AutoCAD System Variable

ACADPREFIX	Lists the full directory path specified by the ACAD environment variable (Read Only)
ACADVER	Displays the AutoCAD version number (Read Only)
AFLAGS	Bit-code for the ATTDEF command settings. This variable will display the following numbers: 0=no attribute mode selected 1=Invisible selected 2=Constant mode selected 4=Verify mode selected 8=Preset mode selected
ANGBASE	Angle measurement 0 direction with respect to the current UCS. This is normally set to 0 angle by default, i.e. 0 angle is toward the east in the World Coordinate system. This should not be changed for architectural drawings.
ANGDIR	Indicates direction of positive angular direction with respect to the current UCS. This should not be changed for architectural drawings. 0=counterclockwise (the default). 1=clockwise
APERTURE	Object snap target height in pixels. This is set to 10 by default, **but should be changed to 7.**
AREA	Displays the last area computed by commands AREA, LIST, or DBLIST (Read Only)
ATTDIA	Controls the use of a dialogue box for entry of attribute values. 0=disables dialogue box and simply issues prompts to the command line 1=enables dialogue box to enter attribute values
ATTMODE	Controls the attribute display mode. 0=attribute display turned off for all attributes 1=attribute display is normal (visibility is set with atribute) 2=attribute display is turned on for all attributes
ATTREQ	Controls request for attribute values during insertion of a block which has been assigned attributes. 0=does not request values but assumes defaults upon insertion of the block 1=requests values for attribute upon insertion of the block
AUDITCTL	Controls whether an audit report file (*.adt) is created by the AUDIT command 0=disables creation of audit report file 1=enables creation of audit report file
AUNITS	Angular units mode 0=decimal degrees 1=degrees/minutes/seconds 2=grads 3=radians 4=surveyor's units

AUPREC	Angular units decimal places <whole number>
BACKZ	Perpendicular distance from back clipping plane to target plane for the current viewport. Useful only if the back clipping plane is turned on (can be turned on or off by VIEWMODE variable). An even more useful dimension would be distance from back clipping plane to camera, which can be derived only by subtracting BACKZ distance from the camera to target distance (Read Only)
BLIPMODE	Controls display of marker blips 0=off 1=on
CDATE	Current calendar date/time in format YYYYMMDD.HHMMSSmsec (Read Only)
CECOLOR	Sets current entity color
CELTYPE	Sets current entity linetype
CHAMFERA	Sets first chamfer distance
CHAMFERB	Sets second chamfer distance
CIRCLERAD	Sets default circle radius -- to set no default, enter 0
CLAYER	Sets current layer
CMDACTIVE	Bit-code indicates what type of command is active: 1=ordinary command 2=ordinary and transparent command 4=script active 8=dialogue box active
CMDDIA	Controls plot command dialogue box use 0=disables dialogue box for plot command 1=enables dialogue box for plot command
CMDECHO	Controls display of prompts and input during execution of a lisp function 0=disables display of prompts and input 1=enables display of prompts and input
CMDNAMES	Displays the name of the command which is currently active (Read Only)
COORDS	Controls appearance of coordinate display on status line 0=coordinate display is updated on pick points only 1=absolute cartesial coordinates of location of cross-hairs is continuously displayed 2=polar coordinates (distance and angle from last point) is continuously displayed when distance and angle is requested by a command
CVPORT	Sets the identification number of the current viewport
DATE	Displays the Julian date/time in format <Julian Date>.<Fraction of Day>
DBMOD	Bit-code indicates status of drawing modification (Read Only)

	1=entity database modified 2=symbol table modified 4=database variable modified 8=window modified 16=view modified
DIASTAT	Displays dialogue box exit status (Read Only) 0=most recent dialogue box exited via "CANCEL" 1=most recent dialogue box exited via "OK"
DIMALT	Dimension alternate units -- default is 0 0=off 1=on
DIMALTD	Dimension alternate unit decimal places -- default is 2, should be set to 0 for SI metric units in millimeters
DIMALTF	Dimension alternate units scale factor: default is 25.4 which is correct for SI millimeter metric units as alternate units (inches converted to millimeters)
DIMAPOST	Dimension alternate unit text suffix -- default is none which OK for SI metric units
DIMASO	Dimension associative dimensions -- default is 1 which is OK 0=disables associative dimensions 1=enables associative dimensions
DIMASZ	Dimension arrow size -- default is 0.18, should be set to 0.09
DIMBLK	Dimension block to use to replace standard arrow design -- default is none
DIMBLK1	Dimension block at first side of dimension to use to replace standard arrow design -- default is none
DIMBLK2	Dimension block at second side of dimension to use to replace standard arrow design -- default is none
DIMCEN	Dimension circle or arc centermark size -- default is 0.09 which is OK If set to a negative number, center mark passes through circle diameter and projects beyond circle by the distance equal to this value
DIMCLRD	Dimension line color -- default is BYBLOCK
DIMCLRE	Dimension extension line color -- default is BYBLOCK
DIMCLRT	Dimension text color -- default is BYBLOCK
DIMDLE	Dimension line extension -- default is 0.0 which is OK for arrows, but should be set to 0.09 if using ticks
DIMDLI	Dimension line increment: this is the distance which each dimension line is automatically spaced from the last one when using the "continue" or "baseline" sub-commands, or when dimension text

	will not fit within the space between extension lines (the next dimension in a string will get offset from that one by the value of this variable) -- default is 0.38, but should be set to 0, unless you intend to use the "baseline" sub-command (which is rare in architectural drawings)
DIMEXE	Dimension extension line extension beyone the dimension line -- default is 0.18, but should be set to 0.09
DIMEXO	Dimension extension line offset: this is the distance from the origin point of the dimension to the end of the extension line -- default is 0.0625 which is OK
DIMGAP	Dimension space between dimension line and text when dimension line is broken to accommodate the dimension text -- default is 0.09 which is OK, since this is never used in architectural drawings -- dimension lines should never be broken for dimension text in architectural drawings
DIMLFAC	Dimension scale factor -- default is 1 This would only change if you were inserting a drawing which is dimensioned at a different scale than 1:1 -- You would set this to the reciprocal of the scale conversion factor as described in the chapter on "Scale" in this text. For instance, say you want to insert a detail to be scaled at 1"=1'-0" into a drawing which will be plotted in model space at 1/4" = 1'-0". The detail drawing would be inserted at a conversion scale factor of 4, so that it appears 4 times the size of the main drawing to be plotted at 1/4"=1'-0". The detail would then be dimensioned, first setting DIMLFAC to 1/4 (that is the reciprocal of the conversion factor. Note that none of this complicated process is necessary if you always plot your drawings in paper space at 1=1 scale, zooming the various viewports to their respective scales.
DIMLIM	Controls the way tolerances are displayed when using tolerances for a dimension 0=off (default) tolerances are displayed as plus-minus number 1=on tolerances are displayed in high value and low value one above the other Tolerances and therefore this variable is rarely used for architectural drawings
DIMPOST	Dimension text prefix, suffix, or both -- default is none If you wanted to have the words "Above Finished Floor" appended to the dimension, while still allowing the dimension as measured by the program to be used and remain associative, you could set the value of this variable to <>Above Finished Floor The two angle brackets together represent the placement of the associative dimension
DIMRND	Dimension rounding value -- default is 0.0, should be set to 0.625, which is 1/16" for architectural drawings
DIMSAH	Allows use of separate arrow blocks for each end of the dimension line 0=off (default) -- when set to off, you are not allowed to use two

	different blocks for the dimensioning arrows 1=on -- when set to on, you may use two different blocks for each end of the dimension line, stored as DIMBLK1 and DIMBLK2
DIMSCALE	Dimension overall feature scale factor -- default is 1.0 but if you want to plot your drawing in model space, you should set this variable to the plot scale factor (e.g. 1/4"=1'-0" plot scale, the plot scale factor is 48), but if you intend to plot in paper space only, set this variable to 0. It is recommended that you always plot in paper space, and that this variable should be set to 0.
DIMSE1	Controls drawing of first extension line -- default is 0 0=off -- extension line is drawn 1=on -- extension line is not drawn
DIMSE2	Controls drawing of second extension line -- default is 0 0=off -- extension line is drawn 1=on -- extension line is not drawn
DIMSHO	Controls display of update to associative dimension as it is being changed by stretching 0=off 1=on (default)
DIMSOXD	Suppresses outside dimension lines -- default is 0, should be set to 1 0=off -- (default) if dimension arrows do not fit between extension lines, the dimension line and arrows will be drawn to the outside of each extension line, -- this is not normally desirable in architectural drawings 1=on -- forces arrows to be drawn toward the inside of extension lines, regardless of whether they will fit, this is a better appearance for most architectural drawings
DIMSTYLE	Dimension style -- default is *UNNAMED (Read Only) -- you can create as many styles as you want by setting variables as you wish, and using the DIM<RET> SAVE<RET>[name] command sequence, with [name] being the name you assign to the dimension style. To use a style that has already been saved, you need to restore it by using the DIM<RET>RESTORE<RET>[name] command sequence
DIMTAD	Dimension text above dimension line -- default is 0, should be set to 1 0=off -- dimension text will be places on the dimension line by breaking the dimension line -- this format is normally used for engineering type drawings, but is not used for architectural drawings 1=on -- dimensions always placed above the dimension line -- this is the one to use for architectural drawings
DIMTFAC	Sets height of tolerance text with relation to main dimension text height -- default is 1.0 Recommended value is 0.75 -- no need to be concerned about this variable, since tolerances are rarely used in architectural drawings.
DIMTIH	Sets orientation of dimension text inside extension lines -- default is 1, should be 0 0=off -- dimension text always parallel with dimension line 1=on -- dimension text always horizontal regardless of dimension line orientation

DIMTIX	Sets whether text is always placed between extension lines -- default is 0, should be 1 0=off -- text will be placed to the right or left side of dimension if it does not fit between extension lines 1=on -- text will always be placed between extension lines regardless of whether it will fit -- this is more desirable for architectural drawings. If it becomes a problem with long dimensions overlapping dimensions, they can always be moved with the DIM<RET>TEDIT<RET> command sequence
DIMTM	Dimension minus tolerance value -- default is 0.0 -- no need to change
DIMTP	Dimension plus tolerance value -- default is 0.0 -- no need to change
DIMTOFL	Sets dimension line to be drawn inside of extension lines, whether or not text is drawn outside of extension lines -- default is 0, should be set to 1 0=off -- will not draw inside dimension line if text is outside of extension lines 1=on -- will force dimension line to always be drawn between extension lines
DIMTOH	Sets orientation of dimension text outside extension lines -- default is 1, should be 0 0=off -- dimension text always parallel with dimension line 1=on -- dimension text always horizontal regardless of dimension line orientation
DIMTOL	Sets tolerance dimensioning -- default is 0, which is OK 0=off -- tolerance dimensioning is not used 1=on -- tolerances set by DIMTM and DIMTP are drawn Tolerances are rare in architectural drawings, and therefore this variable is not normally used
DIMTSZ	Dimension tick size -- default is 0.0 If set to some value other than 0, dimension will have ticks drawn in place of arrows -- if ticks are desired, this value should be set to 0.09. Arrows are recommended.
DIMTVP	Sets dimension text vertical position -- default is 0.0 If set to 1.0 it is equivalent to DIMTAD set to 1 (on); if set to -1.0, it will place dimension text below dimension line
DIMTXT	Dimension text size -- default is 0.18, should be set to 0.09
DIMZIN	Sets zero display with dimension text -- default is 0, should be set to 3 for architectural drawings
DISTANCE	Last distance computed by DIST command (Read Only)
DONUTID	Sets donut inside diameter default -- can be set to 0; is automatically set to ID size of last donut drawn
DONUTOD	Sets donut outside diameter default -- cannot be set to 0; is automatically set to ID size of last donut drawn
DRAGMODE	Controls display of objects when being copies or moved -- default is 2

	0=off -- no dragging 1=on if requested 2=auto -- will show dragging if possible
DRAGP1	Regen-drag input sampling rate -- default = 10
DRAGP2	Fast-drag input sampling rate -- default = 25
DWGCODEPAGE	Set to the system code page when new drawing is created. Can be manually set to any value used by the SYSCODEPAGE variable -- not normally used
DWGNAME	Displays full path drive and directory and name you gave to the drawing at the SAVE or last SAVEAS command. If drawing is not yet named, it has the value of *UNNAMED (Read Only)
DWGPREFIX	Displays full path drive and directory of drawing, but not the name (Read Only)
DWGTITLED	Displays whether drawing has been named yet 0=drawing has not yet been named 1=drawing has been named
DWGWRITE	Sets read-only toggle 0=allows opening of drawing for reading only 1=allows opening of drawing for reading and writing (default)
ELEVATION	Sets current 3d elevation, relative to the current UCS for current space (model space or paper space)
ERRNO	Error code number (Read Only)
EXPERT	Sets level of issuance of "Are you sure?" prompts 0=issues all prompts (default) 1= suppresses "About to regen, proceed?" and "Really want to turn the current layer off?" 2= suppresses preceding prompts and "Block already defined. Redefine it?" and "A drawing with this name already exists. Overwrite it?" 3= suppresses preceding prompts and those issued by the LINETYPE command if you try to load a linetype that is already loaded or create a new linetype in a file that already defines it. 4= suppresses preceding prompts and those issued by the "UCS Save" and "VPORTS Save" if the name you supply already exists 5= suppresses preceding prompts and those issued by the "DIM SAVE" and "DIM OVERRIDE" commands if the dimension style name you supply already exists. Default answer to all suppressed prompts is **Y**
EXTMAX	Upper-right 3d point of drawing extents (Read Only)
EXTMIN	Lower-left 3d point of drawing extents (Read Only)
FILEDIA	Controls use of dialogue boxes for command input

	0= off -- disables file dialogue box use -- can be overridden by placing a ~ (tilde) preceding the command name 1= on -- use file dialogue boxes if possible
FILLETRAD	Sets fillet radius
FILLMODE	Controls display of fill in wide polylines, solids, donuts, arrowheads, and traces 0=fill is turned off, is not visible and will not plot 1=fill is turned on, is visible in views perpendicular to plane of object (plan only) and will plot
FRONTZ	Perpendicular distance from front clipping plane to target plane for the current viewport. Useful only if the front clipping plane is turned on and "front clip not at eye bit" is turned on (these can be turned on or off by VIEWMODE variable). An even more useful dimension would be distance from front clipping plane to camera, which can be derived only by subtracting FRONTZ distance from the camera to target distance (Read Only)
GRIDMODE	Sets display grid on or off 0=grid display turned off 1=grid display turned on
GRIDUNIT	Sets grid spacing for current viewport, X and Y dimensions
GRIPBLOCK	Controls assignment of grips in blocks 0=assigns grip to insertion point (default) 1=assigns grips to entities within the block
GRIPCOLOR	Sets color of non-selected grips, drawn as a box in outline -- default value is 5 (blue)
GRIPHOT	Sets color of selected grips, drawn as a filled box -- default value is 1 (red)
GRIPS	Controls use of selection set grips 0=disables grips 1=enables grips for Stretch, Move, Rotate, Scale, and Mirror modes (default)
GRIPSIZE	Sets size in pixels of grip box -- default is 3, which is OK
HANDLES	Displays Handles setting (Read Only) 0=Handles disabled 1=Handles enabled
HIGHLIGHT	Sets object selection highlighting 0=highlighting of selected objects turned off 1=highlighting of selected objects turned on
HPANG	Sets default hatch pattern angle
HPDOUBLE	Sets whether "U" user-defined hatch patterns will be cross-hatched (doubled) 0=doubling disabled

	1=doubling enabled
HPNAME	Sets default hatch pattern name To set to no default hatch pattern, enter . (that is "dot" or "period")
HPSCALE	Sets default hatch pattern scale factor. Must be non-zero.
HPSPACE	Sets default hatch pattern line spacing for "U" user-defined hatch patterns. Must be non-zero
INSBASE	Insertion Base point in the current coordinate system coordinates
INSNAME	Default Block name for the INSERT command. To set to no default name, enter a . (period)
LASTANGLE	The end angle of the last arc entered (Read Only)
LASTPOINT	The coordinates of the last point entered
LENSLENGTH	Length of the camera lens in millimeters used in perspective viewing (Read Only)
LIMCHECK	Limits checking for the current space, 1=on, 0=off
LIMMAX	Upper right corner of the Limits expressed in World Coordinate system
LIMMIN	Lower-left corner of the Limits expressed in World Coordinate system
LOGINNAME	Displays the Userâs name as configured (REad Only)
LTSCALE	Global Linetype scale factor
LUNITS	Drawing Units 1=scientific (do not use) 2=decimal (use for metric drawings) 3=engineering (do not use) 4=architectural (use for most all drawings -- feet and inches) 5=fractional (do not use)
LUPREC	Drawing unit number of decimal places or fractional denominator smallest value
MACROTRACE	Debugging tool for Diesel language expressions
MAXACTVP	Maximum number of viewports to regenerate at one time
MAXSORT	Maximum number of symbol or file names to be sorted by listing commands. Default value = 200. What this means is that, for instance, if there are more than 200 layers, the layer list will not get sorted alphabetically. To allow it to sort, change the MAXSORT variable to some number greater than the number of layers.
MENUCTL	Controls the page switching of the Screen menu. 0=Screen menu does not switch pages in response to keyboard command entry 1=Screen menu switches pages in response to keyboard command

	entry (default)
MENUECHO	Controls display of menu items: 0=all menu items and system prompts are displayed (default) 1=supresses echo of menu items 2=supresses printing of system prompts during menu 4=Disables ^P toggle of menu echoing 8=debugging aid for Diesel language macros
MENUNAME	The name of the currently loaded menu (Read Only)
MIRRTEXT	0=text retains its right reading direction when mirrored 1=text will read backwards when mirrored.
MODEMACRO	Allows you to display a text string in the status line, such as the name of the current drawing, time/date stamp, or special modes.
OFFSETDIST	Sets the default offset distance. If you enter a negative number, it defaults to the "Through" mode.
ORTHOMODE	0=Ortho Off 1=Ortho On
OSMODE	Sets the Running Mode of OSNAP 0=None 1=Endpoint 2=Midpoint 4=Center 8=Node 16=Quadrant 32=Intersection 64=Insertion 128=Perpendicular 256=Tangent 512=Nearest 1024=Quick Setting number values can be added to obtain combinations of Running OSNAP mode, for instance, number 33 could only mean, Endpoint (1) and Intersection (32) combination
PDMODE	Graphic display of points, for example: 0= normal point 1=no display of point 3=point that looks like an X
PDSIZE	Graphic size display of points
PERIMETER	Perimeter of Pline, circle, arc, etc., as computed by the AREA or LIST commands (Read Only)
PFACEMAX	Maximum number of vertices per face (Read Only)
PICKADD	Controls additive selection of entities 0= Disables PICKADD. The most recently selected entities becomes the object selection set. You cannot add more just by picking more. To add entities to the object selection set, hold the <Shift> key down while picking. This mimics the way most other Windows software works.

	1=Enables PICKADD. Each entity that is selected becomes part of the object selection set. To remove entities from the object selection set, hold the <Shift> key down while picking (default)
PICKAUTO	Controls automatic windowing during object selection. 0=Disables PICKAUTO 1=Allows you to create a selection window or crossing window automatically while selecting objects, if you did not pick directly on an object (default)
PICKBOX	Pickbox (object selection box) size in pixels
PICKDRAG	Controls method of creating a selection window 0=You draw the selection window by clicking the mouse at one corner and then the other corner (default) 1=You draw the selection window by clicking on one corner, holding down the mouse button, and releasing the mouse button at the other corner. This mimics the way most other Windows software works.
PICKFIRST	Controls the method of entity selection so that you can select objects first, and then use an edit/inquiry command 0=Disables Pickfirst 1=Enables Pickfirst (default)
PLATFORM	Indicates which version of AutoCAD is in use (Read Only)
PLINEGEN	Sets linetype pattern generation around the vertices of a 2D polyline. 0=linetype pattern starts and ends at each polyline vertex 1=linetype is generated in a continuous pattern through the polyline vertices. (does not apply to tapered segments)
PLINEWID	Default polyline width. Can be set to 0, which makes polylines plot at pen width.
PLOTID	Default plotter by plotter description set in Config
PLOTTER	Default plotter by plotter number set in Config
POLYSIDES	Default number of sides for Polygon command. Range is 3 to 1024. Default is 4.
POPUPS	0=display driver does not support dialogue boxes, menu bar, pull-down menus, and icon menus. 1=display driver supports dialogue boxes, menu bar, pull-down menus, and icon menus. (Read Only)
PSLTSCALE	Controls paper space linetype scale 0=No special linetype scale in paper space 1=Viewport scaling is multiplied times variable LTSCALE to compute linetype scales in paper space viewports (this setting is recommended)
PSPROLOG	Assigns a name for a Prologue section to be read from the ACAD.PSF file when using the PSOUT command.
PSQUALITY	Controls rendering quality of Postscript images.

QTEXTMODE	0=Quicktext mode off (line of text appears and plots as readable words) 1=Quicktext mode on (line of text appears and plots as rectangles)
REGENMODE	0=Regenauto off 1=Regenauto on
RE-INIT	Reinitializes the I/O ports, digitizer, display, plotter, and ACAD.PGP file. 1=digitizer port reinitialization 2=plotter port reinitialization 4=digitizer reinitialization 8=display reinitialization 16=ACAD.PGP file reinitialization
SAVEFILE	Current AutoSave filename (Read Only)
SAVENAME	Filename you save drawing to (Read Only)
SAVETIME	Automatic save interval, in minutes. If you do not want to save the drawing automatically, set SAVETIME to 0. Drawing is automatically saved to file AUTO.SV$. If your drawing file becomes corrupt for any reason, you can copy the file AUTO.SV$ to your filename, and you will have saved your file.
SCREENBOXES	The number of boxes in the screen menu area of the graphics area. If screen menu is disabled, SCREENBOXES=0 (Read Only)
SCREENMODE	Bit code indicating the state of the display 0=text screen is displayed 1=graphics mode is displayed 2=dual screen configuration (Read Only)
SCREENSIZE	Current viewport size in pixels, X and Y (Read Only)
SHADEDGE	0=faces shaded, edges not highlighted 1=facxes shaded, edges drawn in background color 2=faces not shaded, edges in entity color 3=faces in entity color, edges in background color
SHADEDIF	Ratio of diffuse reflective light to ambient light, in percent of diffuse reflective light
SHPNAME	Default Shape name
SKETCHINC	SKETCH command record increment
SKPOLY	0=SKETCH command generates individual lines 1=SKETCH command generates connected polylines
SNAPANG	Snap/Grid rotation angle for current viewport
SNAPBASE	Snap/grid origin point, in current UCS X and Y coordinates
SNAPISOPAIR	Current isometric plane on which you are drawing 0=left 1=top

	2=right
SNAPMODE	0=snap off 1=snap on
SNAPSTYL	0=rectangular snap grid 1=isometric snap grid
SNAPUNIT	Snap spacing, X and Y dimensions
SORTENTS	Controls display of entity sort order operations. 0=Disables SORTENTS 1=Sort for object selection 2=sort for object snap 4=sort for redraws 8=sort for MSLIDE slide creation 16=sort for regens 32=sort for plotting 64=sort for PostScript operation 96=sort for plotting and PostScript operation (32+64=96) (default)
SPLFRAME	0= invisible edges of 3DFACEs are not displayed, control polygon for spline fit polylines is not displayed, and the fit surface of polygon meshes are displayed, not the defining mesh armature. 1= invisible edges of 3dFACEs are displayed, control polygon for spline fit polylines is displayed, only the defining surface of a mesh is displayed.
SPLINESEGS	Number of line segments to be generated for each spline patch
SPLINETYPE	Type of spline curve to be drawn by PEDIT SPLINE command. 5=quadratic B-Spline 6=Cubic B-Spline
SURFTAB1	Number of tabulations to be drawn for RULESURF and TABSURF commands, and mesh density in the M direction for REVSURF and EDGESURF commands.
SURFTAB2	Mesh density in the N direction for the REVSURF and EDGESURF commands.
SURFTYPE	Type of surface fitting to be performed by PEDIT SMOOTH command. 5=quadratic B-spline surface 6=cubic B-spline surface 8=Bezier surface
SURFU	Surface density in the M direction
SURFV	Surface density in the N direction
SYSCODEPAGE	System code page specified in the ACAD.XMF file (Read Only)
TABMODE	Controls use of the Tablet Mode 0=Disables Tablet Mode 1=Enables Tablet Mode

TARGET	Location of the Target Point (where you are looking) (Read Only)
TDCREATE	Time and Date of drawing creation (Read Only)
TDINDWG	Total editing time (Read Only)
TDUPDATE	Time and Date of last save of drawing (Read Only)
TDUSRTIMER	User elapsed timer (Read Only)
TEMPPREFIX	Directory for placement of temporary files (*.AC$ files) (Read Only)
TEXTEVAL	0= all responses to prompts for text strings are taken literally 1= text starting with an opening parenthesis "(" or an exclamation point "!" is evaluated as an AutoLISP expression (default)
TEXTSIZE	Default height for new text entities, unless the current text style has been given a fixed text height.
TEXTSTYLE	Current text style
THICKNESS	current entity thickness (vertical distance from the bottom of an entity to its top)
TILEMODE	0= Disables paper space and viewport entities 1= Enables paper space and viewport entities
TRACEWID	Trace entity width (Note that Traces are not used very much any more -- they have been functionally replaced by Polylines)
TREEDEPTH	A 4-digit (maximum) code that specifies the number of times the tree-structured spatial index may divide into branches. This code number affects the speed in which AutoCAD searches the database before completing an action.
TREEMAX	Limits memory consumption during drawing regeneration by limiting the maximum number of nodes in the spatial index created with the variable TREEDEPTH.
UCSFOLLOW	0= change to a new UCS will not change the current view of the model 1= change to a new UCS will automatically change to a plan view (X-Y plane) in the new UCS
UCSICON	0= UCSICON is not visible 1= UCSICON is visible 2= UCSICON is located at the origin of the current UCS
UCSNAME	Name of the current UCS (Read Only)
UCSORG	Origin point of the current UCS, in World Coordinate System X,Y, and Z coordinates (Read Only)
UCSXDIR	X-direction of current UCS (Read Only)

UCSYDIR	Y-direction of current UCS (Read Only)
UNDOCTL	Indicates state of UNDO feature 1= UNDO is enabled 2= only one command can be undone 4= Auto-group mode is enabled 8= Undo group is currently active (Read Only)
UNDOMARKS	Number of marks that have been placed in the UNDO control stream by the UNDOâs command Mark option. (Read Only)
UNITMODE	0= displays fractional, feet and inches, and surveyorâs units as previously set. 1= displays fractional, feet and inches, and surveyorâs units in input format
USERI1-5	5 user-created variables for storage and retrieval of integer values
USERR1-5	5 user-created variables for storage and retrieval of Real Numbers
USERS1-5	5 user-created variables for storage and retrieval of text string data
VIEWCTR	Center of view in current viewport in X and Y coordinates (Read Only)
VIEWDIR	Current viewportâs viewing direction (Read Only)
VIEWMODE	1= perspective view active 2= front clipping on 4= back clipping on 8= UCS Follow mode on 16= Front clip not at eye (Read Only)
VIEWSIZE	Height of view in current viewport in feet and inches (Read Only)
VIEWTWIST	View twist (rotation) angle (Read Only)
VISRETAIN	0= current drawing layer visibility, color, and linetype settings do not take precedence over the XREF drawingsâ layer settings 1= current drawing layer visibility, color, and linetype settings overrides the XREF drawingsâ layer settings
VSMAX	The upper-right corner of the current viewportâs virtual screen (Read Only)
VSMIN	The lower-left corner of the current viewportâs virtual screen (Read Only)
WORLDUCS	0= the current UCS is not the World Coordinate System 1= the current UCS is the World Coordinate System (Read Only)
WORLDVIEW	0= when DVIEW or VPOINT commands are used, the coordinate system is not changed, and DVIEW

	and VPOINT command input is relative to the current UCS1= when DVIEW or VPOINT commands are used, the coordinate system is automatically changed to the WCS for the duration of the command (default)
XREFCTL	0= XREF log files are not created 1= XREF log files are created an XREF log file is a text file with the extension *.XLG which lists all XREFs used in the current drawing along with their paths.