drawing and its initial value is 0.0000.

**DIMTMOVE** Integer

The DIMTMOVE variable controls the dimension text movement rules. Its initial value is 0 and it is saved in drawing. 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** Switch

If DIMTOFL variable is turned on, a dimension line is drawn between the extension lines even if the text is located outside the extension lines. When DIMTOFL is off, for radius and diameter dimensions, the dimension line and the arrowheads are drawn inside the arc or circle, while the text and the leader are placed outside. This variable is saved in the drawing and its initial value is Off.

**DIMTOH** Switch

The DIMTOH (DIMension Text Outside Horizontal) variable controls the orientation of the dimension text outside the extension lines. If DIMTOH is on, it forces the dimension text outside the extension lines to be placed horizontally, rather than aligned. In case DIMTOH is off, the dimension text is

aligned with the dimension line. You must have noticed that the variable DIMTOH is same as DIMTIH variable except it controls text drawn outside the extension lines. This variable is saved in the drawing and its initial value is On.

**DIMTOL** Switch

DIMTOL (DIMension with TOLerance) variable is used for controlling the appending of dimension tolerances to the dimension text. With DIMTM and DIMTP you can define the values of the lower and upper tolerances. If the DIMTOL variable is set on, the tolerances are appended to the default text. When DIMTOL is set on, DIMLIM variable is set off. This variable is saved in the drawing and its initial value is Off.

**DIMTOLJ** Integer

DIMTOLJ variable establishes the vertical justification for the tolerance values with respect to the normal dimension text. This variable is saved in the drawing and its initial value is 1. 0 - Bottom, 1 - Middle, 2 - Top.

**DIMTP** Real

The DIMTP (DIMension Tolerance Plus) variable establishes the upper (maximum) tolerance limit for the dimension text. Tolerance is defined as the total amount by which a particular dimension is permitted to vary. The tolerance or limit values are drawn only if DIMTOL or DIMLIM variable is on. If DIMTOL is on and both DIMTM and DIMTP have same value, AutoCAD draws the “ñ” symbol followed by the tolerance value. If DIMTM and DIMTP hold different values, the upper tolerance is drawn above the lower tolerance. Also a positive (+) sign is appended to the DIMTP value if it is positive. This variable is saved in the drawing and its initial value is 0.0000.

**DIMTSZ** Real

The DIMTSZ variable defines the size of ob-



lique strokes (ticks) instead of arrowheads at the end of the dimension lines (just as in architectural drafting), for Linear, Radius, and Diameter dimensioning. This variable is saved in the drawing and its initial value is 0.0000. 0 - Arrows are drawn, >0 - Oblique strokes instead of arrows are drawn. The size of the ticks is computed as DIMTSZ\*DIMSCALE. Hence if DIMSCALE factor is one then the size of the tick is equal to the DIMTSZ value. This variable is also used to determine whether dimension line and dimension text will get accommodated between the extension lines.

**DIMTVP** Real

The DIMTVP (DIMension Text Vertical Position) variable, controls the vertical placement of the dimension text over or under the dimension line. In certain cases DIMTVP is used as DIMTAD to control the vertical position of the dimension text. DIMTVP value holds good only when DIMTAD is off. The vertical placing of the text is done by offsetting the dimension text. The amount of the vertical offset of dimension text is a product of text height and DIMTVP value. If the value of DIMTVP is 1.0, DIMTVP acts as DIMTAD. However if the value of the DIMTVP is less than 0.70, the dimension line is broken into two segments to accommodate the dimension text. This variable is saved in the drawing and its initial value is 0.0000.

**DIMTXSTY** String

The DIMTXTSTY variable specifies the text style of the dimension. This variable is saved in the drawing and its initial value is "STANDARD".

**DIMTXT** Real

The DIMTXT variable is used to control the height of the dimension text except if the current text style has a fixed height. This variable is saved in the drawing and its initial value is 0.1800.

**DIMTZIN** Integer

With the DIMZIN variable you can control the suppression of the zeros for tolerance values. The variable is saved in the drawing and its initial value is 0. 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. You can add 4 to the above values to omit the leading zeroes in all decimal dimensions. If you add 8, the trailing zeroes are omitted. If 12 (both 4 and 8) is added, the leading and the trailing zeroes are omitted.

**DIMUNIT** Integer

The DIMUNIT variable establishes the linear units format for all dimension styles. 1 - Scientific units format, 2 - Decimal units format, 3 - Engineering units format, 4 - Architectural (stacked) units format, 5 - Fractional (stacked) units format, 6 - Architectural, 7 - Fractional, 8 - Window Desktop.

**DIMUPT** Switch

This variable governs the cursor functionality for User Positioned Text. This variable is saved in the drawing and its initial value is Off. 0 - The cursor controls the location of the dimension line only, 1 - The cursor controls the location of both the dimension text and the dimension line.

**DIMZIN** Integer

The DIMZIN (DIMension Zero INch) controls the suppression of the inches part of a feet-inches dimension when the distance is integral number of feet or the suppression of the feet portion when the distance is less than one foot. This variable is saved in the drawing and its initial value is 0. 0 - Suppress zero feet and exactly zero inches, 1 - Include zero feet and, exactly zero inches, 2 - Include zero feet, suppress zero inches, 3 - Include zero inches, suppress zero feet. If the dimension has feet and a fractional inch part, the num-



ber of inches is included even if it is zero. This is independent of the DIMZIN setting. For example a dimension such as 1'-2/3" never exist. It will be in the form 1'-0 2/3". The integer values 0-3 of the DIMZIN variable control the feet and inch dimension only, while as you can add 4 to omit the leading zeroes in all decimal dimensions. For example 0.2600 becomes .2600. If you add 8, the trailing zeroes are omitted. For example, 4.9600 becomes 4.96. If 12 (both 4 and 8) is added, the leading and the trailing zeroes are omitted. For example, 0.2300 becomes .23.

**DISPSILH** Integer

The DISPSILH variable governs the display of silhouette curves of body objects in a wireframe model. The variable is saved in the drawing and its initial value is 0. 0 - Silhouette curves of body objects not displayed, 1 - Silhouette curves of body objects displayed.

**DISTANCE** Real

The DISTANCE variable holds the distance value determined by the DIST command. This command is read-only. DONUTID Real The DONUTID variable establishes the default inside diameter of a donut. The initial value for this variable is 0.5000.

**DONUTOD** Real

The DONUTOD variable establishes the default outside diameter of a donut. It is important that the value of this variable be greater than zero. In case the value of DONUTID is greater than that of DONUTOD, then the two values are interchanged by the next command. The initial value for this variable is 1.0000.

**DRAGMODE** Integer

The DRAGMODE variable establishes the Object Drag mode while carrying out editing operations. 0 - Dragging disabled, 1 - Dragging enabled if invoked, 2 - Auto. This vari-

able is saved in the drawing and is initially set to 2.

**DRAGP1** Integer

The DRAGP1 variable establishes the regen-drag input sampling rate. This variable is saved in registry and is initially set to a value of 10.

**DRAGP2** Integer

The DRAGP2 variable establishes the fastdrag input sampling rate. This variable is saved in registry and is initially set to a value of 25.

**DWGCHECK** Integer

The DWGCHECK variable determines whether a drawing was last edited by a product other than AutoCAD. Its initial value is 0 and it is stored in registry. 0 - The dialog box display is suppressed, 1 - The dialog box will be displayed.

**DWGCODEPAGE** String

The DWGCODEPAGE variable holds the drawing code page. When you create a new drawing, this variable is set to the system code page, Otherwise it is not maintained by AutoCAD. This variable describes the code page of the drawing. You can set this variable to any value by using the SYSCODEPAGE system variable or set it as undefined. It is a read-only variable and is saved in the drawing.

**DWGNAME** String

The DWGNAME variable holds the name of the drawing as specified by the user. In case the drawing has not been assigned a name, the DWGNAME variable conveys that the drawing is unnamed. The drive and directory is also included if it was specified. This variable is a read-only variable.

**DWGPREFIX** String

The DWGPREFIX variable holds the drive and directory prefix for the drawing. This variable is a read-only variable.

**DWGTITLED**

Integer

The DWGTITLED variable reflects whether the present drawing has been named. 0 - Indicates that the drawing has not been named, 1 - Indicates that the drawing has been named.

**EDGEMODE**

Integer

With the EDGEMODE variable you can control how the EXTEND and TRIM commands determine boundary and cutting edges. 0 - In this case the selected edge is used without an extension, 1 - The object is trimmed or extended to an imaginary extension of the cutting or boundary edges. This is the initial value for this variable.

**ELEVATION**

Real

The ELEVATION variable holds the current 3D elevation associated to the current UCS for the current space. This variable is saved in the drawing and has an initial value of 0.0000.

**EXPERT**

Integer

The issuance of some prompts is controlled with the EXPERT variable. The initial value for this variable is 0. 0 - All the prompts are issued, 1 - The “About to regen, proceed?” prompt and “Really want to turn the current layer off?” prompts are suppressed, 2 - The preceding prompts and “Block already defined. Redefine it?” (BLOCK command) and “A drawing with this name already exists. Overwrite it?” (SAVE or WBLOCK commands) are suppressed, 3 - The preceding prompts and the ones issued by LINETYPE if you try to load a linetype that is already loaded or create a new linetype in a file that already defines it are suppressed, 4 - The preceding prompts and the ones issued by UCS Save and VPORTS Save in case the name you provide already exists are suppressed, 5 - The preceding prompts and the ones issued by the DIMSTYLE Save option, and DIMOVERRIDE in case the dimension style

name you provide already exists, are suppressed. Whenever the EXPERT command suppresses a prompt, the corresponding operation is carried out as if you have entered Y as the response to the prompt. The EXPERT command can influence menu macros, scripts, AutoLISP, and the command functions.

**EXPLMODE**

Integer

The EXPLMODE variable govern whether the EXPLODE command can explode nonuniformly scaled blocks. This variable is saved in the drawing and its initial value is 1. 0 - Nonuniformly scaled blocks cannot be exploded, 1 - Nonuniformly scaled blocks can be exploded.

**EXTMAX**

3D Point

The EXTMAX variable holds the upperright point of the drawing extents and is saved in the drawing. The drawing extents increase outward when new objects are drawn and reduce only when ZOOM All or ZOOM Extents is used. The variable is reported in the World coordinates for the current space.

**EXTMIN**

3D Point

The EXTMIN variable holds the lowerleft point of the drawing extents and is saved in the drawing. The drawing extents increase outward when new objects are drawn and reduce only when ZOOM All or ZOOM Extents is used. The variable is reported in the World coordinates for the current space.

**EXTNAMES**

Integer

The EXTNAMES variable controls the parameters for named object names (such as linetypes, lineweights and layers) stored in symbol tables. Its initial value is 1 and is saved in drawing. 0 - Uses Release 14 parameters, which limit names to 31 characters in length, 1 - Uses AutoCAD 2000 parameters. Names can be up to 255 characters in length.

**FACETRATIO** Integer

The FACETRATIO variable sets the aspect ratio of faceting for cylindrical and conic ACIS solids. Its initial value is 0 and it is not saved. 0 - Creates an N by 1 mesh for cylindrical and conic ACIS solids, 1 - Creates an N by M mesh for cylindrical and conic ACIS solids.

**FACETRES** Real

The FACETRES variable adjusts the smoothness of shaded and objects whose hidden lines have been removed. This variable can be assigned values in the range of 0.010 to 10.0. The variable is saved in the drawing and has an initial value of 0.5.

**FILEDIA** Integer

The FILEDIA variable suppresses the display of file dialog boxes. This variable is saved in registry and has an initial value of 1. 0 - The file dialog boxes are disabled. However you can make AutoCAD to display the file dialog box by entering a tilde (~) as the response to the prompt. This applied for AutoLISP and ADS functions also, 1 - The file dialog boxes are enabled except when a script or AutoLISP/ADS program is active in which case only a prompt appears.

**FILLETRAD** Real

The FILLETRAD variable holds the current fillet radius and is saved in the drawing and its initial value is 0.5000.

**FILLMODE** Integer

FILLMODE variable indicates whether objects drawn with SOLID command are filled in. This variable is saved in the drawing and its initial value is 1. 0 - Objects are not filled, 0 - Objects are filled.

**FONTALT** String

The FONTALT variable specifies the alternate font to be used in case the specified font file cannot be found. In case you have not specified an alternate font, AutoCAD issues a

warning. This variable is saved in registry and its initial value is "simplex.shx".

**FONTMAP** String

The FONTMAP variable specifies the font mapping file to be used in case the specified font file cannot be found. This file holds one font mapping per line. The original font and the substitute font are separated by a semicolon (;). This variable is saved in registry and its initial value is "Acad.fmp".

**FRONTZ** Real

The FRONTZ variable contains the front clipping plane offset (in current drawing units) from the target plane for the current viewport. You can determine the distance between the front clipping plane and the camera point by subtracting the FRONTZ value from the camera to target distance. This variable is saved in the drawing and is read-only.

**FULLOPEN** Integer

The FULLOPEN variable checks whether the current drawing is partially open. Its value is not saved and it is a read-only variable. 0 - Indicates a partially open drawing, 1 - Indicates a fully open drawing.

**GRIDMODE** Integer

The GRIDMODE variable specifies whether the grid is turned on or off. This variable is saved in the drawing and its initial value is 0. 0 - The grid is turned off, 1 - The grid is turned on.

**GRIDUNIT** 2D point

The GRIDUNIT variable specifies the X and Y grid spacing for the current viewport. The changes made to the grid spacing are manifested only after using the REDRAW or REGEN command. This variable is saved in the drawing and its initial value is 0.5000,0.5000.

**GRIPBLOCK**

Integer

The GRIPBLOCK variable controls the assignment of grips in blocks. This variable is saved in registry and its initial value is 0. 0 - The grip is assigned only to the insertion point of the block, 1 - Grips are assigned to objects within the block.

**GRIPCOLOR**

Integer

The GRIPCOLOR variable controls the color of nonselected grips. It can take a value in the range of 1 to 255. This variable is saved in registry and its initial value is 5.

**GRIPHOT**

Integer

The GRIPHOT variable controls the color of selected grips. It can take a value in the range of 1 to 255. This variable is saved in registry and its initial value is 1.

**GRIPS**

Integer

With the GRIPS variable you can make use of selection set grips for the Stretch, Move, Rotate, Scale, and Mirror grip modes. This variable is saved in registry and its initial value is 1. 0 - Grips are disabled, 1 - Grips are enabled.

**GRIPSIZE**

Integer

The GRIPSIZE variable allows you to assign a size to the box drawn to show the grip. It sets its half height in pixels. This variable can be assigned a value in the range of 1 to 255. The variable is saved in registry and its initial value is 3.

**HALOGAP**

Integer

This variable is used to set the distance to shorten the length of the haloed line. The initial value of this variable is 0.

**HANDLES**

Integer

The HANDLES variable is always on (1), which states that object handles are enabled and can be accessed by applications. This variable is saved in the drawing and is read-only.

**HIDEPRECISION**

Integer

The HIDEPRECISION variable sets the accuracy of hides and shades. Hides can be calculated in double precision or single precision. 0 - Single precision; uses less memory, 1 - Double precision; uses more memory.

**HIDETEXT**

Switch

Specified whether or not the text will be hidden when the **HIDE** command is executed. If set to **OFF**, the text is not hidden and if set to **ON**, the text is hidden.

**HIGHLIGHT**

Integer

The HIGHLIGHT variable governs object highlighting. Objects selected with grips are not influenced. **FULLOPEN** Integer The FULLOPEN variable checks whether the current drawing is partially open. Its value is not saved and it is a read-only variable. 0 - Indicates a partially open drawing, 1 - Indicates a fully open drawing.

**GRIDMODE**

Integer

The GRIDMODE variable specifies whether the grid is turned on or off. This variable is saved in the drawing and its initial value is 0. 0 - The grid is turned off, 1 - The grid is turned on.

**GRIDUNIT**

2D point

The GRIDUNIT variable specifies the X and Y grid spacing for the current viewport. The changes made to the grid spacing are manifested only after using the REDRAW or REGEN command. This variable is saved in the drawing and its initial value is 0.5000,0.5000.

**GRIPBLOCK**

Integer

The GRIPBLOCK variable controls the assignment of grips in blocks. This variable is saved in registry and its initial value is 0. 0 - The grip is assigned only to the insertion point of the block, 1 - Grips are assigned to objects within the block.

**GRIPCOLOR** Integer

The GRIPCOLOR variable controls the color of nonselected grips. It can take a value in the range of 1 to 255. This variable is saved in registry and its initial value is 5.

**GRIPHOT** Integer

The GRIPHOT variable controls the color of selected grips. It can take a value in the range of 1 to 255. This variable is saved in registry and its initial value is 1.

**GRIPS** Integer

With the GRIPS variable you can make use of selection set grips for the Stretch, Move, Rotate, Scale, and Mirror grip modes. This variable is saved in registry and its initial value is 1. 0 - Grips are disabled, 1 - Grips are enabled.

**GRIPSIZE** Integer

The GRIPSIZE variable allows you to assign a size to the box drawn to show the grip. It sets its half height in pixels. This variable can be assigned a value in the range of 1 to 255. The variable is saved in registry and its initial value is 3.

**HALOGAP** Integer

This variable is used to set the distance to shorten the length of the haloed line. The initial value of this variable is 0.

**HANDLES** Integer

The HANDLES variable is always on (1), which states that object handles are enabled and can be accessed by applications. This variable is saved in the drawing and is read-only.

**HIDEPRECISION** Integer

The HIDEPRECISION variable sets the accuracy of hides and shades. Hides can be calculated in double precision or single precision. 0 - Single precision; uses less memory, 1 - Double precision; uses more memory.

**HIDETEXT** Switch

Specified whether or not the text will be hidden when the **HIDE** command is executed. If set to **OFF**, the text is not hidden and if set to **ON**, the text is hidden.

**HIGHLIGHT** Integer

The HIGHLIGHT variable governs object highlighting. Objects selected with grips are not influenced. 0 - Object selection highlighting is disabled, 1 - Object selection highlighting is enabled. This is the initial value for the variable.

**HPANG** Real

The HPANG variable specifies the angle of the hatch pattern. The initial value for this variable is 0.

**HPBOUND** Real

The HPBOUND variable governs the object type created by the BHATCH and BOUNDARY commands. This variable is saved in the drawing and its initial value is 1. 0 - A region is created, 1 - A polyline is created.

**HPDOUBLE** Integer

The HPDOUBLE variable governs the hatch pattern doubling for user-defined patterns. The initial value of this variable is 0. 0 - Hatch pattern doubling disabled, 1 - Hatch pattern doubling enabled.

**HPNAME** String

The default hatch pattern name is established with HPNAME variable. The name can be up to 34 characters and spaces are not allowed. Empty string ("" ) is returned if no default exists. To set no default enter a period (.). The initial value of this variable is "ANSI31".

**HPSCALE** Real

The hatch pattern scale factor is specified with HPSCALE variable. This variable cannot assume zero value. The initial value of this variable is 1.0000.

**HPSPACE**

Real

The hatch pattern line spacing for userdefined simple patterns is specified by HPSPACE variable. This variable cannot assume zero value. The initial value of this variable is 1.0000.

**HYPERLINKBASE**

String

The HYPERLINKBASE variable specifies the path used for all relative hyperlinks in the drawing. Its initial value is "" and is saved in drawing.

**IMAGEHLT**

Integer

The IMAGEHLT variable Controls whether the entire raster image or only the raster image frame is highlighted. Its initial value is 0 and is stored in registry. 0 - Highlights only the raster image frame, 1 - Highlights the entire raster image.

**INDEXCTL**

Integer

Controls whether layer and spatial indexes are created and saved in drawing. Initial value is 0. 0 - No indexes created, 1 - Layer index created, 2 - Spatial index is created, 3 - Layer and spatial are created.

**INETLOCATION**

Real

Stores the Internet location used by BROWSER. Initial value is "www.autodesk.com/acaduser".

**INSBASE**

3D point

The insertion base point established by the BASE command is stored in this variable. This point is defined in UCS coordinates for the current space. The variable is saved in the drawing and its initial value is 0.0000,0.0000,0.0000.

**INSNAME**

String

The INSNAME variable establishes the default block name for or INSERT commands. To set no default enter a period (.). The initial value of this variable is "".

**INSUNITS**

Integer

The INSUNITS variable specifies a drawing units value, when you drag a block or image from AutoCAD DesignCenter. Its initial value is 0 and it is stored in drawing. 0 - Unspecified (No units), 1 - Inches, 2 - Feet, 3 - Miles, 4 - Millimeters, 5 - Centimeters, 6 - Meters, 7 - Kilometers, 8 - Microinches, 9 - Mils, 10 - Yards, 11 - Angstroms, 12 - Nanometers, 13 - Microns, 14 - Decimeters, 15 - Decameters, 16 - Hectometers, 17 - Gigameters, 18 - Astronomical Units, 19 - Light Years, 20 - Parsecs.

**INSUNITSDEFSOURCE**

Integer

The INSUNITSDEFSOURCE controls source content units value. Valid ranges between 0 to 20. Its initial value is 0 and it is stored in registry.

**INSUNITSDEFTARGET**

Integer

The INSUNITSDEFTARGET controls source content units value. Valid ranges between 0 to 20. Its initial value is 0 and it is stored in registry.

**ISAVEBAK**

Integer

Improves the speed of incremental saves, especially for large drawings on Windows. Initial value is 1. 0 - No BAK file is created, 1 - A BAK file is created.

**ISAVEPERCENT**

Integer

Determines the amount of wasted space tolerated in a drawing file. Initial value is 50.

**ISOLINES**

Integer

The ISOLINES variable specifies the number of isolines per surface on objects. The variable can accept a value in the range of 0 to 2047. This variable is saved in the drawing and its initial value is 4.

**LASTANGLE**

Real

The LASTANGLE variable holds the end angle of the last arc entered, with respect to

the XY plane of the current UCS for the current space. This variable is a readonly variable.

**LASTPOINT** 3D point

The LASTPOINT variable holds the UCS coordinates for the current space of the most recently entered point. This variable is saved in the drawing and its initial value is 0.0000,0.0000,0.0000.

**LASTPROMPT** String

The LASTPROMPT variable stores the last string echoed to the command line. Initial value is "".

**LAYOUTREGENCTL** Integer

Specifies whether or not the objects in the different working environments are regenerated when you switch to a different environment. Initial value is 2. 0 - Drawings are regenerated every time you switch from one environment to the other, 1 - Drawings are regenerated for the **Model** tab and the last active layout, 3 - The drawings are regenerated only the first time when you switch to that environment.

**LENSLENGTH** Real

The LENSLENGTH variable holds the length of the lens (in mm) used in perspective viewing for the current viewport. This variable is saved in the drawing.

**LIMCHECK** Integer

This variable governs the drawing of objects outside the specified drawing limits. This variable is saved in the drawing and its initial value is 0. 0 - Object can be drawn outside the drawing limits, 1 - Object cannot be drawn outside the drawing limits.

**LIMMAX** 2D point

The upper-right drawing limits stated in World coordinates (for the current space) are held in the LIMMAX variable. This variable

is saved in the drawing and its initial value is 12.0000,9.0000.

**LIMMIN** 2D point

The lower-left drawing limits stated in World coordinates (for the current space) are held in the LIMMIN variable. This variable is saved in the drawing and its initial value is 0.0000,0.0000.

**LISPINIT** Integer

Specifies whether AutoLISP defined functions and variables are preserved when you open a new drawing. It is saved in registry and the initial value is 1. 0 - AutoLISP functions and variables are preserved, 1 - AutoLISP functions and variables are valid in current drawing only.

**LOCALE** String

The LOCALE variable shows the ISO language code of the current AutoCAD version in use. The initial value of this variable is "enum" (varies by country)

**LOGFILEMODE** Integer

Specifies whether the contents of the text window are written to a log file. Its initial value is 0. 0 - Log file is not maintained, 1 - Log file is maintained.

**LOGFILENAME** String

Specifies the path for the log file. Initial value is "C:\Program Files AutoCAD 2004 \acad.log".

**LOGFILEPATH** String

The LOGFILEPATH variable specifies the path for the log files for all drawings in a session. The initial value varies depending on where you installed AutoCAD and is saved in registry.



**LOGINNAME** String  
The LOGINNAME variable shows the user's name as specified while configuring when AutoCAD is loaded.

**LTSCALE** Real  
The LTSCALE variable establishes global linetype scale factor. The variable is saved in the drawing and its initial value is 1.0000.

**LUNITS** Integer  
The LUNITS variable establishes the Linear Units mode. The variable is saved in the drawing and its initial value is 2. 1 - Scientific units mode, 2 - Decimal units mode, 3 - Engineering units mode, 4 - Architectural units mode, 5 - Fractional units mode.

**LUPREC** Integer  
The LUPREC variable establishes the linear units decimal places or denominator. The variable is saved in the drawing and its initial value is 4.

**LWDEFAULT** Enum  
The LWDEFAULT variable controls the value for the default lineweight. The default lineweight can be set to any valid lineweight value in millimeters.

**LWDISPLAY** Integer  
The LWDISPLAY variable controls whether the lineweight is displayed on the Model or Layout tab. Its initial value 0 and is saved in drawing. 0 - Lineweight is not displayed, 1 - Lineweight is displayed.

**LWUNITS** Integer  
The LWUNITS variable controls whether lineweight units are displayed in inches or millimeters. Its initial value is 1 and is saved in registry. 0 - Inches, 1 - Millimeters.

**MAXACTVP** Integer  
The MAXACTVP variable specifies the maximum number of viewports to regenerate at

one time. The initial value of this variable is 48.

**MAXSORT** Integer  
The MAXSORT variable sets the maximum number of symbol names of file names that are to be sorted by listing commands. In case the total number of items is greater than this number, then no items are sorted. This variable is saved in registry and its initial value is 200.

**MBUTTONPAN** Integer  
The MBUTTONPAN variable controls the behavior of the third button or wheel on the pointing device. Its initial value is 1 and it is saved in registry. 0 - Supports the action defined in the AutoCAD menu (.mnu) file, 1 - Supports panning by holding and dragging the button or wheel.

**MEASUREINIT** Integer  
The MEASUREMENT variable sets the drawing units as English or metric and controls which hatch pattern and linetype files an existing drawing uses when it's opened. It is saved in drawing and the initial value is 0. 0 - English( AutoCAD uses ANSI hatch patterns and linetypes0, 1 - Metric ( AutoCAD uses ISOHatch and ISOLinetypes).

**MEASUREMENT** Integer  
The MEASUREMENT variable sets the drawing units as English or metric and controls which hatch pattern and linetype files for the current drawing only. It is saved in drawing and the initial value is 0. 0 - English( AutoCAD uses ANSI hatch patterns and linetypes0, 1 - Metric ( AutoCAD uses ISOHatch and ISOLinetypes).

**MENUCTL** Integer  
The MENUCTL variable governs the page switching of the screen menu. This variable is saved in registry and its initial value is 1. 0 - For this value the screen menu does not

switch pages in response to a keyboard command entry, 1 - For this value the screen menu switches pages in response to a keyboard command entry.

**MENUECHO** Integer

The MENUECHO variable sets menu echo and prompt control bits. The initial value for this variable is 0. The variable is the addition of the following: 1 - The echo of menu items is suppressed, 2 - The display of system prompts during menu is suppressed, 4 - The ^P toggle of menu echoing is disabled, 8 - The input/ output strings and debugging aid for DIESEL macros is displayed.

**MENUNAME** String

The MENUNAME variable contains the MENUGROUP name. In case the prevailing primary menu has no MENUGROUP name, the menu file includes the path if the location of the file is not defined in the AutoCAD environment setting. This variable is saved in the application leader and is a read-only variable.

**MIRRTEXT** Integer

The MIRRTEXT variable governs how the MIRROR command mirrors text. This variable is saved in the drawing and its initial value is 1. 0 - The text direction is retained, 1 - The text is mirrored.

**MODEMACRO** String

The MODEMACRO variable shows a text string or DIESEL expression on the status line. This string reveals information like the name of the current drawing, time/ date stamp, or special modes. The initial value of this variable is "".

**MTEXTED** String

The MTEXTED variable sets the name of the program to be used for the editing of mtext objects. This variable is saved in registry and its initial value is "Internal".

**NOMUTT** Short

The NOMUTT variable suppresses the message display (muttering) when it wouldn't normally be suppressed. Its initial value is 0 and it is not saved. 0 - Resumes normal muttering behavior, 1 - Suppresses muttering indefinitely.

**OBSCUREDColor** Integer

This variable is used to set the color and visibility of the obscured lines. The obscured lines are the hidden lines in the solid or surface models that are made visible only when you invoke the **HIDE** or the **SHADEMODE** command. Initial value is 0 and this means that the visibility is set to off. Specify any color number as the value of this variable to set the visibility on.

**OBSCUREDType** Integer

This variable is used to set the linetype for the obscured lines. Initial value for this variable is 0. The other possible values are:

- 1 Solid
- 2 Dashed
- 3 Dotted
- 4 Short Dash
- 5 Medium Dash
- 6 Long Dash
- 7 Double Short Dash
- 8 Double Medium Dash
- 9 Double Long Dash
- 10 Medium Long Dash
- 11 Sparse Dot

**OFFSETDIST** Real

This variable sets the default offset distance. If the value of this variable is less than zero then the offset distance can be specified with the through mode. If the value of this variable is greater than zero then the default offset distance is established. The initial value of this variable is 1.0000.

**OFFSETGAPType** Integer

The OFFSETGAPType variable controls how

to offset polylines when a gap is created as a result of offsetting the individual polyline segments. Its initial value is 0 and it is stored in registry. 0 - Extends the segments to fill the gap, 1 - Fills the gaps with a filleted arc segment where radius is the offset distance, 2 - Fills the gaps with a chamfered line segment.

**OLEHIDE** Integer

Controls the display of OLE objects in AutoCAD. It is saved in registry and its initial value is 0. 0 - All OLE objects are visible, 1 - OLE objects are visible in paper space, 2 - OLE objects are visible in model space, 3 - No OLE objects are visible.

**OLEQUALITY** Integer

The OLEQUALITY variable controls the default quality level for embedded OLE objects. Its initial value is 1 and it is stored in registry. 0 - Line art quality, such as an embedded spreadsheet, 1 - Text quality, such as an embedded Word document, 2 - Graphics quality, such as an embedded pie chart, 3 - Photograph quality, 4 - High quality photograph.

**OLESTARTUP** Integer

The OLESTARTUP variable controls whether the source application of an embedded OLE object loads when plotting. Its initial value is 0 and it is saved in drawing. 0 - Does not load the OLE source application, 1 - Loads the OLE source application when plotting.

**ORTHOMODE** Integer

The ORTHOMODE variable governs the orthogonal display of lines or polylines. This variable is saved in the drawing and its initial value is 0. 0 - The Ortho mode is turned off, 1 - The Ortho mode is turned on.

**OSMODE** Integer

The OSMODE variable sets the running Object Snap modes using the following bit-codes: 0 - NONE object snap, 1 - ENDpoint object snap, 2 - MIDpoint object snap, 4 - CENTER object snap, 8 - NODE object snap, 16 -

QUAdrant object snap, 32 - INTERsection object snap, 64 - INSertion object snap, 128 - PERpendicular object snap, 256 - TANGent object snap, 512 - NEArest object snap, 1024 - QUIck object snap, 2048 - APPint object snap. If you want to specify more than one object snap, enter the sum of their values. For example, if you want to specify the node and center object snaps, enter  $4+8 = 12$  as the value for the OSMODE variable. This variable is saved in the drawing and its initial value is 0.

**OSNAPCOORD** Integer

Controls whether coordinates entered on the command line override running object snaps. 0 - Running object snap settings override keyboard entry, 1 - Keyboard entry overrides object snap settings, 2 - (Initial value) Keyboard entry overrides object snap setting except in scripts.

**PAPERUPDATE** Integer

The PAPERUPDATE variable 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. Its initial value is 0 and it is saved in registry. 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** Integer

The PDMODE variable sets Point Object Display mode. This variable is saved in the drawing and its initial value is 0.

**PDSIZE** Real

The PDSIZE variable sets the display size of the point object. This variable is saved in the drawing and its initial value is 0.0000. 0 - For this value, point is created at 5 percent of the graphics height, >0 - In this case the value entered specifies the absolute size, <0 - In

this case the value entered specifies the percentage of the viewport size.

**PEDITACCEPT** Integer

The PEDITACCEPT variable controls the display of the **Object Selected Is Not a Polyline** prompt in the PEDIT command. The prompt is followed by “**Do you want it to turn into one?**” Entering y converts the selected object to a polyline. When the prompt is suppressed, the selected object is automatically converted to a polyline. Initial value of this variable is 0. 0 The prompt is displayed, 1 The prompt is suppressed

**PELLIPSE** Integer

The PELLIPSE controls the type of ellipse created with the ELLIPSE command. This variable is saved in the drawing and its initial value is 0. 0 - A true ellipse object is drawn, 1 - A polyline representation of an ellipse is drawn.

**PERIMETER** Real

The PERIMETER variable holds the most recently perimeter value computed by AREA, DBLIST, or LIST commands. This variable is a read-only variable.

**PFACEVMAX** Integer

The PFACEVMAX variable sets the maximum number of vertices per face. This variable is a read-only variable.

**PICKADD** Integer

The PICKADD variable controls additive selection of objects. This variable is saved in registry and its initial value is 1. 0 - PICKADD variable is disabled, 1 - PICKADD variable is enabled. All the objects selected by any method are added to the selection set. If you want to remove objects from the selection set, hold down the Shift key and select the objects.

**PICKAUTO** Integer

PICKAUTO variable controls the automatic windowing feature when the “Select object” prompt appears. This variable is saved in registry and its initial value is 1. 0 - PICKAUTO variable is disabled, 1 - PICKAUTO variable is enabled and a selection window is automatically drawn at the Select objects prompt.

**PICKBOX** Integer

The PICKBOX variable sets the object selection target half height (in pixels). This variable is saved in registry and its initial value is 3.

**PICKDRAG** Integer

The PICKDRAG variable governs the method of drawing a selection window: 0 - For this value the selection window is drawn by clicking the pointing device at one corner and then clicking again at the other corner of the window, 1 - For this value the selection window is drawn by clicking the pointing device at one corner, holding down the pick button, dragging the cursor, and finally releasing the pick button of the pointing device at the other corner of the window. This variable is saved in registry and its initial value is 0.

**PICKFIRST** Integer

The PICKFIRST variable governs the method of object selection in such a manner that you can first select the object and then specify the desired edit or inquiry command. Its initial value is 1. 0 - PICKFIRST variable disabled, 1 - PICKFIRST variable enabled.

**PICKSTYLE** Integer

The PICKSTYLE variable controls the associative hatch selection and group selection. This variable is saved in the drawing and its initial value is 1. 0 - Associative hatch selection and group selection not possible, 1 - Group selection possible, 2 - Associative hatch selection possible, 3 - Associative hatch selection and group selection possible.

**PLATFORM** String

The PLATFORM variable specifies the platform of AutoCAD that is in use. This is a read-only variable. Some of the platforms are: Microsoft Windows - Sun/SPARCstation, 386 DOS Extender - DECstation, Apple Macintosh - Silicon Graphics Iris Indigo.

**PLINEGEN** Integer

The PLINEGEN variable sets the linetype pattern generation around the vertices of a 2D polyline. This variable does not affect polylines with tapered segments. This variable is saved in the drawing and its initial value is 0. 0 - Polylines are generated with a dash at each vertex, 1 - Linetype is created in a continuous pattern around the vertices of the polyline.

**PLINETYPE** Integer

Specifies whether AutoCAD uses optimized 2D polylines. 0 - PLINE creates old format polylines, 1 - PLINE creates optimized polylines, 2 - PLINE creates optimized polylines and the polylines in older drawings are converted on open.

**PLINEWID** Real

The default polyline width is stored in this variable. This variable is saved in the drawing and its initial value is 0.0000.

**PLOTID** String

The PLOTID variable stores the current plotter's description. The plotter configuration can be changed by entering the plotter's full or partial description. This variable is saved in registry and its initial value is "".

**PLOTROTMODE** Integer

The PLOTROTMODE variable controls the orientation of plots. This variable is saved in the drawing and its initial value is 1. 0 - The effective plotting area is rotated in order to align the corner with the Rotation icon with

the paper at the lowerleft for a rotation of 0, top-left for a rotation of 90, top-right for a rotation of 180, and lower left for a rotation of 270, 1 - The lower-left corner of the effective plotting area is aligned with the lower-left corner of the paper.

**PLOTTER** Integer

The PLOTTER variable stores an integer number assigned for configured plotter. This integer number can be in the range of 0 to the number of configured plotters. You can change to some other configured plotter by entering the integer number assigned to the plotter. If you have 6 plotters, the valid numbers are 0, 1, 2, 3, 4, 5. This variable is saved in registry and its initial value is 0.

**PLQUIET** Integer

The PLQUIET variable controls the display of optional dialog boxes and nonfatal errors for batch plotting and scripts. Its initial value is 0 and it is stored in registry. 0 - Displays plot dialog boxes and nonfatal errors, 1 - Logs nonfatal errors and doesn't display plot-related dialog boxes.

**POLARADDANG** String

The POLARADDANG variable contains all user-defined polar angles. You can add up to 10 angles. Each angle can be up to 25 characters, separated with semicolons (;). Its initial value is null and it is stored in registry.

**POLARANG** Real

The POLARANG variable controls the polar angle increment. Values are 90, 45, 30, 22.5, 18, 15, 10, and 5. Its initial value is 90 and is saved in registry.

**POLARDIST** Real

The POLARDIST variable controls the snap increment when the SNAPSTYL system variable is set to 1 (polar snap). Its initial value is 0.0000 and it is stored in registry.

**POLARMODE** Integer

The POLARMODE variable Controls settings for polar and object snap tracking. Its initial value is 1 and it is saved in registry. 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** Integer

The POLYSIDES variable establishes the default number of sides for a polygon. This variable can take values in the range of 3 to 1024. The initial value of this variable is 4.

**POPUPS** Integer

The POPUP variable shows the status of the presently configured display driver. This is a read-only variable. 0 - The dialog boxes, menu bar, pull-down menus, and icon menus are not supported, 1 - The dialog boxes, menu bar, pull-down menus, and icon menus are supported.

**PRODUCT** String

This variable displays the name of the product which is AutoCAD.

**PROGRAM** String

This variable displays the name of the program which is acad.

**PROJECTNAME** String

Stores the current project name. Initial value is “ ”.

**PROJMODE** Integer

The PROJMODE variable establishes the current Projection mode for Extend or Trim operations. Its initial value is 1. 0 - True 3D mode

established (no projection), 1 - Projection to XY plane of the current UCS, 2 - Projection to current view plane.

**PROXYGRAPHICS** Integer

Specifies whether images of proxy objects are saved in the drawing. The initial value is 1. 0 - Image is not saved, 1 - Image is saved.

**PROXYNOTICE** Integer

Displays a notice when you open a drawing containing custom objects created by an application that is not present. The initial value is 1. 0 - No proxy warning displayed, 1 - Proxy warning displayed.

**PROXYSHOW** Integer

Controls the display of proxy objects in a drawing. The initial value is 1. 0 - Proxy objects are not displayed, 1 - Graphic images are displayed for all proxy objects, 2 - Only bounding box is displayed for all proxy objects.

**PROXYWEBSEARCH** Integer

This variable is used to specify how AutoCAD checks for Object Enablers. Initial value is 1. 0 - Prevents AutoCAD from checking for Object Enablers regardless of your settings in the **AutoCAD 2002 Today** window, 1 - AutoCAD checks for Object Enablers only if **Autodesk Point A** is open in the **AutoCAD 2002 Today** window. It is not necessary for the Today window to be open, 2 - Specifies the number of times AutoCAD will continue to check for Object Enablers after unsuccessful attempts.

**PSLTSCALE** Integer

The PSLTSCALE variable governs the paper space linetype scaling. This variable is saved in the drawing and its initial value is 1. 0 - Special linetype scaling not allowed. Linetype dash lengths depend on the drawing units of the space in which the objects were drawn, 1 - Linetype scaling governed by viewport scaling. In case TILEMODE is set to 0, dash



lengths depend on the paper space drawing units, even if objects are in model space.

**PSTYLEMODE** Read only  
The PSTYLEMODE variable indicates whether the current drawing is in a Color-Dependent or Named Plot Style mode. Its initial value is 0 and it is saved in drawing. 0 - Uses named plot style tables in the current drawing, 1 - Uses color-dependent plot style tables in the current drawing.

**PSTYLEPOLICY** Integer  
The PSTYLEPOLICY variable controls whether an object's color property is associated with its plot style. The new value you assign affects only newly created drawings. Its initial value is 1 and it is saved in registry. 0 - No association is made between color and plot style. 1 - An object's plot style is associated with its color.

**PSVPSCALE** Real  
The PSVPSCALE variable controls 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. A value of 0 means the scale factor is "Scaled to Fit". A scale must be a positive real value. Its initial value is 0 and it is not saved.

**PUCSBASE** String  
The PUCSBASE variable stores the name of the UCS that defines the origin and orientation of orthographic UCS settings in paper space only. Its initial value is "" and it is stored in drawing.

**QTEXTMODE** Integer  
The QTEXTMODE controls the Quick Text mode. This variable is saved in the drawing and its initial value is 0. 0 - The Quick Text mode is turned off and characters are dis-

played, 1 - The Quick Text mode is turned on and a box instead of text is displayed.

**RASTERPREVIEW** Integer  
The RASTERPREVIEW variable determines whether the drawing preview images are saved with the drawing and in which format they will be saved. This variable is saved in the registry and its initial value is 1. 0 - No preview image created, 1 - Preview image is created.

**REFEDITNAME** String  
The REFEDITNAME variable indicates whether a drawing is in a reference-editing state; also, stores the reference file name. Its initial value is "" and it is not saved. This is a read-only variable.

**REGENMODE** Integer  
The REGENMODE variable controls the automatic regeneration of the drawing. This variable is saved in the drawing and its initial value is 1. 0 - REGENAUTO is turned off, 1 - REGENAUTO is turned on.

**RE-INIT** Integer  
The RE-INIT variable reinitializes the I/O ports, plotter, digitizer, display, and acad.pgp file. The following bit-codes are used for this process:  
0 - Reinitialization not allowed (initial value),  
1 - Reinitialization of digitizer port,  
2 - Reinitialization of plotter port,  
4 - Reinitialization of digitizer,  
8 - Reinitialization of display,  
16 - Reinitialization of PGP file. You can specify more than one reinitialization by entering the sum of the values of the desired reinitializations.

**REMEMBERFOLDERS** Integer  
This variable is used to control the default path for the **Select File** and the **Save** dialog boxes. 0 - The start in path for the shortcut icon is used for all open and save dialog boxes,



1 - The last path used is saved and used later.

**RTDISPLAY** Integer

Controls the display of raster images during realtime zoom or pan. 0 - Displays raster image content, 1 - Displays raster image outline only (initial value).

**SAVEFILE** String

The present auto-save filename is held in the SAVEFILE variable. This variable is a read-only variable. The initial value is "auto.sv\$".

**SAVEFILEPATH** String

The SAVEFILEPATH variable specifies the path to the directory for all automatic save files for the AutoCAD session. Its initial value is "C:\Temp\" and it is saved in registry.

**SAVENAME** String

You can save the current drawing to a different name and this name is held in the SAVENAME variable. This variable is a read-only variable.

**SAVETIME** Integer

AutoCAD has provided the facility of automatically saving your work at specific intervals. You can specify the automatic save time intervals (in minutes) with the SAVETIME variable. This variable is set to an initial value of 120. 0 - Automatic save facility is disabled, 0 - The drawing is saved according to the intervals specified. Once you make changes to the drawing, the SAVETIME timer starts. SAVE, SAVEAS, or QSAVE commands reset and restart this timer. AutoCAD saves the drawing under the filename auto.sv\$.

**SCREENBOXES** Integer

The SCREENBOXES variable stores the number of boxes in the screen menu area of the graphics area. In case the screen menu is disabled, this variable is set to zero. The value of this variable is susceptible to change during an editing session on platforms that allow the AutoCAD graphics window to be resized or the screen menu to be reconfigured

while you are in an editing session. This is a readonly variable.

**SCREENMODE** Integer

The SCREENMODE variable holds a bitcode specifying the graphics/text state of the AutoCAD display. This is a read-only variable. Following are the bit values: 0 - Text screen is displayed, 1 - Graphics mode is displayed, 2 - Dual-screen display (text and graphics) is displayed.

**SCREENSIZE** 2D point

This variable holds the current viewport size in pixels. This is a read-only variable.

**SDI** Integer

This variable controls whether AutoCAD runs in single- or multiple-document interface. Its initial value is 0 and it is saved in registry.

0 - Turns on multiple-drawing interface,  
1 - Turns off multiple-drawing interface,  
2 - (Read-only) Multiple-drawing interface is disabled, 3 - (Read-only) Multiple-drawing interface is disabled.

**SHADEDGE** Integer

The SHADEDGE variable governs the shading of edges in rendering. This variable is saved in the drawing and its initial value is 3. 0 - Faces are shaded and edges are not highlighted, 1 - Faces are shaded and edges are drawn in background color, 2 - Faces are not filled and edges are in object color, 3 - Faces are in object color and edges are drawn in background color.

**SHADEIF** Integer

The SHADEIF variable establishes the ratio of diffuse reflective light to ambient light (in percent of diffuse reflective light). This variable is saved in the drawing and its initial value is 70.

**SHORTCUTMENU** Integer

This variable Controls whether Default, Edit,

and Command mode shortcut menus are available in the drawing area. Its initial value is 11 and it is stored in registry. The following bitcodes are used by SHORTCUTMENU: 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.

**SHPNAME** String  
The SHPNAME variable establishes the name of the default shape. The initial value for this variable is "". To set no default enter a period (.).

**SKETCHINC** Real  
The SKETCHINC specifies the record increment for the SKETCH command. This variable is saved in the drawing and its initial value is 0.1000.

**SKPOLY** Integer  
The SKPOLY variable decides whether SKETCH command generates lines or polylines. This variable is saved in the drawing and its initial value is 0. 0 - Lines are generated; 1 - polylines are generated.

**SNAPANG** Real  
The SNAPANG variable specifies the snap/grid rotation angle relative to the UCS for the current viewport. This variable is saved in the drawing and its initial value is 0. Changes to this variable are manifested only after a redraw is performed.

**SNAPBASE** 2D point  
The SNAPBASE variable specifies the snap/

grid origin point (in UCS X, Y coordinates) for the current viewport. This variable is saved in the drawing and its initial value is 0.0000,0.0000. Changes to this variable are manifested only after a redraw is performed.

**SNAPISOPAIR** Integer  
The SNAPISOPAIR variable controls the current isometric plane for the current viewport. This variable is saved in the drawing and its initial value is 0. 0 - Left, 1 - Top, 2 - Right.

**SNAPMODE** Integer  
The SNAPMODE variable controls the Snap mode. This variable is saved in the drawing and its initial value is 0. 0 - Snap disabled; 1 - Snap enabled for the current viewport.

**SNAPSTYL** Integer  
The SNAPSTYL variable establishes the snap style for the current viewport. This variable is saved in the drawing and its initial value is 0. 0 - Standard; 1 - Isometric.

**SNAPTYPE** Integer  
This variable the snap style for the current viewport. Its initial value is 0 and it is saved in registry. 0 - Grid, or standard snap; 1 - Polar snap.

**SNAPUNIT** 2D point  
The SNAPUNIT variable specifies the X and Y snap spacing for the current viewport. This variable is saved in the drawing and its initial value is 0.5000,0.5000. The changes to this variable are manifested only after a redraw is performed.

**SOLIDCHECK** Integer  
This variable turns the solid validation on and off for the current AutoCAD session. Its initial value is 1 and it is not saved. 0 - Turns off solid validation; 1 - Turns on solid validation.

**SORTENTS** Integer  
The SORTENTS variable governs the display

of object sort order operations using the following values: 0 - SORTENTS is disabled, 1 - Sorts for object selection, 2 - Sorts for object snap, 4 - Sorts for redraw, 8 - Sorts for MSLIDE slide creation, 16 - Sorts for regens, 32 - Sorts for plotting, 64 - Sorts for PostScript output. More than one options can be selected by specifying the sum of the values of these options. Its initial value is 96. This value specifies sort operations for plotting and PostScript output.

**SPLFRAME** Integer

The SPLFRAME variable governs the display of spline-fit polylines. This variable is saved in the drawing and its initial value is 0. 0 - The control polygon for spline fit polylines is not displayed. The fit surface of a polygon mesh is displayed while as the defining mesh is not displayed. Also invisible edges of 3D faces or polyface meshes are not displayed, 1 - The control polygon for spline fit polylines is displayed. The fit surface of a polygon mesh is not displayed while as the defining mesh is displayed. Also invisible edges of 3D faces or polyface meshes are displayed.

**SPLINESEGS** Integer

The SPLINESEGS variable governs the number of line segments used to construct each spline. Hence with this variable you can control the smoothness of the curve. This variable is saved in the drawing and its initial value is 8. With this value a reasonably smooth curve is generated which does not need a much regeneration time. The greater the value of this variable, the smoother the curve and greater the regeneration time and the space occupied by the drawing file.

**SPLINETYPE** Integer

The SPLINETYPE variable specifies the type of spline curve that will be generated by Spline option of PEDIT command. This variable is saved in the drawing and its initial value is 6. 5 - Quadratic B-spline is generated, 6 - Cu-

bic B-spline is generated.

**SURFTAB1** Integer

The SURFTAB1 variable governs the number of intervals (tabulated surfaces) to be generated for TABSURF and RULESURF commands along the path curve. This variable also defines the mesh density in the M direction for REVSURF and EDGESURF commands. This variable is saved in the drawing and its initial value is 6. In case the path curve is a line, arc, circle, spline-fit polyline, or an ellipse, the path curve is divided into intervals equal to the value of SURFTAB1 by the tabulation lines. Else, if the path curve is a polyline (not spline-fit), the tabulation lines are generated at the ends of the polyline segments and if there are any arc segments, each segment is divided into intervals equal to the value of SURFTAB1 by the tabulation lines.

**SURFTAB2** Integer

The SURFTAB2 variable defines the mesh density in the N direction for REVSURF and EDGESURF commands. This variable is saved in the drawing and its initial value is 6.

**SURFTYPE** Integer

The SURFTYPE variable governs the type of surface-fitting to be performed by the Smooth option of PEDIT command. This variable is saved in the drawing and its initial value is 6. 5 - Quadratic B-spline surface, 6 - Cubic B-spline surface, 8 - Bezier surface.

**SURFU** Integer

The SURFU variable specifies the surface density of polygon meshes in the M direction. This variable is saved in the drawing and its initial value is 6.

**SURFV** Integer

The SURFV variable specifies the surface density of polygon meshes in the N direction. This variable is saved in the drawing and its initial value is 6.

**SYSCODEPAGE** String

The SYSCODEPAGE variable expresses the system code page specified in the acad.xml file. This variable is a read-only variable and is saved in the drawing.

**TABMODE** Integer

The TABMODE variable governs the use of Tablet mode. 0 - Tablet mode disabled (initial value), 1 - Tablet mode enabled.

**TARGET** 3D point

The TARGET variable holds the position of the target point (in UCS coordinates) for the current viewport. This is a read-only variable and is saved in the drawing.

**TDCREATE** Real

The TDCREATE variable holds the creation time and date of a drawing. This is a read-only variable and is saved in the drawing.

**TDINDWG** Real

The TDINDWG variable holds the total editing time. This is a read-only variable and is saved in the drawing.

**TDUCREATE** Real

This variable stores the universal time and date the drawing was created. This is a read-only variable and saved in drawing.

**TDUPDATE** Real

The TDUPDATE variable holds the time and date of most recent update/save. This is a read-only variable and is saved in the drawing.

**TDUSRTIMER** Real

The TDUSRTIMER variable stores the user elapsed timer. This is a read-only variable and is saved in the drawing.

**TDUUPDATE** Real

This variable stores the universal time and

date of the last update/save. This is a read-only variable and stored in drawing.

**TEMPPREFIX** String

The TEMPPREFIX variable stores the directory name configured for the placement of temporary files. The path separator is included. This is a read-only variable.

**TEXTEVAL** Integer

The TEXTEVAL variable determines the procedure of evaluation of text strings. The initial value for this variable is 0. 0 - All the responses to prompts for text strings and attributes are accepted as literals, 1 - In case the starting character of the text string is “(“ or “!”, it is treated as an AutoLISP expression. The TEXTEVAL setting does not affect the TEXT command. TEXT command accepts all input as literals.

**TEXTFILL** Integer

The TEXTFILL variable governs the filling of TrueType fonts. This variable is saved in the registry and has an initial value of 1. 0 - The text is displayed as outlines, 1 - The text is displayed as filled images.

**TEXTQLTY** Real

The TEXTQLTY variable defines the resolution of TrueType, Bitstream, and Adobe Type 1 fonts. The higher the value of this variable, the higher the resolution and lower the display and plotting speed. On the other hand the lower the value of this variable, the lower the resolution and higher the display and plotting speed. This variable is saved in the drawing and can take values in the range of 0 to 100.0, its initial value is 50.

**TEXTSIZE** Real

The TEXTSIZE variable controls the text height of the text drawn with the current text style. But this is possible only if the style does not have a fixed height. This variable is saved in the drawing and its initial value is 0.2000.

**TEXTSTYLE**

String

The TEXTSTYLE variable stores the name of the current text style. This variable is saved in the drawing and its initial value is STANDARD.

**THICKNESS**

Real

The THICKNESS variable defines the current 3D thickness. This variable is saved in the drawing and its initial value is 0.0000.

**TILEMODE**

Integer

The TILEMODE variable governs entry into paper space and also how the AutoCAD viewports act. This variable is saved in the drawing and its initial value is 1. 0 - The paper space and viewport objects are enabled. The graphics area is cleared and you are prompted to use the MVIEW command to define viewports, 1 - Release 10 Compatibility mode is enabled. Automatically you are taken into Tiled Viewport mode and previously active viewport configuration is restored on the screen. Paper space objects including viewport objects are not displayed. MSPACE, PSPACE, VPLAYER, and MVIEW commands are disabled.

**TOOLTIPS**

Integer

The TOOLTIPS variable determines the display of ToolTips. Its initial value is 1. 0 - The display of ToolTips is turned off, 1 - The display of ToolTips is turned on.

**TRACEWID**

Real

The TRACEWID variable establishes default value for the width of the trace. This variable is saved in the drawing and its initial value is 0.0500.

**TRACKPATH**

Integer

This variable controls the display of polar and object snap tracking alignment paths. Its initial value is 0 and it is saved in registry. 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**

Integer

The TREEDEPTH variable specifies how many times the tree-structured spatial index may divide into branches. This variable is saved in the drawing and its initial value is 3020. 0 -The spatial index is totally suppressed. In this case the objects are processed in database order and hence it is not necessary to set the SORTENTS variable, >0 - TREEDEPTH variable is enabled. You can enter an integer of up to four digits. The first two digits indicate the depth of model space nodes and the second two digits indicate the depth of paper space nodes, <0 - If the value is negative then the model space objects are treated as 2D objects. Negative values are relevant for 2D drawings. This way memory is more efficiently utilized and there is no trade-off with the performance.

**TREEMAX**

Integer

The TREEMAX variable sets the limit to the maximum number of nodes in the spatial index. This way the memory use during regeneration of drawing is limited. Its initial value is 10000000.

**TRIMMODE**

Integer

The TRIMMODE variable determines whether selected edges for chamfers and fillets will be trimmed. 0 - Selected edges are not trimmed after chamfering and filleting, 1 - Selected edges are trimmed after chamfering and filleting (initial value).

**TSPACEFAC**

Real

This variable controls the multiline text line

spacing distance measured as a factor of text height. The valid values are 0.25 to 4.0. The initial value is 1 and it is not saved.

**TSPACETYPE** Integer

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. Its initial value is 1 and it is not saved. 1 - At Least, 2 - Exactly.

**TSTACKALIGN** Integer

This variable controls the vertical alignment of stacked text. Its initial value is 1 and it is saved in drawing.

0 - Bottom aligned, 1 - Center aligned, 2 - Top aligned.

**TSTACKSIZE** Integer

This variable controls the percentage of stacked text fraction height relative to selected text's current height. The valid value ranges from 1 to 127. Its initial value is 70 and it is saved in drawing.

**UCSAXISANG** Integer

This variable stores the default angle when rotating the UCS around one of its axes using the X, Y, or Z options of the UCS command. The values must be entered as an angle in degrees (valid values are: 5, 10, 15, 18, 22.5, 30, 45, 90, 180). Its initial value is 90 and it is stored in registry.

**UCSBASE** String

This variable stores the name of the UCS that defines the origin and orientation of orthographic UCS settings. The valid values include any named UCS. Its initial value is "World" and is stored in drawing.

**UCSFOLLOW** Integer

The UCSFOLLOW variable controls the automatic displaying of a plan view when you switch from one UCS to another. All the viewports have the UCSFOLLOW facility and

hence you need to specify the UCSFOLLOW setting separately for each viewport. This variable is saved in the drawing and its initial value is 0. 0 - Switch from one UCS to another, does not alter the view, 1 - Plan view of the new UCS is automatically displayed when you switch from one UCS to another.

**UCSICON** Integer

The UCSICON variable displays the present UCS icon using bit-code for the current viewport. The value of this variable is the sum of the following:

1 - Icon display is enabled, 2 - The icon moves to the UCS origin if the icon display is enabled. In case more than one viewport is active, each of the viewport can have a different value for the UCSICON variable. If you are in paper space, the UCSICON variable will contain the setting for the UCS icon of the paper space. This variable is saved in the drawing and its initial value is 1.

**UCSNAME** String

The UCSNAME variable contains the name of the current UCS. This is a read-only variable and is saved in the drawing. In case the current UCS is unnamed, then a null string is returned.

**UCSORG** 3D point

The coordinate value of the origin of the current UCS is held in the UCSORG variable. This is a read-only variable and is saved in the drawing.

**UCSORTHO** Integer

This variable Determines whether the related orthographic UCS setting is restored automatically when an orthographic view is restored. Its initial value is 1 and it is stored in registry. 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** Integer

This variable determines whether the current UCS is saved with a named view. Its initial value is 1 and is saved in registry. 0 - Does not save current UCS with a named view, 1 - Saves current UCS whenever a named view is created.

**UCSVP** Integer

The UCSVP variable determines whether the UCS in active viewports remains fixed or changes to reflect the UCS of the currently active viewport. Its initial value is 1 and it is stored in drawing (Viewport specific). 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** 3D point

The X axis direction of the current UCS for the current space is held in UCSXDIR variable. This is a read-only variable and is saved in the drawing.

**UCSYDIR** 3D point

The Y axis direction of the current UCS for the current space is held in UCSYDIR variable. This is a read-only variable and is saved in the drawing.

**UNDOCTL** Integer

The UNDOCTL variable holds a bit-code expressing the state of the UNDO command. This is a read-only variable. The value of this variable is the addition of following values: 0 - UNDO command is disabled, 1 - UNDO command is enabled, 2 - Just one command can be undone, 4 - Auto-group mode is enabled, 8 - Some group is presently active.

**UNDOMARKS** Integer

The UNDOMARKS variable contains the number of marks that have been put in the UNDO command's control stream by the Mark option. In case a group is presently active, the Mark and Back options cannot be accessed. This variable is a read-only variable.

**UNITMODE** Integer

The UNITMODE variable governs the units display format. This variable is saved in the drawing and its initial value is 0. 0 - The fractional, feet and inches, and surveyor's angles are displayed as previously defined, 1 - The fractional, feet and inches, and surveyor's angles are displayed in the input format. This variable is saved in the drawing and its initial value is 0.

**USERI1-5** Integer

Stores and retrieves integer values. Initial value is 0.

**USERR1-5** Integer

Stores and retrieves real numbers. Initial value is 0.0000.

**USERS1-5** Integer

Stores and retrieves text string data. Initial value is “ ”.

**VIEWCTR** 3D point

The VIEWCTR variable stores the center of view in the current viewport, defined in the UCS coordinates. This variable is a read-only variable and is saved in the drawing.

**VIEWDIR** 3D vector

The VIEWDIR variable contains the viewing direction in the current viewport expressed in the UCS coordinates. The camera position is expressed as a 3D offset from the target position. This variable is a read-only variable and is saved in the drawing.

**VIEWMODE** Integer

The VIEWMODE variable governs Viewing



mode for the current viewport using bit-code. The value for this variable is the addition of the following bit values: 0 - Viewing mode disabled, 1 - Perspective view active, 2 - Front clipping on, 4 - Back clipping on, 8 - UCS Follow mode on, 16 - Front clip not at eye. In case it is on, the front clipping plane is determined by the front clip distance stored in the FRONTZ variable. If it is off, the front clipping plane passes through the camera point and in this case FRONTZ variable is not taken into consideration. If the front clipping bit (2) is off then this flag is neglected. This variable is a read-only variable and is saved in the drawing.

**VIEWSIZE** Real

The VIEWSIZE variable contains the view height in the current viewport and is defined in the drawing units. This variable is a read-only variable and is saved in the drawing.

**VIEWTWIST** Real

The VIEWTWIST variable contains the view twist angle for the current viewport. This variable is a read-only variable and is saved in the drawing.

**VISRETAIN** Integer

The VISRETAIN variable determines whether changes to the visibility of layers in xref are saved in the current drawing. 0 - Changes to On/Off, Freeze/Thaw, color, and linetype settings for the xref-dependent layers are not saved in the current drawing, 1 - Changes to the xref layer definitions in the current drawing are saved with the current drawing.

**VSMAX** 3D point

The VSMAX variable contains the upper right corner of the virtual screen of the current viewport and is expressed in UCS coordinates. This variable is a read-only variable and is saved in the drawing.

**VSMIN** 3D point

The VSMIN variable contains the lower left corner of the virtual screen of the current viewport and is expressed in UCS coordinates. This variable is a read-only variable and is saved in the drawing.

**WHIPARC** Integer

This variable controls whether the display of circles and arcs is smooth. Its initial value is 0 and it is saved in registry. 0 - Circles and arcs are not smooth, but rather are displayed as a series of vectors, 1 - Circles and arcs are smooth, displayed as true circles and arcs.

**WHIPTHREAD** Integer

Controls whether or not to use the additional processor (also called multithread processing) for improving the performances of the commands that lead to redrawing or regeneration of drawings. 0 - Multithread processing is not used, 1 - Only for regeneration, 2 - Only for redrawing, 3 - For regeneration and redrawing both.

**WMFBKGND** Integer

The WMFBKGND variable 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 or Dragged and dropped from AutoCAD as a Windows metafile. Its initial value is 1 and the values are not saved. 0 - The background is transparent, 1 - The background color is the same as the AutoCAD current background color.

**WMFFOREGND** Integer

Controls the assignment of the foreground color of AutoCAD objects in the windows metafile.

**WORLDUCS** Integer

The WORLDUCS variable expresses whether

the UCS is the same as the WCS. This variable is a read-only variable. 0 - Current UCS and WCS are different, 1 - Current UCS and WCS are not different.

**WORLDVIEW** Integer

The WORLDVIEW variable determines whether UCS changes to WCS during DVIEW or VPOINT commands. This variable is saved in the drawing and its initial value is 1. 0 - Current UCS is not changed, 1 - Current UCS is changed to WCS till the DVIEW or VPOINT command is in progress. The DVIEW and VPOINT command input is with respect to the current UCS.

**WRITESTAT** Read-only

This variable indicates whether a drawing file is read-only or can be written to. For developers who need to determine write status through AutoLISP. Its initial value is 1 and values are not saved. 0 - Can't write to the drawing, 1 - Can write to the drawing.

**XCLIPFRAME** Integer

This variable controls visibility of xref clipping boundaries and its initial value is 0. 0 - Clipping boundary is not visible, 1 - Clipping boundary is visible.

**XEDIT** Integer

This variable controls whether the current drawing can be edited in-place when being referenced by another drawing. Its initial value is 1 and it is saved in drawing. 0 - Can't use in-place reference editing, 1 - Can use in-place reference editing.

**XFADECTL** Integer

This variable controls the fading intensity for references being edited in-place. Its initial value is 50 and it is stored in registry. 0 - 0 percent fading, minimum value, 90 - 0 percent fading, maximum value

**XLOADCTL** Integer

Turns demand load on and off and controls

whether it loads the original drawing or a copy. Initial value is 1. 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 reference file is opened.

**XLOADPATH** String

Creates a path for storing temporary copies of demand-loaded xref files. Initial value is “.”.

**XREFCTL** Integer

The XREFCTL variable determines whether AutoCAD writes .xlg files (external reference log files). This variable is saved in registry and its initial value is 0. 0 - Xref log files are not written, 1 - Xref log files are written.

**ZOOMFACTOR** Integer

This value 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. Its initial value is 10 and it is stored in registry.