

Appendix B

AutoCAD Commands

3D Draws a 3D polygon entities with surfaces. Options:

B - Draws a 3D box, C - Draws a wire frame having a cone shape, DI - Draws a 3D dish shaped (lower half of a sphere) polygon mesh by specifying the center and then the diameter or radius, DO - Draws the 3D upper half of the spherical polygon mesh by specifying the center and then the diameter or radius, M - Draws a polygon mesh by specifying the corners and the M and N sizes, P - Draws a 3D tetrahedron or a pyramid by specifying the relevant number of base points and the apex point or the top points, S - Draws a spherical polygon mesh by specifying the center and then the diameter or radius, T - Draws a polygon mesh having a toroidal shape. It is drawn by specifying the center and then the diameter or the radius, W - Draws a polygon wire frame having the shape of a wedge. It is drawn by specifying a corner, height, length, and width.

3DARRAY Draws a 3D rectangular or polar array. Options:

R - Rectangular 3D array, P - Polar array .

3DCLIP Starts the interactive 3D viewing and opens the **Adjust Clipping Planes** window.

3DCORBIT Starts the interactive 3D viewing and enables you to set the drawing objects in the 3D view into continuous motion.

3DDISTANCE Simulates the effect of moving the camera closer or farther to the drawing objects.

3DFACE Draws a 3D surface with three or four sides. Options: First point, I- Invisible.

3DMESH Draws a polygon mesh by specifying the size of M and N and the location of the vertices.

3DORBIT Controls the interactive viewing of 3D objects in the drawing area.

3DPAN Allows you to drag the view horizontally, vertically or diagonally.

3DPOLY Draws a 3D polyline having segments of a straight line. Options:

Point - Draws the 3D polyline to the specified point, C - Closes the 3D polyline by joining the last point with the first point, U - Deletes the last segment.

3DSIN Displays a dialog box that allows you to import the specified objects in a 3D Studio file.

3DSOUT Displays a dialog box that exports the AutoCAD objects with surface characteristics to a 3D studio file.

3DSWIVEL Simulates the effects of turning the camera on a tripod.

3DZOOM Allows you to zoom in and out on the view.

ABOUT Displays AutoCAD version and serial numbers and other information.

ACISIN Imports an ASCII ACIS file into the AutoCAD drawing.

ACISOUT AutoCAD exports the selected 3D solid objects to an ASCII ACIS file.

ADCCLOSE Closes the AutoCAD Design Center.

ADCCENTER Starts and displays AutoCAD Design Center.

ADCNAVIGATE Directs the Desktop in AutoCAD DesignCenter.

ALIGN Allows specified objects to align with other 2D or 3D objects by moving and rotating them.

AMECONVERT Changes the regions and solids of AME to region and solids of AutoCAD.

APERTURE Controls the size of the object snap target box.

APPLOAD Displays a dialog box that loads certain applications such as AutoLISP, ADS, and ARX at startup.

ARC Draws an arc of any size. The default method is to specify two endpoints and a point along the arc. Options:

A - Included angle, C - Center point, D - Starting direction, E - Endpoint, L - Length of chord, R - Radius.

AREA Computes the area and perimeter of different objects and of a region formed by specifying a sequence of points. Options:

A - Add mode, F - First point (Area by specifying points), O - Area of the object, S - Subtract mode.

ARRAY Creates specified number of copies of a selected object using the **Array** dialog box. Dialog box tabs:

Rectangular array, Polar Array.

ARX Loads, unloads, and provides information about ARX applications.

ASSIST This command is used to invoke the active assistance. This is a dynamic help mode that will automatically display the help related to the active command or dialog box when you move over it.

ATTDEF Creates an attribute definition (characteristics of an attribute). Options:

I - Invisible mode: Attribute remains invisible, C - Constant mode: Constant value of attribute, V - Verify mode: Verifies attribute value is correct, P - Preset mode: Default value to attribute.

ATTDISP Controls the visibility of the attribute globally. Options:

ON - Attributes made visible, OFF - Attributes made invisible, N - Current visibility kept.

ATTEDIT Edits attributes within a block or blocks. Options:

Y - Yes (one attribute at a time), N - No (all attributes at a time).

ATTEXT Attribute information is extracted from the drawing. Options:
C - CDF: Comma-Delimited File, D - DXF: Drawing Interchange File, S - SDF: Space-Delimited File, O - Drawing objects.

ATTREDEF An existing block is redefined and its attributes are updated.

ATTSYNC This command updates all the block insertions with the current values of the attributes.

AUDIT Identifies errors in a drawing. Options:
Y - Corrects the errors, N - Informs about the error without correcting it

BACKGROUND Creates a background for your scene.

BASE Sets the point of origin for inserting a drawing into another drawing.

BATTMAN This command displays the **Block Attribute Manager** used to edit and manage the block attributes in the current drawing.

BHATCH A specified enclosed area is filled with an associative hatch pattern through a dialog box. Previewing a hatch and adjusting the boundary is also possible. Options:
I - Internal point, P - Properties, S - Select, R - Remove islands, A - Advanced.

'BLIPMODE Controls the appearance of marker blip that is displayed on the screen when a point is picked. Options:
ON - Marker blip displayed, OFF - Marker blip not displayed.

BLOCK Creates a compound object as a block definition from a set of entities. Options:
? - Lists names of previously defined blocks.

BLOCKICON Generates preview images for the blocks created in Release 14 or earlier.

BMPOUT Creates a bitmap image of the drawing and saves the screen to a file having a .bmp extension.

BOUNDARY Creates a polyline or region of a boundary that defines an enclosed area. Options:
A - Advanced Options.

BOX Creates a solid box that is 3D in nature with its base parallel to the XY plane. Options:
C - Corner of box, CE - Center of box.

BREAK Removes specified portions of an object or splits the object. Options:
F - Respecifies first point.

BROWSER Launches the default Web browser defined in the registry of the system.

'CAL Calculates expressions that can be mathematical as well as geometrical.

CAMERA Sets a different camera location and target locations.

CHAMFER Connects two nonparallel objects with a beveled line. Options:
A - Chamfer distance is set using angle and distance, D - Sets chamfer distance, P - Chamfers entire polyline, T - Controls the trimming of the edges to chamfer line endpoints, M - Trim method.

CHANGE Alters the properties of selected objects. Options:
C - Change point - Changes lines, circles, Text, Attribute Definitions, Blocks, P - Changes properties like Color, Elev, LAYER, LType, Thickness.

CHECKSTANDARDS This command is used to check for any violations of the standard by the standards configured in the current drawing. The **Check Standards** dialog box is dis-

played when you invoke this command.

CHPROP Alters the drawing properties of selected objects. Options:

C - Changes color, LA - Changes layer, LT - Changes linetype, S - Changes linetype scale factor, LW - Changes lineweight, T - Changes thickness, PL - Changes the plotstyle of the selected objects.

CIRCLE Draws a circle using any of the four methods available. Options:

C - Circle drawn on the basis of center point and diameter or radius, 3P - Drawn on the basis of 3 points on circumference, 2P - Drawn on the basis of 2 endpoints of the diameter, TTR - Circle drawn tangent to two objects with a specified radius.

CLOSE Closes the current drawing if no change has occurred since last save or will prompt you to save the drawing.

CLOSEALL This command is used to close all the drawings that are open. If the files have some unsaved data then u will be prompted to specify whether or not you want to save the changes in the drawings.

COLOR Sets color for subsequent objects drawn. Options:

value - Sets color by number (1-255), name - Sets color by name, Byblock - The current color setting is inherited by the block at the time of insertion, Bylayer - Objects inherit the color of the layer in which they are drawn.

COMPILE Shape files and PostScript font files are compiled.

CONE Draws a 3D solid cone. Options:

Center - Cone having circular base, E - Cone having elliptical base.

CONVERT This command is used to convert a polyline or a hatch pattern created in a

AutoCAD Release 13 or earlier release into the AutoCAD 2004 polyline or hatch pattern.

CONVERTCTB This command is used to convert a color-dependent plot style table into a named plot style table.

CONVERTPSTYLES This command is used to convert the current drawing into a colordependent plot style or into a named plot style.

COPY Draws a copy of the selected object leaving the original object intact. The default method is to specify the base point. Options: M - Multiple copies of object in single **COPY** command.

COPYBASE Copies the drawing objects with a specified base point.

COPYCLIP Copies the selected objects to the Clipboard.

COPYHIST Copies the text in the command line history to the Clipboard.

COPYLINK Copies the current view to the Clipboard to link with other OLE applications.

CUTCLIP Copies objects to the Clipboard but erases them from the drawing.

CYLINDER Draws a 3D solid cylinder. Options:

Center - Specifies the center of the circular base, E - Forms an elliptical base.

DBC_CLOSE Closes the **dbConnect Manager**.

DBLCLKEDIT This command is used to set the double-clicking option. When you doubleclick on the entities then either the properties window is displayed or the related

editor or dialog box is displayed. This is because of the reason that this value is set to ON. If set to **OFF** then there is no response upon double-clicking on the objects.

DBCONNECT Provides an AutoCAD interface to external database tables by starting **dbConnect Manager**.

DBLIST Lists all the database information for each object in the drawing.

DDEDIT Displays a dialog box that allows the user to edit text and attribute definitions.

DDPTYPE Displays a dialog box that sets the point style and also the size of the point object.

DDVPOINT Controls the direction of 3D views through a dialog box.

DELAY The execution of the next command is postponed for a specified time duration. In other words a specified pause is provided within the script.

DIM and **DIM1** Dimensioning mode is invoked and permits the use of dimension subcommands from AutoCAD's previous releases.

DIMALIGNED Creates a linear dimension aligned to the specified points or object. **DIMANGULAR** Creates an angular dimension.

DIMBASELINE Starts drawing from the baseline of the previous dimension. The new dimension can be linear, angular, or ordinate. **DIMCENTER** Draws center mark or center lines in circles and arcs.

DIMCONTINUE Starts drawing a new dimension from the second extension line of the previous or selected dimension. The new di-

mension can be a linear, angular, or ordinate dimension.

DIMDIAMETER Draws diameter dimensions for different circles and arcs.

DIMDISASSOCIATE This command is used to disassociate the true associative dimensions from the objects to which they are associated.

DIMEDIT Edits dimension text and extension lines. Options:

H - Dimension text moved back to default position, N - Dimension text is replaced, R - Dimension text is rotated, O - Extension lines placed at obliquing angle.

DIMLINEAR Draws linear dimensions.

DIMORDINATE Creates ordinate dimensions.

DIMOVERRIDE The settings of the dimensioning system variables concerning the dimension object are overridden. The current dimension style is not affected.

DIMRADIUS Draws radial dimensions for different circles and arcs.

DIMREASSOCIATE This command is used to convert a dimension into a true associative dimension by associating them to the selected objects.

DIMREGEN This command is used to update the true associative dimensions.

DIMSTYLE New dimension styles are created and the existing ones are modified. Options: R - Dimensioning system variable setting changed, S - Current settings of dimensioning variables saved, ST - Current values of dimensioning variables displayed, V - Dimensioning variable setting of a style is listed, A - Selected dimension objects are updated, ? - Named dimension styles are listed.

DIMTEDIT Dimension text is moved and rotated. Options:

A - Angle of dimension text changed, H - Dimension text moved to default position, C - Dimension text center justified, L - Dimension text left justified, R - Dimension text right justified.

DIST Distance and angle between two points is measured.

DIVIDE Places blocks or points as markers at equal distance along the length or perimeter of an entity, thus dividing it into a specified number of equal parts. Options:

B - Places blocks as markers.

DONUT Draws wide polyline with specified inside and outside diameters, thus forming a ring.

DRAGMODE Controls the dragging feature for appropriate commands. Options:

ON - Permits dragging, OFF - Ignores dragging, A - Permits dragging wherever possible.

DRAWORDER Changes the display order of images and other objects.

DSETTINGS Specifies the settings for Snap mode, grid, polar, and object snap tracking.

DSVIEWER Displays the Aerial View window.

DVIEW Parallel projection or perspective views are defined. Options:

CA - Sets camera position by rotating about the target, TA - Sets target position by rotating about the camera, D - Camera to target distance is set, PO - Locates target and camera points, PA - Pans image, Z - Zooms In/Out, TW - Tilts view around line of sight, CL - The view is clipped in front and back, H - Hidden lines removed on selected objects, OFF - Perspective viewing turned off, U - Last DVIEW operation reversed.

DWGPROPS Sets and displays all the properties of the current drawing.

DXBIN Specially coded binary files are imported into a drawing.

EATTEDIT This command is used to display the **Enhanced Attribute Editor** for editing the block attributes. You can change the block attributes, values, position, justification and so on using this editor.

EATTEXT This command is used to extract the block attributes and copy them into a file. This command has an advantage that you do not need a template file for extracting the attributes if you are using this command. When you invoke this command, the **Attribute Extraction Wizard** is displayed that guides you through the step by step procedure of extracting the attributes.

EDGE The visibility of 3D sides is altered. By default the selected edge is hidden. Options: D - Display mode invisible edges are highlighted.

EDGESURF A 3D polygon mesh is created with four adjoining edges that define a Coons surface patch.

ELEV The elevation and extrusion thickness is set for the new objects.

ELLIPSE Draws ellipses or elliptical arcs using different options. Specifying the axis endpoint is the default method. Options:

A - Draws elliptical arc, C - Specifies the center point of the ellipse, I - Draws isometric circle in current isometric plane.

ETRANSMIT This command is used to electronically transmit a drawing file. The resultant file can be converted into an EXE file, a zip file or can be copied into a folder.

ERASE Erases the selected objects from the drawing.

EXPLODE Compound objects (blocks, dimensions, polylines, 3D solids, regions, polygon meshes, multilines) are broken into their constituent parts.

EXPORT Objects are saved to other file formats via a dialog box.

EXTEND Lengthens a selected entity to meet another entity. Options:

P - Specifies projection mode like UCS and View, E - Controls the extension to implied or actual edge, U - Latest extension is undone.

EXTRUDE Solids are created by extruding 2D entities along a selected path. By default, height of extrusion is to be specified which extrudes the object along the positive Z axis. Options:

P - Extrusion path is selected.

FILL Controls whether multilines, traces, solids, or wide polylines are filled or not filled. Options:

ON - Fill mode is enabled, OFF - Fill mode is disabled.

FILLET The edges of two specified lines, arcs, or circles are filleted by construction of an arc of specified radius. The default method is to specify the two objects. Options:

P - Entire polyline is filleted, R - The radius of the fillet arc is specified, T - Controls the trimming of the edges to fillet arc endpoints.

FILTER Creates a list of properties on the basis of which the objects are selected.

FIND Finds, replaces, selects, or zooms to specified text through a dialog box.

FOG Provides visual clues for the apparent distance of objects.

GRAPHSCR Flips to the graphics window from the text window.

GRID A grid of dots at specified spacing is displayed. Options:

Grid spacing(X) - Grid set to specified value, ON - Grid turned on at current spacing, OFF - Grid turned off, S - Grid spacing set to current Snap interval, A - Grid set to different spacing in X and Y.

GROUP Creates and changes object selection groups, which are sets of objects having specific names. Options:

? - Lists names and descriptions of all the group, O - Changes the numerical order of objects in the group, A - Adds objects to the group, R - Remove objects from the group, E - Explodes the group into its component objects, REN - Assigns a new name to an existing group, S - Specifies whether a group is selectable, C - Creates a new group.

HATCH A specified area is filled with a selected pattern. Options:

? - Lists the hatch patterns in acad.pat file, name - A pattern name as defined in acad.pat file is specified, U - User-defined hatch pattern is specified. U can be followed by a comma and a hatch style, S - Specifies a solid fill.

HATCHEDIT Edits a hatch block through a dialog box. It sets the pattern type and properties and then applies it to a block. Options: D - Removes the associative quality from an associative hatch, S - Changes the hatch style type, P - Specifies new hatch properties.

HELP(F1) Displays help for a specific command and also lists the commands and data entry options.

HIDE Regenerating of a 3D object is performed with the removal of hidden lines.

HYPERLINK Attaches a hyperlink to a graphical object or modifies an existing hyperlink through a dialog box.

HYPERLINKOPTIONS Controls the visibility of the hyperlink cursor and the display of hyperlink tooltips. Options:
Y - Shows the hyperlink cursor and tooltip,
N - Does not display.
ID The UCS coordinates of a specified point are displayed.

IMAGE Inserts images into an AutoCAD drawing file.

IMAGEADJUST Controls the brightness, contrast, and fade values of the selected image.

IMAGEATTACH Attaches a new image object and definition.

IMAGECLIP Creates new clipping boundaries for single image objects.

IMAGEFRAME Controls the display of image frame on the screen. Options:
On - Displays image frames, Off - Hides image frames.

IMAGEQUALITY Controls the display quality of images. Options:
H - Produces high quality display, D - Produces low quality display.

IMPORT Imports the different file formats into an AutoCAD drawing via a dialog box.

INSERT Places a previously defined named block or drawing into the current drawing.

INSERTOBJ Inserts a previously linked or embedded object.

INTERFERE Highlights all of the interfering solids and then creates new solids from the intersections of the interfering pairs of solids.

INTERSECT A new composite solid is created from the intersecting region of two or more solids.

ISOPLANE An isometric plane is selected to be the current plane for an orthogonal drawing. Options:
L - Left-hand plane, T - Top plane, R - Right-hand plane.

JUSTIFYTEXT This command is used to change the justification point of the existing text. However, this command does not change the location of the text.

LAYER Creates layers and sets different properties for the specified layers. Options:
? - Lists defined layers, M - Creates a layer and makes it the current layer, S - Makes a specified already existing layer current, N - Creates one or more new layers, ON - Turns on the specified layers, OFF - Turns off the specified layers, C - Sets the color of the specified layer, L - Sets the linetype of the specified layer, LW - Changes the lineweight, P - Controls whether visible layers are plotted, PS - Sets the plot style assigned to a layer, F - Makes a layer invisible by freezing it, T - The frozen layer is thawed, LO - Locks layers, thus prevents editing on them, U - Unlocks specified locked layers.

LAYERP This command is similar to the undo command but is valid for only the layers. This command undoes the changes made using the last **LAYERS** command. However, this command has some limitations. This command can not undo the renaming of layers, the deletion of layers or the creation of new layers.

LAYERPMODE This command is used to control whether or not the changes made to the layers are tracked or not. If set to **ON**, the changes made to the layers are tracked otherwise they are not tracked.

LAYOUT Creates a new layout and renames, copies, saves, or deletes an existing layout. Options:

C - Copies a layout, D - Deletes a layout, N - Creates a new layout tab, T - Creates a new template, R - Renames a layout, SA - Saves a layout, S - Makes a layout current, ? - Lists all the available layouts.

LAYOUTWIZARD Starts the Layout wizard to designate page and plot settings for a new layout.

LAYTRANS This command is used to translate the selected layer into a layer of the specified standard.

LEADER Creates a line segment with an arrowhead that connects the text to a feature. The leader is created from a specified point to another point depending upon the options. Options:

A - Annotation is inserted at the end of leader line, F - Controls the type of leader (Spline, Straight, Arrow), U - The last vertex point is removed.

LENGTHEN Alters the length of specified entities and the included angle of arcs. Options: DE - Lengthens the object by a specified incremental distance, P - Alters the length by a specified percentage of its total length, T - Alters the length by specified total absolute length, DY - The object is lengthened to where its endpoint is dragged.

LIGHT Controls the lighting effects in the model space via a dialog box. It creates, modifies, deletes the lights and controls the color system in a drawing. It manages different

lights (Point light, Distant light, Spotlight) through a series of dialog boxes.

LIMITS Sets the drawing boundaries (and WCS grid extents) for the current space. Options:

Lower - Specifies 2 points-lower left corner and the left-upper right corner, ON - Limits checking is enabled, OFF - Limits checking is disabled.

LINE Draws straight line segments of any length by specifying the endpoints. Options: ENTER - Continues from end of previous line or arc, U - Removes the most recent segment, C - Closes polygon.

LINETYPE Defines line characteristics, loads linetypes and sets them for new entities. It also creates new linetype definitions to a library file. Options:

? - Lists linetypes in a file, C - Creates new linetype definition, L - Loads an already existing linetype definition, Sets linetype for new entities, S - Set the current linetype.

LIST Lists database information (type, layer, X,Y,Z position, thickness, and so on) about the specified entity.

LOAD Loads the shapes from the shape file to be used by the **SHAPE** command.

LOGFILEOFF The log file already opened is closed by this command.

LOGFILEON The subsequent contents of the text window are recorded into the log file.

LSEdit Lets you edit a landscape object. LSLIB Lets you maintain libraries of landscape objects.

LSNEW Lets you add realistic landscape items, such as trees and bushes, to your drawing.

LTSCALE Sets the global scale factor of the linetype so as to alter the relative length of dashes and dots.

LWEIGHT Sets the current lineweight, lineweight display options, and lineweight units.

MASSPROP Calculates and lists the mass characteristics of 2D and 3D objects. The properties displayed are Area, Perimeter, Bounding box, Centroid. For Coplanar regions, additional properties displayed are Moments of Inertia, Products of Inertia, Radii of Gyration, and Principal Moments. The properties displayed for solids are Mass, Volume, and the properties of the Coplanar region.

MATCHPROP Copies the properties from one object to one or more objects. **MATLIB** Displays a dialog box that lists all the predefined materials (material list) and lists the materials in the selected library (library list). It also imports and exports materials between those two lists.

MEASURE Places blocks or points as markers at measured intervals along the length or perimeter of an entity. Options:
B - Places blocks as markers.

MENU Loads a customized menu file into the menu area. The menu file contains the command strings and menu syntax.

MENULOAD Displays a dialog box that loads and permits you to add partial menu files to an already present base menu file.

MENUUNLOAD Displays the same dialog box as in the case of the **MENULOAD** command that can also be used to unload the partial menu files.

MINSERT Places multiple copies of a previ-

ously drawn named block or drawing into the current drawing in a rectangular array. Options:

? - Lists the defined block definitions, ~ - Displays a dialog box.

MIRROR Reflects objects so as to create their mirror images about a specified line.

MIRROR3D Reflects objects so as to create their mirror images about a specified plane. Options:

3points - 3points specify the mirroring plane, O - The plane of a planar object specifies the mirroring plane, L - The previous mirroring plane is taken as the present one, Z - The point on the plane and another point on the Z axis (normal) of the plane specifies the mirroring plane, V - A point on the viewing plane specifies the mirroring plane, XY/YZ/ZX - The mirroring plane is aligned to any one of the standard planes.

MLEDIT Displays a dialog box that controls intersection between multiple parallel lines and edits them. Different types of cross, tee, corner joints, and vertices can be created between multilines via the dialog box. It is also possible to cut and weld segments of a single multiline.

MLINE Draws multiple parallel lines between two points. Options:

J - Justification- How multiline is drawn between two points, S - Scale- Sets the width of the multiline, ST - Sets the multiline style.

MLSTYLE Displays a dialog box that creates a multiline style, makes a specific style current, saves, adds a style to the current list, renames a style, adds a description to a style, and loads a style from the library file. It also controls the element properties (number, offset, color, linetype) and the multiline properties (start and end caps, angle, background color).

MODEL Switches from a layout tab to the Model tab and makes it current.

MOVE Moves objects from one location to another by specifying a displacement.

MSLIDE Creates a slide file from the current display.

MSPACE Switches to model space in a floating viewport from paper space.

MTEXT Creates paragraph text within a specified text boundary. Displays a dialog box where you can specify the different options for the multiline text.

MULTIPLE Causes the repetition of the next command until it is cancelled.

MVIEW Creates viewports and controls the number and layout of paper space viewports. You specify diagonal corners of new viewport as the default option. Options:

ON - Viewport is turned on, OFF - Viewport is turned off, H - Hideplot- Hidden lines removed during plotting, F - Fit- Single viewport created that fills the display area completely, L - Locks the selected viewport, O - Specifies a closed polygon to convert it into viewport, P - Creates an irregularly shaped viewport using specified points, 2 - The specified area is divided into two viewports either horizontally or vertically, 3 - The specified area is divided into 3 viewports, 4 - The specified area is divided into 4 viewports, R - Restore - Viewport configurations changed into individual viewports.

MVSETUP The specifications of a drawing are set. Depending upon the system variable TILEMODE, the working of MVSETUP is different. When TILEMODE is On, drawing scale factor, units type, and paper size is set and lastly a bounding box is drawn. When TILEMODE is Off a set of floating viewports

is created. Options (TILEMODE Off):

A - Aligns the view in a viewport with another viewport. The view can be panned in a specified direction, aligned horizontally, vertically, or rotated, C - The viewport can be created, S - Sets the scale factor of objects in the viewport, O - Options - The layer can be set, reset limits, set units, Xref attach, T - Creates a title block and drawing border, U - Reverses the previous operation

NEW Displays a dialog box that creates a new drawing.

OFFSET Creates offset curves, concentric circles, and parallel lines at a specified distance from the original object. Options: value - Specify the offset distance, T - Through- The offset object passes through the specified point.

OLELINKS Updates, changes, and cancels existing OLE links.

OLESCALE Displays the OLE Properties through a dialog box.

OOPS Restores those entities that have been erased by the last **ERASE** command.

OPEN Displays a dialog box through which an existing drawing can be opened. The dialog box also displays the directory, files, preview, name of the file, and the pattern.

OPTIONS Customizes the AutoCAD settings through a dialog box.

ORTHO The movement of the cursor is restrained to only vertical or horizontal directions and aligned with the grid. Options: ON - Constrains cursor movement, OFF - Does not constrain cursor movement.

OSNAP Specifies a point at an exact location on an object by setting the Object Snap

modes. Options:

END - Closest endpoint of arc (arcs and lines include polyline segments), elliptical arc, ray, mline, line and closest corner of trace, solid, 3D face, **MID** - Midpoint of arc, elliptical arc, spline, ellipse, ray, solid, xline, mline, or line, **INT** - Intersection of line, arc, spline, elliptical arc, ellipse, ray, xline, mline, or circle, **EXT** - Snaps to the extension point of an object, **APP** - Apparent or extended (projected) intersection (which may not actually intersect in 3D space) of line, arc, spline, elliptical arc, ellipse, ray, xline, mline, or circle, **CEN** - Center of arc, elliptical arc, ellipse, or circle, **QUA** - Quadrant point of arc, elliptical arc, ellipse, solid, or circle, **PER** - Point perpendicular to arc, elliptical arc, ellipse, spline, ray, xline, mline, line, solid, or arc, **TAN** - Tangent to arc, elliptical arc, ellipse, or circle, **NOD** - Point object, **INS** - Insertion point of text, block, shape, or attribute, **NEA** - Nearest point of arc, elliptical arc, ellipse, spline, ray, xline, mline, line, circle, or point, **QUI** - First snap point, **NON** - Turns Object Snap mode off.

PAGESETUP Specifies the layout page, paper size, plotting device, and settings for each new layout through a dialog box.

PAN Moves the drawing display by a specified displacement.

PARTIALLOAD Loads additional geometry into a partially opened drawing.

PARTIALOPEN Loads additional geometry from a selected view or layer into a drawing. **PASTEBLOCK** Pastes a copied block in a new drawing.

PASTECLIP Inserts data from the Clipboard.

PASTEORIG Pastes a copied object in the new drawing using the geometric coordinates of the original drawing.

PASTESPEC Inserts data from the Clipboard and controls the format of the data.

PCINWIZARD Starts a wizard to import PCP and PC2 configuration file plot settings in current layout or model tab.

PEDIT Editing of 2D polyline, 3D polyline, or 3D mesh. Options:

2D polyline **C** - Closes polyline segment, **O** - Closing segment removed, **J** - Joins to polyline, **W** - Specifies uniform width, **E** - Edits the vertices. The first vertex is marked by placing an X. Editing includes moving the X to next or previous vertex, adding a new vertex, setting the first vertex for break, moving the vertex, regenerating, straightening and attaching a tangent direction to the current vertex, **F** - Fits arc curves smoothly to the polyline, replacing each line segment with a pair of arcs, **S** - Vertices are used as a frame for spline curve, **L** - Linetype generation in a continuous pattern, **U** - Reverses the previous operation, 3D polyline **C** - Closes polyline segment, **O** - Closing segment removed, **E** - Edits the vertices. Same suboptions as in 2D Edit except the tangent suboption, **S** - Vertices are used as a frame for spline curve, **D** - Removes a spline curve to its control frame, **U** - reverses the previous option, **X** - Exits **PEDIT**. 3D polygon mesh **E** - Edits vertices. The first vertex is marked by placing an X. Editing includes moving the X to next or previous vertex, moving the X marker to the next vertex or the previous vertex in the N direction, moving the marker to the next or previous vertex in the M direction, regenerating the mesh, **S** - Fits a smooth surface, **D** - The control point polygon mesh is restored, **Mc**close - M-direction polylines are closed, **M**open - M-direction polylines are opened, **Nc**close - N-direction polylines are closed, **N**open - N-direction polylines are opened, **U** - Reverses editing operations as far back as the beginning of the **PEDIT** session.

PFACE A 3D polyface mesh is created vertex by vertex.

PLAN Allows you to view the drawing from plan view of a User Coordinate System. Options:

C - Plan view of the current UCS, U - Plan view of the specified UCS, W - Plan view of the World Coordinate System.

PLINE Draws 2D polylines. The default is to draw a polyline between two specified points. Options:

A - Arc mode- Arc segments can be added to polyline. The arc segment starts from the endpoint of the previous polyline segment and can be drawn by specifying the endpoint of the arc, the included angle, center of the arc, starting direction of the arc, halfwidth of the arc, radius of the arc, Width. You can also close the polyline with the arc segment, or reverse the previous operation or you can shift to the Line mode, C - Closes the polyline, H - Sets the halfwidth, L - Draws polyline of specified length, U - Last polyline segment is removed, W - The width of the next segment is specified.

PLOT Displays a dialog box that allows you to plot the drawing to the plotting device or file. Through a series of dialog boxes you can set the different parameters, device information, drawing extents and limits, plot size, paper size, orientation, plot scale, rotation, and origin. You can also plot a view or a specific portion of the drawing and also preview the plot.

PLOTSTAMP This command is used to add additional text to the printed drawings. This additional text is called stamp and is not displayed in the plot preview.

PLOTSTYLE Sets the current plot style for new drawing objects, or the assigned plot style for selected objects through a dialog box.

PLOTTERMANAGER Opens the **Plotter Manager** and enables you to launch Add-a-plotter wizard and Plotter Configuration Editor.

POINT Draws a point object at a specified location.

POLYGON Draws a polygon (closed polyline object) having specified number of sides. Options:

C - Specifies the center of polygon. Suboptions:

I - Inscribed in the circle, C - Circumscribed about the circle. E - Defines one edge of the polygon.

PREVIEW Shows how the drawing will look when it is printed or plotted.

PROPERTIES Controls properties of a drawing object and displays Properties window.

PROPERTIESCLOSE Closes the Properties window.

PSETUPIN Imports a user-defined page setup into a new drawing layout.

PSPACE Switches from a model space viewport to paper space.

PUBLISH Creates multi-sheet drawing sets for publishing to a single multi-sheet DWF (Design Web Format) file, a plotting device, or a plot file.

PUBLISHTOWEB This command allows you to create a web page that will comprise of the text and specified drawings. You can specify the layouts of the drawings to be included in the web page. When you invoke this command, the **Publish to Web** wizard start that will guide you through the process of creating the web page.

PURGE Removes those references from the database that are not being used. Options:

B - Removes unused blocks, D - Removes unused dimstyles, LA - Removes unused layers, LT - Removes unused linetypes, P - Removes unused plotstyle, SH - Removes unused shape files, ST - Removes unused text styles, M - Removes unused mline styles, A - Removes all unused objects.

QDIM Quickly creates a dimension on the drawing objects.

QLEADER Quickly creates a leader and its annotation.

QNEW Opens a new AutoCAD file.

QSAVE Saves and backs up the drawing without asking for a filename.

QSELECT Quickly creates selection sets based on filtering criteria.

QTEXT Sets the text and the attribute objects to be displayed without drawing the text detail. Options:

ON - Text displayed as a bounding box. OFF - Quick text mode off.

QUIT Exits AutoCAD without saving.

RAY Draws a semi-infinite line used as a construction line.

RECOVER Recovers a damaged and corrupted drawing.

RECTANG Creates a polyline rectangle by specifying the diagonally opposite corners.

REDEFINE Restores an AutoCAD built-in command that has been previously overridden by **UNDEFINE**.

REDO The effect of the previous command if

it was **UNDO** is reversed.

REDRAW Cleans up the current viewport by removing the blip marks and other stray pixels and redrawing missing portions of objects.

REDRAWALL Refreshes or cleans up all the viewports.

REFCLOSE Saves back or discards changes made during in-place editing of a reference.

REFEDIT Selects an external reference for editing.

REFSET Adds or removes objects from a working set during in-place editing of a reference.

REGEN Regenerates the current viewport.

REGENALL Regenerates all the viewports.

REGENAUTO Controls automatic regeneration of the drawing. Options:

ON - Permits automatic regeneration. OFF - Does not permit automatic regeneration.

REGION Region entities (2D enclosed areas) are created from a selection set.

REINIT Reinitializes the I/O ports, digitizer, display, or parameters file through a dialog box.

RENAME Alters the name of entities. Options: B - Renames block, D - Renames dimstyle, LA - Renames layers, LT - Renames linetype, S - Renames style, U - Renames UCS, VI - Renames view, VP - Renames viewport configuration.

RENDER Displays a dialog box that shades a 3D wireframe or solid, so that a realistically shaded image is created. It is possible to render the current scene or just the specified

objects. You can also control the color map and the shading of different materials.

REPLAY The BMP, TGA, or TIFF images are displayed via a dialog box.

RESUME Resumes an interrupted script.

REVCLLOUD Creates a revision cloud. Options: Object - Converts a closed loop into a revision cloud. Arc length - Lets you specify the minimum and maximum arc length of the arcs in the revision cloud.

REVOLVE By revolving a 2D entity (polygon, closed polyline, circle, ellipse, donuts, and so on), a solid is formed. Options:

point - The axis of revolution is specified by two points, O - The axis of revolution is specified by selecting an existing line or a segment polyline, X - The positive X axis used as the axis direction, Y - The positive Y axis used as the axis direction.

REVSURF A polygon mesh is constructed by rotating a curve or profile around a specified axis.

RMAT Displays a dialog box that manages the materials used for rendering. A new material can be created or the existing ones can be modified through a series of dialog boxes. It is possible to adjust the value and color of the materials. AutoCAD's color index can also be attached by layers or by using a color wheel.

RMLIN This command is used to copy redline markup language in the current drawing for viewing the electronic comments on the design. The RML insertions are placed on a separate layer called **MARKUP**.

ROTATE Rotates specified entities about a base point. Options:
angle - Rotates object through a specified

angle, R - Rotates object with respect to the reference angle.

ROTATE3D Rotates object about a 3D axis. Options:

2points - The axis of rotation is given by specifying 2 points, A - Axis by object- The axis of rotation is aligned with an object, L - The previous rotation axis is considered, V - The axis of rotation is aligned with the viewing direction, X/Y/Z - The axis of rotation is aligned with any one of the axes (X-axis, Y-axis, Z-axis).

RPREF Displays a dialog box that controls the rendering preferences. It controls the color map, the behavior of the **RENDER** command by default, rendering display, and the image output setting. Through a series of subdialog boxes, the type of shading used and 3D solid faces can be controlled. You can also set the color and the aspect ratio of the output file.

RSCRIPT Repeats a script continuously.
RULESURF Creates a polygon mesh representing a ruled surface between two curves.

SAVE A name is requested under which the drawing is saved. If the drawing is already named, then it is saved under the current filename.

SAVEAS An unnamed drawing is saved with a filename or the current drawing is renamed.

SAVEIMG Displays a dialog box that saves a rendered image to a file. Through the subdialog boxes, image compression for TGA and TIFF formats is possible.

SCALE The size of the existing objects is changed. The default is to specify a scale factor. Options:
R - The object is scaled according to the reference length and a new length.

SCALETEXT This command is used to change the scale of the existing text. The original location of the text is not changed even after scaling.

SCENE Controls different scenes (particular view) in model space. Through a series of dialog boxes all the scenes in the current drawing are listed, new scenes can be added, scene names can be modified, and the lights can be controlled in the scene.

SCRIPT Executes a command script.

SECTION Creates regions from the intersection of a plane and solids. Options:

3points - Specifying 3 points on sectioning plane, O - Sectioning plane is aligned with the object, Z - Sectioning plane is aligned with the plane's normal direction, V - Sectioning plane is aligned with the viewing plane of current viewport, XY - Sectioning plane aligned with XY plane of UCS, YZ - Sectioning plane aligned with YZ plane of UCS, ZX - Sectioning plane aligned with ZX plane of UCS.

SELECT Creates a selection set of specified group of objects. Options:

AU - Automatic selection, A - Add mode - Objects are added to the selection set, ALL - Selects all objects, BOX - Objects inside or crossing a rectangle are selected, C - Objects are selected that lie inside and crossing an area specified by two points, CP - Those objects are selected that lie inside and crossing the polygon created by specifying points around the objects, F - Those objects are selected that are crossing the specified fence, G - Objects within a group are selected, L - Recently created object is selected, M - Objects are picked without highlighting them, P - Recent selection set is selected, R - Remove mode - Objects can be removed from the selection set, SI - Selects first object or a set of objects, U - Removes the most recently added object from the selection set, W - Selects those

objects that lie completely inside an area specified by two points, WP - Selects those objects that lie completely inside an area specified by picking points around the objects.

SETUV Lets you map materials onto geometry.

SETVAR Sets the values of the system variables. Options:

? - Lists the variables with their current values.

SHADEMODE Displays a shaded picture of the drawing in the current viewport. Options: 2D - Displays the objects using lines and curves to represent the boundaries, 3D - Displays the objects using lines and curves to represent the boundaries, 3D wireframe - Displays the objects using 3D wireframe representation, H - Hides lines representing back faces, F - Shades the objects between the polygon faces, G - Shades the objects and smooths the edges, L - Combines the Flat Shaded and Wireframe options, O - Combines the Gouraud Shaded and Wireframe options.

SHAPE Predefined shapes are inserted. Options:

? - Lists the shape names.

SHELL Permits the access to the commands in the operating system while in AutoCAD.

SHOWMAT Lists the material type and attachment method for a selected object.

SKETCH Allows you to draw freehand drawings. Options:

P - Pen- sketching pen raised and lowered, X - Reports the number of temporary lines drawn and then exits SKETCH Q - Temporary lines discarded and then exits SKETCH R - Temporary lines recorded as permanent, E - Removes portion of the temporary line, C

- Pen lowered for sketching, . - Draws a straight line from endpoint of sketched line to current position of pen.

SLICE Solid is cut with a plane. Options:
3points - Cutting plane specified by defining 3 points, O - Cutting plane aligned with an object (Circle, ellipse, elliptical arc, 2D spline, or polyline), Z - Cutting plane specified by locating a point on Z-axis, V - Cutting plane aligned to the viewing plane of the current viewport, XY - Cutting plane aligned with the XY plane, YZ - Cutting plane aligned with the YZ plane, ZX - Cutting plane aligned with the ZX plane.

'SNAP The movement of the cursor is constrained to the snap spacing. Options:
ON - Snap mode is turned on, OFF - Snap mode is turned off, A - Sets different X and Y spacings, R - Snap grid is rotated, S - Sets the style (Standard or Isometric) of the snap grid, T - Specifies the snap type (Polar or Grid).

SOLDRAW Generates profiles and sections in viewports created with **SOLVIEW**.

SOLID Draws polygons that are solid-filled.

SOLIDEDIT Edits faces and edges of 3D solid objects.

SOLPROF Creates profile images of three-dimensional solids.

SOLVIEW Creates floating viewports using orthographic projection to lay out multi and sectional view drawings of 3D solid and body objects.

SPACETRANS This command is used to translate the length values in one of the working environment (model space or paper space) of AutoCAD into the value equivalent to it in the other environment.

SPELL Allows spellcheck of text objects in a drawing. If an ambiguous word is found, then the dialog box is displayed that lists the alternatives for the word, or permits you to replace the current word with another one, or add the word to the dictionary.

SPHERE A 3D solid sphere is drawn. Options:
R - Radius of the sphere, D - Diameter of the sphere.

SPLINE Draws smooth spline curves between points. Options:
Point - Specify points to define the spline curve. Suboptions:
Point - Adds spline curve segments by specifying points, C - Spline curve is closed, F - Fit, Tolerance - The tolerance for fitting is changed, O - 2D or 3D splinefit polylines are changed to splines.

SPLINEDIT Allows you to edit a spline entity. Options:
F - Fit data is edited. Suboptions: A - Fit points are added, C - An open spline is closed, O - A closed spline is opened, D - Fit points are removed, M - Fit points are moved, P - A spline fit data is removed from database, T - Beginning and end tangents are edited, L - Tolerance value for spline fit are changed, X - Exits fit data option, C - An open spline is closed, O - A closed spline is opened, M - Move Vertex - The position of the control vertices is changed, R - Refines a spline by adding control points, or by increasing its order, or by changing the weight, E - Spline direction is reversed, U - Reverses the previous operation of **SPLINEDIT**.

STANDARDS This command is used to configure new standards to the current drawing.

STATS Displays a dialog box that provides the rendering statistics. It also saves the statistics to a file.

STATUS Lists the drawing statistics, modes, and extents.

STLOUT Creates a binary or ASCII file and stores the solid in the specified file.

STRETCH Stretches lines, arcs, and polylines by moving the endpoints to another specified location, and moves the objects.

STYLE Creates new text styles or modifies the existing ones through a dialog box.

STYLESMANAGER Displays Plot style Manager dialog box.

SUBTRACT Subtracts the area of one set of regions from another and subtracts the volume of one set of solids from another, thus creating a new composite region or solid.

SYSWINDOWS Arranges windows and is equivalent to standard Window menu options in Windows applications.

TABLET Aligns the tablet with the coordinate system of a paper drawing. Options: ON - Tablet mode is turned on, OFF - Tablet mode is turned off, CAL - Calibrates the tablet, CFG - Configures tablet menu area and screen pointing area.

TABSURF Creates a polygon mesh that represents a tabulated surface formed from a path curve and direction vector.

TEXT Writes text using a variety of character patterns. Displays text on screen as it is entered. Options:

Start Point - Specifies a start point for the text object, J - Controls justification of the text, Suboptions:

A - Specifies both text height and text orientation, F - Specifies that text fits within an area, C - Center aligned, M - Horizontally aligned, R - Right justified, TL - Top left, TC

- Top centered, TR - Top right, ML - Middle left, MC - Middle centered, MR - Middle right, BL - Bottom left, BC - Bottom centered, BR - Bottom right. S - Specifies the text style, which determines the appearance of the text characters.

TEXTSCR Flips to the text window from the graphics window.

TIME The date and time of drawing creation is displayed. It also displays the time and the date when the current drawing was last updated and controls an elapsed timer. Options: D - Displays the updated times, O - Elapsed timer is turned on, OFF - Elapsed timer is turned off, R - Resets the user elapsed timer.

TOLERANCE Creates and adds geometric tolerances to a drawing through a dialog box.

TOOLBAR Displays, hides, and customizes toolbars.

TOOLPALETTES Open the ToolPalettes to insert blocks or hatches in the current drawing.

TOOLPALETTESCLOSE Closes the ToolPalettes.

TORUS Draws a solid having the shape of a donut. Options: R - Radius of the tube, D - Diameter of the tube.

TRACE Draws filled lines having a specified width.

TRANSPARENCY Controls whether background pixels in an image are transparent or opaque.

TREESTAT Displays the current spatial index (position of objects in space) of a drawing. The information includes the number of

nodes, number of objects, depth of branch, and so forth.

TRIM Removes the extra portion of an entity which extends beyond a specified boundary. Options:

P - Sets projection mode, E - Controls trimming of objects until the implied edge, U - Reverses the previous operation of the **TRIM** command, O - Specifies the object to trim. U Reverses the most recent operation.

UCS Sets and modifies the user coordinate system. Options:

W - Current UCS set to World Coordinate System, N - New UCS, M - Moves the origin, G - Specifies one of the six orthographic UCSs, P - Restores the previous UCS, R - Restores a saved UCS so that it becomes the current, S - Saves the current UCS to a specified name, D - Removes the specified UCS from the list, A - Applies the current UCS setting, ? - Lists names of user coordinate systems.

UCSICON Manages the location and the visibility of the UCS icon. Options:

ON - Coordinate system icon is enabled, OFF - Coordinate system icon is disabled, A - Icon is changed in all active viewports, N - Icon displayed at the lower left corner, OR - Icon displayed at the origin of current coordinate system, P - displays the **UCS Icon** dialog box for controlling the settings related to the UCS icon.

UCSMAN Manages defined user coordinate

UNDEFINE A built-in AutoCAD command is disabled.

UNDO Reverses the effect of commands. Options:

N - The effect of a specified number of previous commands used is reversed, A The effect of the menu items is reversed by a single U command, C - The **UNDO** command is lim-

ited or is turned off, BE - A number of operations are grouped together and are treated as a single operation, E - The group is terminated, M - Mark - A marker is placed in the undo information, B - Back - Undoes all work until the marker is encountered.

UNION Combines the area of two or more regions, or the volume of two or more solids to create a composite region or solid.

UNITS Sets the coordinate and angle display formats and precision.

VBAIDE Displays the Visual Basic Editor.

VBALOAD Loads a global VBA project into the current AutoCAD session.

VBAMAN Loads, unloads, saves, creates, embeds, and extracts VBA projects.

VBARUN Runs a VBA macro applications.

VBASTMT Executes a VBA statement on the AutoCAD command line.

VBAUNLOAD Unloads a VBA global project. VIEW The graphics display is saved and restored as a view with a specified name through a dialog box.

VIEWRES Controls the appearance of objects by setting their resolution in the current viewport.

VLISP Displays the Visual LISP interactive development environment (IDE).

VPCLIP Clips the specified viewport objects.

VPLAYER Controls the visibility of layers in different viewports. Options:

? - Lists the frozen layers in a specified viewport, F - Layers are frozen in current, or all, or specified viewport, T - Layers are

thawed in current, or all, or specified viewport, R - Rests the layers default visibility, N - New layers that are frozen in all viewports are created, V - Viewport Visibility Default - Controls thawing and freezing of layers.

VPOINT The viewing direction for 3D visualization. Options:

ENTER - Displays compass and axis tripod for controlling viewing direction, V - Specifies a point from which drawing can be viewed, R - New direction using two angles is specified.

VPORTS Divides the graphics display into a number of viewports through a dialog box.

VSLIDE Displays an existing raster image slide file in the current viewport.

WBLOCK Writes a block definition or specified objects to a new disk file through a dialog box.

WEDGE Creates a 3D solid in the shape of a wedge having its one of the faces as tapered and sloping. Options:

point - Specifies the first corner of the wedge. Suboptions:

point - Specifies the other corner of the wedge, C - Wedge having sides of equal length, L - Wedge with specified length, width, and height, CE - Creates wedge with specified center point. Suboptions:

point - Specifies the other corner of the wedge, C - Creates wedge having all sides equal, L - Creates wedge with specified length, width, and height.

WHOHAS This command displays the information related to the owner of the selected drawing.

WIPEOUT Covers the specified area of the

drawing with a blank polygonal area.

WMFIN Imports a Windows metafile.

WMFOPTS Sets options for WMFIN.

WMFOUT Saves objects to a Windows metafile.

XATTACH Attaches an external reference to the current drawing.

XBIND Adds Xref's dependent symbols to a drawing through a dialog box.

XCLIP Defines an xref clipping boundary and sets the front or back clipping planes. Options:

ON - Displays the clipped portion, OFF - Displays all of the geometry of the xref ignoring the clipping boundary, C - Sets the front and back clipping planes, D - Removes a clipping boundary, P - Automatically draws a polyline coincident with the clipping boundary, N - Defines a rectangular or polygonal clipping boundary.

XLINE Creates a line of infinite length. Options:

point - Specifies the point through which the xline passes, H - Creates a horizontal xline, V - Creates a vertical xline, A - Creates a xline at an angle, B - Creates an xline through the vertex of two lines so that it bisects the angle between those two lines, O - Creates an xline parallel to another linear object.

XOPEN Open a selected xref file in a separate file. If the xref contains nested xrefs, the deepest-level xref of the selected object is opened.

XPLODE Breaks a compound object into its individual objects. Options:

G - Changes selected objects. Suboptions: E - Explodes the entire compound object, A - Sets color, linetype, layer of the component enti-

ties, C - Sets the color, LA - Sets the layer, LT - Sets the linetype, I - Sets all the properties to that of the original compound object, I - Changes selected objects one by one.

XREF Manages external references to a drawing through Xref Manager.

ZOOM Changes the display of the entities in the current drawing. Options:
value - Scale(X/XP) - Changes the display by a specified scale factor, Scale X - Zoom relative to current scale, Scale XP - Scale relative to paper space, A - Zooms the entire drawing in current viewport, C - Displays at a specified center point, D - Displays the portion of the drawing with a view box, E - Displays the drawing extents, S - Zooms the display at a specified scale factor, W - Displays an area specified by two corners of the window, Realtime - Using the pointing device, zooms interactively to a logical extent.