CMD,C,8	HELPLINE,C,78
3DFACE	The 3DFACE command (ADE-3) is similar to the SOLID command, but it
3DFACE	accepts Z coordinates for the corner points and can generate a section
3DFACE	of a plane or a nonplanar figure.
3DFACE	
3DFACE	Format: 3DFACE First point: (3D point)
3DFACE	Second point: (3D point)
3DFACE	Third point: (3D point)
3DFACE	Fourth point: (3D point, or RETURN for triangular section)
3DFACE	Third point: (3D point, or RETURN to end 3D face)
3DFACE	
3DFACE	To draw a face with four points, enter the points in a clockwise
3DFACE	or counterclockwise fashion to avoid a "bow tie" figure. Note that
3DFACE	this is different from the order expected by the SOLID command.
3DFACE	
3DFACE	3D faces are not solid-filled. The HIDE command considers them to be
3DFACE	opaque if they are planar. If nonplanar, HIDE draws a "wireframe"
3DFACE	representation.
3DFACE	
3DFACE	See also: The Release 9 reference manual supplement.
3DLINE	The 3DLINE command (ADE-3) draws straight lines and accepts Z
3DLINE	coordinates to form fully general space lines.
3DLINE	
3DLINE	Format: 3DLINE From point: (3D point)
3DLINE	To point: (3D point)
3DLINE	To point: (3D point)
3DLINE	To point: ...RETURN to end 3D line sequence
3DLINE	
3DLINE	To erase the latest line segment without exiting the 3DLINE command,
3DLINE	enter "U" when prompted for a "To" point.
3DLINE	
3DLINE	You can continue the previous line or arc by responding to the
3DLINE	"From point:" prompt with a space or RETURN. If you are drawing
3DLINE	a sequence of 3D lines that will become a closed polygon, you can
3DLINE	reply to the "To point" prompt with "C" to draw the last segment
3DLINE	(close the polygon).
3DLINE	
3DLINE	When using a pointing device, 3D lines may be constrained to horizontal
3DLINE	or vertical (in the XY plane) by the ORTHO command.
3DLINE	
3DLINE	See also: The Release 9 reference manual supplement.
APERTURE	The APERTURE command governs the size of the "target" crosshairs
APERTURE	for object snap purposes. This is an ADE-2 feature.
APERTURE	
APERTURE	Format: APERTURE
APERTURE	Object snap target height (1-50 pixels) <default>: (number)
APERTURE	
APERTURE	See also: Section 8.6 of the Autocad Reference Manual.
ARC	The ARC command draws an arc (circle segment) as specified by any of

ARC	the following methods. (If you have the ADE-2 package, you can "drag"
ARC	the last parameter of each method.)
ARC	
ARC	- three points on the arc
ARC	- start point, center, end point
ARC	- start point, center, included angle
ARC	- start point, center, length of chord
ARC	- start point, end point, radius
ARC	- start point, end point, included angle
ARC	- start point, end point, starting direction
ARC	- continuation of previous line or arc
ARC	
ARC	3-point format: ARC Center/<Start point>: (point)
ARC	Center/End/<Second point>: (point)
ARC	End point: (point)
ARC	
ARC	Options: A = included Angle D = starting Direction L = Length of chord
ARC	C = Center point E = End point R = Radius
ARC	To continue previous line or arc, reply to first prompt with RETURN
ARC	
ARC	See also: Section 4.4 of the Autocad Reference Manual.
AREA	The AREA command calculates the area and perimeter of an enclosed space
AREA	and adds the area to (or subtracts it from) a running total. You can
AREA	define the space by designating three or more points, as in:
AREA	
AREA	Format: AREA <First point>/Entity/Subtract: (point)
AREA	(ADD mode) Next point: (point)
AREA	(ADD mode) Next point: (point)
AREA	(ADD mode) Next point: ...press RETURN to end point entry
AREA	Area = nnnn Perimeter = nnnn
AREA	Total area = nnnn
AREA	<First point>/Entity/Subtract: ...press RETURN to exit
AREA	
AREA	The command begins in "add" mode. The options are:
AREA	
AREA	Add - (When in "subtract" mode) switches to "add" mode.
AREA	Subtract - (When in "add" mode) switches to "subtract" mode.
AREA	Entity - Computes the area of a selected Circle or Polyline
AREA	RETURN - A null reply exits the AREA command
AREA	
AREA	See also: The Release 9 supplement.
ARRAY	The ARRAY command makes multiple copies of selected objects, in a
ARRAY	rectangular or circular pattern.
ARRAY	
ARRAY	Format: ARRAY Select objects: (Show what to copy)
ARRAY	Rectangular/Polar array (R/P):
ARRAY	
ARRAY	For a rectangular array, you are asked for the number of columns and
ARRAY	rows, and the spacing between them. The array is built along a baseline

ARRAY	defined by the current Snap rotation angle set by the "SNAP Rotate" command.
ARRAY	
ARRAY	For a polar, or circular, array, you must first supply a center point.
ARRAY	Following this, you must supply two of the following three parameters:
ARRAY	
ARRAY	- the number of items in the array
ARRAY	- the number of degrees to fill
ARRAY	- the angle between items in the array
ARRAY	
ARRAY	Optionally, you can rotate the items as the array is drawn.
ARRAY	
ARRAY	See also: Section 5.2 of the Autocad Reference Manual.
ATTDEF	The ATTDEF command creates an Attribute Definition (ADE-2 feature).
ATTDEF	First, you specify the modes for this Attribute Definition. The modes are:
ATTDEF	
ATTDEF	Invisible - Do not display, but allow extraction.
ATTDEF	Constant - All occurrences of this Attribute have the same Value.
ATTDEF	Verify - Issue extra prompts to verify a proper Value.
ATTDEF	Preset - Do not prompt for this Attribute during Block insertion.
ATTDEF	
ATTDEF	Format: ATTDEF Attribute modes -- Invisible:N Constant:N Verify:N Preset:N
ATTDEF	Enter (ICVP) to change, RETURN when done:
ATTDEF	
ATTDEF	The tag, prompt, and default value for the Attribute are then requested,
ATTDEF	as are its location, height, and rotation angle.
ATTDEF	
ATTDEF	Attribute tag: (up to 31 chars; letters, digits, $, -, _)
ATTDEF	Attribute prompt:
ATTDEF	Default attribute value:
ATTDEF	Start point or Align/Center/Fit/Middle/Right/Style:
ATTDEF	Height <default>:
ATTDEF	Rotation angle <default>:
ATTDEF	
ATTDEF	See also: Section 9.2 of the Autocad Reference Manual, and the Release 9 sup
ATTDISP	The ATTDISP command (ADE-2 feature) can be used to override the visibility
ATTDISP	mode set for Attributes on a global basis.
ATTDISP	
ATTDISP	Format: ATTDISP Normal/ON/OFF <current>:
ATTDISP	
ATTDISP	Normal - Visible Attributes are displayed, invisible Attributes are not.
ATTDISP	On - All Attributes are made visible.
ATTDISP	Off - All Attributes are made invisible.
ATTDISP	
ATTDISP	See also: Section 9.2 of the Autocad Reference Manual.
ATTEDIT	The ATTEDIT command (ADE-2 feature) allows you to modify certain aspects
ATTEDIT	of Attributes independent of the Blocks in which they reside. You can
ATTEDIT	perform global or individual editing, and you can restrict the operation
ATTEDIT	to certain Blocks, Attribute Tags, and Attribute Values, or to just those
ATTEDIT	Attributes that are currently visible on the screen.

ATTEDIT	
ATTEDIT	Format: ATTEDIT Edit Attributes one by one? <Y> (N = global)
ATTEDIT	Block name specification <*>:
ATTEDIT	Attribute tag specification <*>:
ATTEDIT	Attribute value specification <*>:
ATTEDIT	
ATTEDIT	See also: Section 9.2 of the Autocad Reference Manual.
ATTEXT	The ATTEXT command (ADE-2 feature) is used to extract Attribute information
ATTEXT	from a drawing for analysis by another program or for transfer to a database.
ATTEXT	The whole drawing can be extracted or only a selected set of entities.
ATTEXT	
ATTEXT	Format: ATTEXT CDF, SDF, or DXF attribute extract (or Entities)? <C>:
ATTEXT	
ATTEXT	If you respond with an "CDF", "SDF", or "DXF", the entire drawing will
ATTEXT	be extracted. If you respond with an "E," the "Select objects:" prompt
ATTEXT	appears, and you may select a set of entities to extract. ATTEXT then
ATTEXT	again prompts:
ATTEXT	
ATTEXT	CDF, SDF or DXF Attribute extract? <C>:
ATTEXT	
ATTEXT	The extracted format is as follows:
ATTEXT	CDF - Comma Delimited Format
ATTEXT	SDF - Fixed-field format
ATTEXT	DXF - Drawing interchange format
ATTEXT	
ATTEXT	CDF format is the default, since it is simplest to read and permits
ATTEXT	selective extraction of just the desired Attributes.
ATTEXT	
ATTEXT	See also: Section 9.2 of the Autocad Reference Manual.
AXIS	The AXIS command controls the display of axes, or ruler lines,
AXIS	along the edge of the graphics display. This is an ADE-1 feature.
AXIS	
AXIS	Format: AXIS Tick spacing(X) or ON/OFF/Snap/Aspect <default>:
AXIS	
AXIS	Spacing(X) - A simple number sets axis tick spacing in drawing
AXIS	units. A number followed by "X" (e.g., "2X")
AXIS	sets the tick spacing to a multiple of the current
AXIS	Snap resolution. A value of zero locks the tick
AXIS	spacing to the current Snap resolution.
AXIS	ON - Turns axis on with previous spacing.
AXIS	OFF - Turns axis off.
AXIS	Snap - Locks the tick spacing to the current Snap
AXIS	resolution (same as a spacing value of zero).
AXIS	Aspect - (ADE-2) Permits an axis with different
AXIS	horizontal and vertical spacing.
AXIS	
AXIS	See also: Section 8.3 of the Autocad Reference Manual.
BASE	The BASE command defines a reference point for insertion
BASE	and rotation of the current drawing in subsequent drawings.

BASE	
BASE	Format: BASE Base point <default>: (point)
BASE	
BASE	See also: Section 9.1 of the Autocad Reference Manual.
BLIPMODE	The BLIPMODE command controls the generation of marker "blips" - the
BLIPMODE	small temporary marks drawn whenever you designate a point. When
BLIPMODE	BLIPMODE is "On", blips are drawn; when "Off", blips are suppressed.
BLIPMODE	
BLIPMODE	Format: BLIPMODE ON/OFF <current>:
BLIPMODE	
BLIPMODE	See also: Section 6.8 of the Autocad Reference Manual.
BLOCK	The BLOCK command allows you to name a group of objects that can
BLOCK	then be INSERTed as a unit anywhere in the current drawing, with
BLOCK	specified X and Y scales and rotation.
BLOCK	
BLOCK	Format: BLOCK Block name (or ?): (name)
BLOCK	Insertion base point: (point)
BLOCK	Select objects: (select)
BLOCK	
BLOCK	The objects you select will be erased as they are copied into the
BLOCK	Block. If you want to restore them, use the OOPS command.
BLOCK	
BLOCK	If you respond to the "Block name" prompt with a "?", AutoCAD will
BLOCK	list the names of all Blocks currently defined in this drawing.
BLOCK	
BLOCK	See also: Section 9.1 of the Autocad Reference Manual.
BREAK	The BREAK command deletes part of a Line, Trace, Circle, Arc, or
BREAK	Polyline, or splits the object into two objects of the same type.
BREAK	This is an ADE-1 feature.
BREAK	
BREAK	Format: BREAK Select object: (select one object)
BREAK	Enter first point: (point)
BREAK	Enter second point: (point)
BREAK	
BREAK	If you break a circle, it changes to an arc by deleting the portion from
BREAK	the first point to the second, going counterclockwise. Breaking a polyline
BREAK	with nonzero width will cause the ends to be cut square.
BREAK	If you select the object by pointing to it, the break is assumed
BREAK	to begin at the selection point, and the next prompt is:
BREAK	
BREAK	Enter second point (or F for first point):
BREAK	
BREAK	If you want to begin the break at a point where some other object
BREAK	intersects with the object to be broken, choose an unambiguous point
BREAK	to select the object, and then enter "F" in response to this prompt.
BREAK	You can then select the beginning and ending points of the break.
BREAK	
BREAK	See also: Section 5.3 of the Autocad Reference Manual.
CHAMFER	The CHAMFER command trims two intersecting lines (or two adjacent

CHAMFER	segments of a Polyline) at a given distance from their intersection and
CHAMFER	connects the trimmed ends with a new line. Different trim distances can
CHAMFER	be set for the two lines, and are retained with the drawing. If the
CHAMFER	specified lines do not intersect, CHAMFER will extend them until they do,
CHAMFER	and then proceed as above. If the ADE-3 package is present, chamfers can
CHAMFER	be applied to an entire Polyline, chamfering all the intersections.
CHAMFER	
CHAMFER	Format: CHAMFER Polyline/Distances/<select first line>:
CHAMFER	
CHAMFER	D - Set chamfer distances
CHAMFER	P - Chamfer entire Polyline
CHAMFER	
CHAMFER	See also: Section 5.3 of the Autocad Reference Manual.
CHANGE	The CHANGE command allows you to modify or change the properties
CHANGE	of existing objects in the drawing.
CHANGE	
CHANGE	Format: CHANGE Select objects: (select)
CHANGE	Properties/<Change point>:
CHANGE	
CHANGE	In the following descriptions, the selected Change Point is abbreviated
CHANGE	"CP". If the ADE-2 package is present, the selected point can be dragged.
CHANGE	
CHANGE	Line - Endpoint closest to CP changes to CP (ORTHO can affect this).
CHANGE	Circle - Radius changes so that CP is on circumference.
CHANGE	Block - Location changes to CP. New angle may be specified.
CHANGE	Text - Location changes to CP. New text style, height, angle,
CHANGE	and text string may be specified.
CHANGE	Attribute Definition - Same as Text, plus Attribute tag may be changed.
CHANGE	
CIRCLE	The CIRCLE command is used to draw a circle. You can specify the circle
CIRCLE	in several ways. The simplest method is by center point and radius.
CIRCLE	
CIRCLE	Format: CIRCLE 3P/2P/TTR/<Center point>: (point)
CIRCLE	Diameter/<Radius>: (radius value)
CIRCLE	
CIRCLE	To specify the radius, you can designate a point to be on the circumference.
CIRCLE	ADE-2 users can enter "DRAG" in response to the "Diameter/<Radius>" prompt
CIRCLE	to specify the circle size visually. If it is more convenient to enter the
CIRCLE	diameter than the radius, reply to the "Diameter/<Radius>" prompt with "D".
CIRCLE	
CIRCLE	The circle can also be specified using three points on the circumference
CIRCLE	(reply "3P" when prompted for the center point), or by designating two
CIRCLE	endpoints of its diameter (reply "2P"). For these methods, you can "drag"
CIRCLE	the last point or specify object snap "Tangent" points if you have the
CIRCLE	ADE-2 package.
CIRCLE	
CIRCLE	In addition, if you have the ADE-2 package, you can draw a circle by
CIRCLE	specifying two lines (and/or other circles) to which the circle should be
CIRCLE	tangent, and a radius. Enter "TTR" for this option.

CIRCLE	
CIRCLE	See also: Section 4.3 of the Autocad Reference Manual.
COLOR	Color numbers 1 through 7 have standard meanings as follows:
COLOR	
COLOR	1 - Red, 2 - Yellow, 3 - Green, 4 - Cyan, 5 - Blue, 6 - Magenta, 7 - White
COLOR	
COLOR	You can control the color of each entity individually or by layer. To change
COLOR	the color of existing objects, use the CHANGE command. To control layer
COLOR	colors, use LAYER. The COLOR command sets the color for new entities.
COLOR	
COLOR	Format: COLOR New entity color <current>:
COLOR	
COLOR	You can respond with a color number from 1 to 255, or a standard color name
COLOR	such as "Red". All new entities will be drawn in this color, regardless
COLOR	of which layer is current, until you again use the COLOR command.
COLOR	
COLOR	If you respond with "BYLAYER", new objects you draw will inherit the
COLOR	color assigned to the layer upon which they are drawn.
COLOR	
COLOR	If you respond with "BYBLOCK", objects will be drawn in white until they are
COLOR	grouped into a Block. Then, whenever that Block is inserted, the objects
COLOR	will inherit the color of the Block insertion.
COLOR	
COLOR	See also: Sections 5.3, 7.7, and 7.8 of the Autocad Reference Manual.
COPY	The COPY command is used to duplicate one or more existing drawing
COPY	entities at another location (or locations) without erasing the original.
COPY	
COPY	Format: COPY Select objects: (select)
COPY	<Base point or displacement>/Multiple:
COPY	Second point of displacement: (if base selected above)
COPY	
COPY	If you have the ADE-2 package, you can "drag" the object into position
COPY	on the screen. To do this, designate a reference point on the object in
COPY	response to the "Base point..." prompt, and then reply "DRAG" to the
COPY	"Second point:" prompt. The selected objects will follow the movements
COPY	of the screen crosshairs. Move the objects into position and then press
COPY	the pointer's "pick" button.
COPY	
COPY	To make multiple copies, respond to the "Base point" prompt with "M".
COPY	The "Base point" prompt then reappears, followed by repeated "Second point"
COPY	prompts. When you have made all the copies you need, give a null response
COPY	to the "Second point" prompt.
COPY	
COPY	See also: Section 5.2 of the Autocad Reference Manual.
DBLIST	The DBLIST command produces a complete list of the contents
DBLIST	of the drawing database for the current drawing. This command
DBLIST	is used mostly for debugging.
DBLIST	
DBLIST	Format: DBLIST

DBLIST	
DBLIST	You can use CTRL S to pause, and CTRL C to cancel the listing.
DBLIST	If you want to echo the listing to your printer, use CTRL P.
DBLIST	
DBLIST	See also: Section 5.6 of the Autocad Reference Manual.
DDATTE	The DDATTE command (ADE-3) lets you examine or change the values
DDATTE	of a Block's Attributes by means of a dialogue box. Dialogue boxes
DDATTE	work only with certain display drivers.
DDATTE	
DDATTE	See also: The Release 9 reference manual supplement.
DDEMODES	The DDEMODES command (ADE-3) lets you change various entity drawing
DDEMODES	modes (current layer, color, linetype, elevation, and thickness) using
DDEMODES	dialogue boxes. Dialogue boxes work only with certain display drivers.
DDEMODES	
DDEMODES	See also: The Release 9 reference manual supplement.
DDLMODES	The DDLMODES command (ADE-3) lets you create new layers, rename existing
DDLMODES	layers, select a different current layer, and control the visibility,
DDLMODES	color, freeze/thaw state, and linetype assigned to existing layers, using
DDLMODES	dialogue boxes. Dialogue boxes work only with certain display drivers.
DDLMODES	
DDLMODES	See also: The Release 9 reference manual supplement.
DDRMODES	The DDRMODES command (ADE-3) lets you control the settings of various
DDRMODES	drawing aids, such as Snap, Grid, and Axis, using dialogue boxes.
DDRMODES	Dialogue boxes work only with certain display drivers.
DDRMODES	
DDRMODES	See also: The Release 9 reference manual supplement.
DELAY	The DELAY command is used in command scripts to allow the display
DELAY	to be viewed before the next command is automatically issued. DELAY
DELAY	times are designed to be approximately 1 millisecond per increment, but
DELAY	are ultimately a function of the computer equipment running AutoCAD.
DELAY	
DELAY	Format DELAY Delay time in milliseconds: (number)
DELAY	
DELAY	The larger the number, the longer the delay.
DELAY	
DELAY	See also: Command scripts, Section 11.1 of the Autocad Reference Manual.
DIM	The DIM command (ADE-1 feature) enters Dimensioning mode. The commands
DIM	allowed during Dimensioning mode are listed below. Each may be
DIM	abbreviated to its first three characters. A space or RETURN will
DIM	repeat the previous DIM subcommand.
DIM	
DIM	ALIgned - Linear dimensioning, aligned with extension line origins
DIM	ANGular - Angular dimensioning
DIM	BASeline - Continue from 1st extension line of previous dimension
DIM	CENter - Draw center mark or center lines
DIM	CONtinue - Continue from 2nd extension line of previous dimension
DIM	DIAmeter - Diameter dimensioning
DIM	EXIt - Return to normal command mode
DIM	HORizontal - Linear dimensioning, horizontal dimension line

DIM	LEAder - Draw a leader to the dimension text
DIM	RADius - Radius dimensioning
DIM	REDraw - Redraw the display
DIM	ROTated - Linear dimensioning at specified angle
DIM	STAtus - List dimensioning variables and their values
DIM	STYle - Switches to a new text style
DIM	UNDo - Erase the annotation drawn by the last dimensioning command
DIM	VERtical - Linear dimensioning, vertical dimension line
DIM	
DIM1	The DIM1 command allows you to execute one dimensioning command, and
DIM1	then returns to normal command mode.
DIM1	
DIM1	Format: DIM1 Dim: (enter dimensioning command)
DIM1	
DIM1	See also: Section 10.1 of the Autocad Reference Manual.
DIST	The DIST command displays the distance (in drawing units), the angle,
DIST	and the delta-X/Y between two designated points.
DIST	
DIST	Format: DIST First point: (point)
DIST	Second point: (point)
DIST	
DIST	If a single number is entered in response to the "First point:" prompt,
DIST	DIST displays that number in the current UNITS format (ADE-1 feature).
DIST	
DIST	See also: Section 5.6 of the Autocad Reference Manual.
DIVIDE	The DIVIDE command (ADE-3) allows you to divide an entity into a
DIVIDE	specified number of equal length parts, placing markers along the
DIVIDE	objects at the dividing points.
DIVIDE	
DIVIDE	Format: DIVIDE Select object to divide: (point)
DIVIDE	<Number of segments>/Block:
DIVIDE	
DIVIDE	You can select a single line, arc, circle, or polyline. If you enter a
DIVIDE	segment count between 2 and 32767, Point entities will be placed along
DIVIDE	the object to divide it into that number of equal segments. You can
DIVIDE	request a specific Block to be inserted instead of the Point entities
DIVIDE	by responding to the second prompt with "B". AutoCAD will ask:
DIVIDE	
DIVIDE	Block name to insert:
DIVIDE	Align block with object? <Y>
DIVIDE	Number of segments:
DIVIDE	
DIVIDE	The block must currently be defined within the drawing. If you answer
DIVIDE	"Yes" to the "Align block?" prompt, the block will be rotated around its
DIVIDE	insertion point so that it is drawn tangent to the object being divided.
DIVIDE	
DOUGHNUT	The DOUGHNUT (or DONUT) command (ADE-3) draws a filled circle or ring.
DOUGHNUT	
DOUGHNUT	Format: DOUGHNUT Inside diameter <last>: (value or two points)

DOUGHNUT	Outside diameter <last>: (value or two points)
DOUGHNUT	Center of doughnut: (enter point)
DOUGHNUT	
DOUGHNUT	The "Center of doughnut" prompt is repeated for multiple locations of
DOUGHNUT	the doughnuts. You can "drag" the center point if you wish. A null
DOUGHNUT	response ends the DOUGHNUT command.
DOUGHNUT	
DOUGHNUT	The DOUGHNUT command constructs a closed Polyline (composed of wide
DOUGHNUT	arc segments) representing the specified object. Consequently, you may
DOUGHNUT	edit the resulting doughnut with PEDIT or any of the other editing
DOUGHNUT	commands that operate on Polylines. The solid-filling of doughnuts
DOUGHNUT	is subject to Fill mode.
DOUGHNUT	
DOUGHNUT	See also: Section 4.6 of the Autocad Reference Manual.
DRAG	\DRAGMODE
DRAG	The ADE-2 package allows you to draw certain entities (Circles, Arcs,
DRAG	Polylines, Blocks, and Shapes) dynamically, "dragging" them into
DRAG	position on the screen. Also, many of the editing commands can drag any
DRAG	existing object. Dragging is turned on by entering the word "DRAG" at
DRAG	appropriate points in the command prompt sequence.
DRAG	
DRAG	With some computer configurations, the dragging process may be time-
DRAG	consuming.
DRAG	
DRAG	Format: DRAGMODE ON/OFF/Auto <current>:
DRAG	
DRAG	When Drag mode is off, all "DRAG" requests are ignored, including those
DRAG	embedded in menu items. When Drag mode is on, dragging is permitted, and
DRAG	"DRAG" requests are honored when appropriate.
DRAG	
DRAG	If you set Drag mode to "Auto", dragging is enabled for every command that
DRAG	supports it. Dragging will be performed whenever possible, without the
DRAG	need to enter "DRAG" each time.
DRAG	
DRAG	See also: Section 6.9 of the Autocad Reference Manual.
DXBIN	The DXBIN command loads a ".dxb" ("drawing interchange binary") file into
DXBIN	an AutoCAD drawing. These files have a very compact format and are mainly
DXBIN	for internal use by programs such as CAD/camera (tm).
DXBIN	
DXBIN	Format: DXBIN DXB file: (filename)
DXBIN	
DXBIN	Do not type the ".dxb" file type; it is assumed.
DXBIN	
DXBIN	See also: Appendix C of the Autocad Reference Manual.
DXFIN	The DXFIN command reads a Drawing Interchange File and creates or
DXFIN	appends a drawing from it. If you want to DXFIN a total drawing, create
DXFIN	a new drawing using Main Menu task 1, and issue the DXFIN command before
DXFIN	drawing anything.
DXFIN	

DXFIN	Format: DXFIN File name: (name)
DXFIN	
DXFIN	If AutoCAD determines that the current drawing is not empty, it prints
DXFIN	the message
DXFIN	
DXFIN	Not a new drawing -- only ENTITIES section will be input.
DXFIN	
DXFIN	and proceeds to ignore all sections of the input file other than the
DXFIN	ENTITIES section.
DXFIN	
DXFIN	See also: Appendix C of the Autocad Reference Manual.
DXFOUT	The DXFOUT command creates a Drawing Interchange File from the current
DXFOUT	drawing or from selected entities in the drawing.
DXFOUT	
DXFOUT	Format: DXFOUT File name: (name or RETURN)
DXFOUT	Enter decimal places of accuracy (0 to 16) (or Entities) <6>:
DXFOUT	
DXFOUT	If you respond with "E", the normal "Select objects:" prompt appears, and
DXFOUT	you may select the set of entities to be output. You are then again
DXFOUT	prompted with:
DXFOUT	
DXFOUT	Enter decimal places of accuracy (0 to 16) <6>:
DXFOUT	
DXFOUT	See also: Appendix C of the Autocad Reference Manual.
ELEV	The ELEV command is part of 3D Level 1(tm), contained in the ADE-3 package.
ELEV	It allows you to specify the current Elevation and Extrusion Thickness for
ELEV	subsequently drawn objects. The elevation is the Z plane on which an
ELEV	object's base is drawn, while its extrusion thickness is its height above
ELEV	that base elevation. Negative thickness extrudes downward.
ELEV	
ELEV	Format: ELEV
ELEV	New current elevation <current>: (RETURN or number)
ELEV	New current thickness <current>: (RETURN or number)
ELEV	
ELEV	See also: Section 14.2 of the Autocad Reference Manual.
ELLIPSE	The ELLIPSE command (ADE-3) allows you to draw ellipses.
ELLIPSE	
ELLIPSE	Format: ELLIPSE <Axis endpoint 1>/Center: (point)
ELLIPSE	Axis endpoint 2: (point)
ELLIPSE	<Other axis distance>/Rotation:
ELLIPSE	
ELLIPSE	If you enter a distance to the "<Other axis distance>/Rotation" prompt,
ELLIPSE	AutoCAD interprets it as half the length of the other axis. If you reply
ELLIPSE	with "R", the first axis is assumed major and AutoCAD prompts:
ELLIPSE	
ELLIPSE	Rotation around major axis:
ELLIPSE	
ELLIPSE	The major axis is now treated as the diameter line of a circle which will
ELLIPSE	be rotated a specified amount around the axis, into the third dimension.

ELLIPSE	You can enter a rotation angle between 0 and 89.4 degrees.
ELLIPSE	
ELLIPSE	If you respond to the "<Axis endpoint 1>/Center" prompt with "C", AutoCAD
ELLIPSE	prompts for the center point, and one endpoint of each axis. The
ELLIPSE	"<Other axis distance>/Rotation:" prompt appears for this method also,
ELLIPSE	so you can specify the ellipse's rotation rather than the second axis.
ELLIPSE	
END	The END command exits the Drawing Editor (after saving the updated
END	version of the current drawing), and returns to the Main Menu. If you
END	then wish to exit entirely, select item 0 from the Main Menu.
END	
END	Format: END
END	
END	See also: Section 3.2 of the Autocad Reference Manual.
ERASE	The ERASE command lets you delete selected entities from the drawing.
ERASE	
ERASE	Format: ERASE Select objects: (select)
ERASE	
ERASE	You can easily erase just the last object you drew by responding to
ERASE	the "Select objects" prompt with "L".
ERASE	
ERASE	The OOPS command can be used to retrieve the last thing you erased.
ERASE	
ERASE	See also: Section 5.1 of the Autocad Reference Manual.
EXPLODE	The EXPLODE command (ADE-3) replaces a block reference with copies of
EXPLODE	the simple entities comprising the block, forms simple lines and arcs
EXPLODE	from a polyline, or forms individual lines, arrows, and text entities
EXPLODE	from an Associative Dimension entity.
EXPLODE	
EXPLODE	Format: EXPLODE Select block reference, polyline, or dimension.
EXPLODE	
EXPLODE	When a Block or Dimension is exploded, the resulting image on the screen
EXPLODE	is identical, except that the color and linetype of entities may change
EXPLODE	due to floating layers, colors, or linetypes. Therefore, be careful to
EXPLODE	select the desired object.
EXPLODE	
EXPLODE	When a polyline is exploded, any associated width or tangent information
EXPLODE	is discarded and the resulting lines and arcs follow the polyline's
EXPLODE	center line.
EXPLODE	
EXPLODE	See also: Section 5.4 of the Autocad Reference Manual.
EXTEND	The EXTEND Command (ADE-3) allows you to lengthen existing objects in
EXTEND	a drawing so they end precisely at a boundary defined by one or more
EXTEND	other objects in the drawing.
EXTEND	
EXTEND	Format: EXTEND Select boundary edges(s)...
EXTEND	Select objects:
EXTEND	
EXTEND	You may use any form of entity selection to define the boundary objects.

EXTEND	Lines, Arcs, Circles, and Polylines may serve as boundary objects. When
EXTEND	using a Polyline as a boundary, all width information associated with the
EXTEND	Polyline is ignored so that objects are extended to its center line.
EXTEND	
EXTEND	All the selected edges are highlighted and will remain highlighted for
EXTEND	the rest of the EXTEND command. Next the prompt:
EXTEND	
EXTEND	Select object to extend:
EXTEND	
EXTEND	appears. Pick objects to extend by pointing to the part of the object to
EXTEND	be extended. Answer with RETURN to end the command. Lines, Arcs, and
EXTEND	open Polylines can be extended.
EXTEND	
EXTEND	See also: Section 5.3 of the Autocad Reference Manual.
FILES	The FILES command is used to gain access to disk file directories.
FILES	
FILES	Format: FILES
FILES	
FILES	This invokes the File Utility menu, which displays a list of subtasks.
FILES	Using this menu, you can list the names of files on disk, delete
FILES	selected files, rename a file, or copy a file to another file.
FILES	
FILES	When listing user-specified files or deleting files, you can use the "*"
FILES	and "?" wild-card characters. "?" matches any character in that position,
FILES	and "*" matches all characters up to a period, or to the end of the name.
FILES	Thus, "*.*" means all files.
FILES	
FILES	See also: Section 3.7 of the Autocad Reference Manual.
FILL	The FILL command controls whether Solids, Traces, and wide Polylines
FILL	are to be solid-filled or just outlined.
FILL	
FILL	Formats: FILL ON - Solids, Traces, and wide Polylines filled
FILL	FILL OFF - Solids, Traces, and wide Polylines outlined
FILL	
FILL	See also: Section 6.6 of the Autocad Reference Manual.
FILLET	The FILLET command connects two lines, arcs, or circles with a smooth
FILLET	arc of specified radius. It adjusts the lengths of the original lines
FILLET	or arcs so they end exactly on the fillet arc. If the Polyline option
FILLET	is used, you can apply fillets to an entire Polyline, or remove the
FILLET	fillets from a Polyline. FILLET is an ADE-1 feature, but the Polyline
FILLET	extensions require ADE-3.
FILLET	
FILLET	Format: FILLET Polyline/Radius/<select two objects>:
FILLET	
FILLET	P - Fillet an entire Polyline
FILLET	R - Set the fillet radius
FILLET	
FILLET	See also: Section 5.3 of the Autocad Reference Manual.
FILMROLL	The FILMROLL command lets you produce a file for use by the AutoShade

FILMROLL	shaded rendering package.
FILMROLL	
FILMROLL	Format: FILMROLL Enter the filmroll file name <default>:
FILMROLL	
FILMROLL	Enter the name of the filmroll file you wish to create. The name of
FILMROLL	the current drawing is offered as the default. Do not include a file
FILMROLL	type in your response; file type ".flm" is assumed.
FILMROLL	
FILMROLL	See also: The AutoShade user guide
GRAPHSCR	The GRAPHSCR and TEXTSCR commands are provided as a convenient means
GRAPHSCR	of selecting either the graphics or text screens from within menus and
GRAPHSCR	scripts.
GRAPHSCR	
GRAPHSCR	Format: GRAPHSCR or TEXTSCR
GRAPHSCR	
GRAPHSCR	See also: Section 11.1 of the Autocad Reference Manual.
GRID	The GRID command controls the display of a grid of alignment dots to assist
GRID	in the placement of objects in the drawing.
GRID	
GRID	Format: GRID Grid spacing(X) or ON/OFF/Snap/Aspect <current>:
GRID	
GRID	The various options are described below.
GRID	
GRID	Spacing(X) - A simple number sets grid spacing in drawing
GRID	units. A number followed by "X" (e.g., "2X")
GRID	sets the grid spacing to a multiple of the current
GRID	Snap resolution. A value of zero locks the grid
GRID	spacing to the current Snap resolution.
GRID	ON - Turns grid on with previous spacing.
GRID	OFF - Turns grid off.
GRID	Snap - Locks the grid spacing to the current Snap
GRID	resolution (same as a spacing value of zero).
GRID	Aspect - (ADE-2) Permits a grid with different
GRID	horizontal and vertical spacing.
GRID	
GRID	See also: Section 8.2 of the Autocad Reference Manual.
HATCH	The HATCH command is used to crosshatch or pattern-fill an area.
HATCH	This is an ADE-1 feature.
HATCH	
HATCH	Format: HATCH Pattern (? or name/U,style) <default>:
HATCH	
HATCH	Styles: N - Normal (BRICK,N or U,N)
HATCH	O - Outermost area only (BRICK,O or U,O)
HATCH	I - Ignore internal structure (BRICK,I or U,I)
HATCH	
HATCH	If you reply with a standard pattern, a scale and angle for the pattern
HATCH	are requested. "?" lists the standard patterns on file, and "U" prompts
HATCH	you to define a pattern on the fly:
HATCH	

HATCH	Angle for crosshatch lines <default>:
HATCH	Spacing between lines <default>:
HATCH	Double hatch area (Y/N) <default>:
HATCH	
HATCH	The specified parameters are remembered and are displayed as the defaults
HATCH	for subsequent HATCH command.
HATCH	
HATCH	See also: Section 10.2 of the Autocad Reference Manual.
HELP	\?
HELP	The HELP (or "?") command displays help information.
HELP	
HELP	Formats: HELP (or ?)
HELP	Command name (RETURN for list):
HELP	
HELP	If you reply with a command name, information about that command is
HELP	displayed. Otherwise, the display consists of a list of valid commands,
HELP	and a brief reminder of the methods of point specification.
HELP	
HELP	If the help information does not fit on one screen, AutoCAD will pause
HELP	and display:
HELP	
HELP	Press RETURN for further help.
HELP	
HELP	To continue the help display, press RETURN. If you want to cancel
HELP	the help display, enter CTRL C.
HELP	
HELP	See also: Section 3.1 of the Autocad Reference Manual.
HIDE	The HIDE command is part of 3D Level 1(tm), contained in the ADE-3 package.
HIDE	When the VPOINT command is used to generate a 3D visualization, it is in
HIDE	"wire frame" form; that is, all lines are drawn, even those that would be
HIDE	hidden by other objects. HIDE, which has no parameters, regenerates the
HIDE	drawing with the "hidden" lines suppressed.
HIDE	
HIDE	Format: HIDE
HIDE	
HIDE	See also: Section 14.2 of the Autocad Reference Manual.
ID	The ID command displays the coordinates of a designated point
ID	in the drawing.
ID	
ID	Format: ID Point: (point)
ID	
ID	See also: Section 5.6 of the Autocad Reference Manual.
IGESIN	The IGESIN command (ADE-3) reads an IGES ASCII format file and creates a
IGESIN	drawing from it. Create a new drawing using Main Menu task 1, and issue
IGESIN	the IGESIN command before drawing anything.
IGESIN	
IGESIN	Format: IGESIN File name: (name)
IGESIN	
IGESIN	See also: Appendix C of the Autocad Reference Manual.

IGESOUT	The IGESOUT command (ADE-3) creates a IGES ASCII format file from the
IGESOUT	current drawing.
IGESOUT	
IGESOUT	Format: IGESOUT File name: (name)
IGESOUT	
IGESOUT	See also: Appendix C of the Autocad Reference Manual.
INSERT	The INSERT command inserts one occurrence of a defined Block into the
INSERT	current drawing at a designated point, applying scale factors and
INSERT	rotation. If the named Block is not defined in the current drawing, but
INSERT	another drawing exists with that name, a Block Definition is first
INSERT	created from the other drawing.
INSERT	
INSERT	Format: INSERT Block name (or ?) <default>:
INSERT	Insertion point:
INSERT	X scale factor <1> / Corner / XYZ:
INSERT	Y scale factor (default = X):
INSERT	Rotation angle <0>:
INSERT	
INSERT	The X/Y scales may be specified simultaneously by using the insertion point
INSERT	as the lower left corner of a box, and a new point as the upper right corner;
INSERT	just enter the new point in response to the "X scale factor" prompt.
INSERT	
INSERT	If the ADE-2 package is present, you can enter "DRAG" to dynamically specify
INSERT	the insertion point, X/Y scales, and rotation angle. You can preset the
INSERT	scale and rotation for the dragged image by using the "Scale" or "Rotate"
INSERT	option at the "Insertion point:" prompt. See your user manual supplement
INSERT	for other options available at this prompt.
INSERT	
ISOPLANE	The ISOPLANE command permits selection of the current drawing plane
ISOPLANE	(top, left, or right) when the Isometric snap style (ADE-2 feature)
ISOPLANE	is in effect.
ISOPLANE	
ISOPLANE	Format: ISOPLANE Left/Top/Right/(Toggle):
ISOPLANE	
ISOPLANE	Left - Plane defined by 150 and 90 degree axis pair
ISOPLANE	Top - Plane defined by 30 and 150 degree axis pair
ISOPLANE	Right - Plane defined by 30 and 90 degree axis pair
ISOPLANE	RETURN - Toggles to the next plane in a circular fashion
ISOPLANE	
ISOPLANE	See also: Section 8.5 of the Autocad Reference Manual.
KEYS	\TOGGLES
KEYS	The following control keys are used to toggle various modes on and off.
KEYS	
KEYS	CTRL B - Snap mode on/off
KEYS	CTRL D - (ADE-1 feature) Coordinate display control. Static,
KEYS	dynamic with length<angle, dynamic with coordinates only.
KEYS	CTRL E - (ADE-2 feature) Circular toggle of ISO plane
KEYS	CTRL G - Grid on/off
KEYS	CTRL O - Ortho mode on/off

KEYS	CTRL P - Printer echo on/off
KEYS	CTRL T - Tablet mode on/off
KEYS	
KEYS	See also: Section 8.8 of the Autocad Reference Manual.
KEYS	Chapter 2 of your Installation Guide.
LAYER	The LAYER command allows you to control which drawing layer you are
LAYER	currently drawing on, and which drawing layers are to be displayed. It
LAYER	also controls the color and linetype associated with each drawing layer.
LAYER	
LAYER	Format: LAYER ?/Set/New/ON/OFF/Color/Ltype/Freeze/Thaw:
LAYER	
LAYER	? wildname - List layers, with colors and linetypes
LAYER	MAKE name - Create a new layer and make it current
LAYER	SET name - Set current layer
LAYER	NEW name,name - Create new layers
LAYER	ON wildname - Turn on specified layers
LAYER	OFF wildname - Turn off specified layers
LAYER	COLOR c wildname - Assign color "c" to specified layers
LAYER	LTYPE x wildname - Assign linetype "x" to specified layers
LAYER	FREEZE wildname (+3) - Completely ignore layers during regeneration
LAYER	THAW wildname (+3) - "Unfreeze" specified layers
LAYER	LTYPE ? - List loaded linetypes
LAYER	
LAYER	Where "wildname" appears above, the layer name(s) may include "*" and "?"
LAYER	wild cards. A single "*" selects all existing layers.
LAYER	
LAYER	See also: Section 7.7 of the Autocad Reference Manual.
LIMITS	The LIMITS command allows you to change the upper and lower limits
LIMITS	of the drawing area while working on a drawing, and to turn limits
LIMITS	checking ON or OFF.
LIMITS	
LIMITS	Format: LIMITS
LIMITS	ON/OFF/Lower left corner <current>:
LIMITS	Upper right corner <current>:
LIMITS	
LIMITS	See also: Section 3.5 of the Autocad Reference Manual.
LINE	The LINE command allows you to draw straight lines.
LINE	
LINE	Format: LINE From point: (point)
LINE	To point: (point)
LINE	To point: (point)
LINE	To point: ...RETURN to end line sequence
LINE	
LINE	To erase the latest line segment without exiting the LINE command,
LINE	enter "U" when prompted for a "To" point.
LINE	
LINE	You can continue the previous line or arc by responding to the
LINE	"From point:" prompt with a space or RETURN. If you are drawing
LINE	a sequence of lines that will become a closed polygon, you can

LINE	reply to the "To point" prompt with "C" to draw the last segment
LINE	(close the polygon).
LINE	
LINE	Lines may be constrained to horizontal or vertical by the ORTHO command.
LINE	
LINE	See also: Section 4.1 of the Autocad Reference Manual.
LINETYPE	You can control the dot-dash linetype of each entity individually,
LINETYPE	or by layer. To change the linetype of existing objects, use the
LINETYPE	CHANGE command. To control layer linetypes, use the LAYER command.
LINETYPE	
LINETYPE	The LINETYPE command sets the linetype for new entities. It
LINETYPE	can also load linetype definitions from a library file, write new
LINETYPE	definitions to a library file, and list the linetype definitions in a
LINETYPE	library file.
LINETYPE	
LINETYPE	Format: LINETYPE ?/Create/Load/Set:
LINETYPE	
LINETYPE	? - Lists the linetypes defined in a specified library file.
LINETYPE	Create - Allows creation of a new linetype and stores it in a specified
LINETYPE	library file.
LINETYPE	Load - Loads selected linetypes from a specified library file.
LINETYPE	Set - Sets the current linetype used for newly drawn entities.
LINETYPE	
LINETYPE	Note: The "Set" option and The "LAYER Ltype" command automatically load
LINETYPE	linetypes from the standard linetype library file. The "Load" option
LINETYPE	is needed only if you are storing linetypes in a different library file.
LINETYPE	
LIST	The LIST command displays database information about selected
LIST	objects.
LIST	
LIST	Format: LIST Select objects: (select)
LIST	
LIST	If the listing is lengthy, you can use CTRL S to pause momentarily,
LIST	or CTRL C to abort the listing. To echo the listing to your printer,
LIST	use CTRL P.
LIST	
LIST	See also: Section 5.6 of the Autocad Reference Manual.
LOAD	The LOAD command is used to load Shape definitions from a library file.
LOAD	
LOAD	Format: LOAD Name of shape file to load (or ?): (Shape file name)
LOAD	
LOAD	No file type should be specified; type ".shx" is assumed.
LOAD	
LOAD	If you respond to the LOAD command's prompt with "?", AutoCAD will
LOAD	display a list of the currently-loaded Shape files.
LOAD	
LOAD	See also: Section 4.11 of the Autocad Reference Manual.
LTSCALE	The LTSCALE command governs the global scale factor for linetype dash
LTSCALE	lengths.

LTSCALE
LTSCALE Format: LTSCALE New scale factor <current>:
LTSCALE
LTSCALE See also: Section 7.11 of the Autocad Reference Manual.
LTYPE There is no LTYPE command; see LAYER and LINETYPE.
LTYPE
LTYPE See also: Sections 7.7 and 7.9 of the Autocad Reference Manual.
MEASURE The MEASURE command (ADE-3) allows you to measure an entity, placing
MEASURE markers along the object at intervals of the specified distance.
MEASURE
MEASURE Format: MEASURE Select object to measure: (point)
MEASURE <Segment length>/Block:
MEASURE
MEASURE You can select a single line, arc, circle, or polyline. If you enter a
MEASURE segment length, the object is measured into segments of that length,
MEASURE starting at the endpoint closest to the point by which the entity was
MEASURE selected. Point entities will be placed where each pair of segments meet.
MEASURE You can request a specific Block to be inserted instead of the Point
MEASURE entities by responding to the second prompt with "B". AutoCAD will ask:
MEASURE
MEASURE Block name to insert:
MEASURE Align block with object? <Y>
MEASURE Segment length:
MEASURE
MEASURE The block must currently be defined within the drawing. If you answer
MEASURE "Yes" to the "Align block?" prompt, the block will be rotated around its
MEASURE insertion point so that it is drawn tangent to the object being measured.
MENU The MENU command is used to load a new set of commands into the
MENU screen, tablet, and button menus from a disk file.
MENU
MENU Format: MENU
MENU MEnu file name or . for none <current>:
MENU
MENU If you give a null response, the previous menu file is retained. If
MENU you respond with ".", the current menu will be cleared and no menu
MENU file will be loaded.
MENU
MENU See also: Section 3.7 of the Autocad Reference Manual.
MINSERT The MINSERT command is very similar to the INSERT command in that it is
MINSERT used to insert a Block. However, the MINSERT command creates multiple
MINSERT instances of the block in a rectangular pattern, or array.
MINSERT
MINSERT During the MINSERT command, AutoCAD asks the same questions as for the
MINSERT INSERT command (insert point, X/Y scaling, rotation angle, etc.).
MINSERT "MINSERT *" is not permitted, however. Following the standard INSERT
MINSERT prompts, the MINSERT command will prompt:
MINSERT
MINSERT Number of rows (---):
MINSERT Number of columns (|||):

MINSERT	Unit cell or distance between rows (---): (if row count is
MINSERT	2 ore more)
MINSERT	
MINSERT	Distance between columns (|||): (if column count is 2 or more
MINSERT	and unit cell was not selected)
MINSERT	
MINSERT	The Unit cell allows you to designate two opposite corners of a rectangle
MINSERT	to "show" AutoCAD the row and column spacing in one operation.
MINSERT	
MINSERT	You cannot EXPLODE a MINSERT. See also INSERT and ARRAY.
MINSERT	
MINSERT	See also: Section 9.1 of the Autocad Reference Manual.
MIRROR	The MIRROR command (ADE-2 feature) allows you to mirror selected
MIRROR	entities in your drawing. The original objects can be deleted (like a
MIRROR	MOVE) or retained (like a COPY).
MIRROR	
MIRROR	Format: MIRROR Select objects: (select)
MIRROR	First point of mirror line: (point)
MIRROR	Second point: (point)
MIRROR	Delete old objects? <N> (Yes, No, or RETURN)
MIRROR	
MIRROR	The mirror line you designate is the axis about which the selected objects
MIRROR	are mirrored; it may be at any angle.
MIRROR	
MIRROR	Often, you will want to reflect a section of a drawing but keep all its
MIRROR	annotation readable the usual way. AutoCAD permits this through the
MIRROR	MIRRTEXT system variable. When MIRRTEXT is set to 1 (the default value),
MIRROR	text will be reflected normally and will be mirror-inverted. If you set
MIRROR	MIRRTEXT to zero (using the SETVAR command or AutoLISP), the MIRROR command
MIRROR	will handle text items (and Attribute entities) specially, preventing them
MIRROR	from being reversed or turned upside down in the mirrored image.
MIRROR	
MIRROR	See also: Section 5.2 of the Autocad Reference Manual.
MOVE	The MOVE command is used to move one or more existing drawing
MOVE	entities from one location in the drawing to another.
MOVE	
MOVE	Format: MOVE Select objects: (select)
MOVE	Base point or displacement:
MOVE	Second point of displacement: (if base selected above)
MOVE	
MOVE	If you have the ADE-2 package, you can "drag" the object into position
MOVE	on the screen. To do this, designate a reference point on the object in
MOVE	response to the "Base point..." prompt, and then reply "DRAG" to the
MOVE	"Second point:" prompt. The selected objects will follow the movements
MOVE	of the screen crosshairs. Move the objects into position and then press
MOVE	the pointer's "pick" button.
MOVE	
MOVE	See also: Section 5.2 of the Autocad Reference Manual.
MSLIDE	The MSLIDE command (ADE-2 feature) "takes a picture" of the current

MSLIDE	display, and saves it in a slide file for later viewing with the VSLIDE
MSLIDE	command.
MSLIDE	
MSLIDE	Format: MSLIDE Slide file <current>: (name)
MSLIDE	
MSLIDE	The current drawing name is supplied as a default.
MSLIDE	
MSLIDE	The display is redrawn as the slide is being made.
MSLIDE	
MSLIDE	See also: Section 11.2 of the Autocad Reference Manual.
MULTIPLE	The MULTIPLE command instructs AutoCAD to repeat the next command you
MULTIPLE	enter, until cancelled by a CTRL C. No prompt is issued when you enter
MULTIPLE	the MULTIPLE command, so you can think of it as a modifier for the next
MULTIPLE	command. For instance:
MULTIPLE	
MULTIPLE	Command: MULTIPLE CIRCLE
MULTIPLE	
MULTIPLE	would cause the CIRCLE command to be repeated until you enter CTRL C to
MULTIPLE	stop it. Only the command name is repeated (not the options you may
MULTIPLE	have entered during the command).
MULTIPLE	
MULTIPLE	See also: The Release 9 reference manual supplement.
OFFSET	The OFFSET command (ADE-3) constructs an entity parallel to another
OFFSET	entity at either a specified distance or through a specified point.
OFFSET	You can OFFSET a Line, Arc, Circle, or Polyline.
OFFSET	
OFFSET	Format: OFFSET Offset distance or Through <last>:
OFFSET	Select object to offset: (point to the object)
OFFSET	
OFFSET	To offset from a wide Polyline, measure the offset distance from the
OFFSET	center-line of the Polyline. Once the object is selected, it is
OFFSET	highlighted on the screen. Depending on whether you specified an
OFFSET	offset distance or selected "through point" in the original prompt, you
OFFSET	will receive one of the following prompts:
OFFSET	
OFFSET	Side to offset:
OFFSET	Through point:
OFFSET	
OFFSET	The offset is then calculated and drawn. The selected object will be
OFFSET	de-highlighted and the "Select object to offset" prompt is re-issued.
OFFSET	RETURN exits the command.
OFFSET	
OFFSET	See also: Section 5.3 of the Autocad Reference Manual.
OOPS	The OOPS command re-inserts the object or objects that were deleted
OOPS	by the most recent ERASE command.
OOPS	
OOPS	Format: OOPS
OOPS	
OOPS	For a general method of reversing the effect of most commands,

OOPS	see the UNDO command.
OOPS	
OOPS	See also: OOPS command, Section 5.1 of the Autocad Reference Manual.
OOPS	UNDO command, Section 5.5 of the Autocad Reference Manual.
ORTHO	The ORTHO command allows you to control "orthogonal" drawing
ORTHO	mode. All lines and traces drawn while this mode is on are constrained
ORTHO	to be horizontal or vertical.
ORTHO	
ORTHO	Formats: ORTHO ON - Turn orthogonal mode on.
ORTHO	ORTHO OFF - Turn orthogonal mode off.
ORTHO	
ORTHO	Note: If the ADE-2 package is present, the Snap grid may be rotated.
ORTHO	If this is the case, Ortho mode rotates accordingly. Also, if the
ORTHO	Isometric snap style is in effect, Ortho mode is applied to the axis pair
ORTHO	associated with the current ISO plane.
ORTHO	
ORTHO	See also: Section 8.4 of the Autocad Reference Manual.
OSNAP	The OSNAP command is used to set "running" object snap modes. Object
OSNAP	(geometric) snap is an ADE-2 feature allowing you to designate points that
OSNAP	are related to objects already in your drawing.
OSNAP	
OSNAP	Format: OSNAP Object snap modes:
OSNAP	
OSNAP	CENter - Center of Arc or Circle
OSNAP	ENDpoint - Closest endpoint of Line, Arc, or 3D Line
OSNAP	INSertion - Insertion point of Text/Block/Shape/Attribute
OSNAP	INTersection - Intersection of Lines/Arcs/Circles or corner of
OSNAP	Trace/Solid/3D Face
OSNAP	MIDpoint - Midpoint of Line, Arc, or 3D Line
OSNAP	NEArest - Nearest point on Line/Arc/Circle/Point
OSNAP	NODe - Nearest Point entity (or Dimension definition point)
OSNAP	NONe - None (off)
OSNAP	PERpendicular - Perpendicular to Line/Arc/Circle
OSNAP	QUAdrant - Quadrant point of Arc or Circle
OSNAP	QUIck - Quick mode (first find, not closest)
OSNAP	TANgent - Tangent to Arc or Circle
OSNAP	
PAN	The PAN command allows you to move the display window in any direction,
PAN	without changing its magnification. This lets you see details that are
PAN	currently off the screen.
PAN	
PAN	You can specify a relative movement, as in:
PAN	
PAN	Format: PAN Displacement: (relative coordinates)
PAN	Second point: (RETURN)
PAN	
PAN	Or you can designate two points to specify the displacement you wish.
PAN	
PAN	Format: PAN Displacement: (point)

PAN	Second point: (point)
PAN	
PAN	See also: Section 6.2 of the Autocad Reference Manual.
PEDIT	The PEDIT command, part of the ADE-3 package, supports numerous ways of
PEDIT	editing Polylines. You can:
PEDIT	
PEDIT	- Open or close Polylines;
PEDIT	- Break polylines into pieces or join pieces into Polylines;
PEDIT	- Change the width and/or taper of the Polyline or specific segments;
PEDIT	- Move existing vertices, or insert new ones.
PEDIT	- Fit curves to the line, or remove curves and kinks;
PEDIT	
PEDIT	Format:
PEDIT	
PEDIT	PEDIT Select Polyline: (select)
PEDIT	Close/Join/Width/Edit vertex/Fit curve/Spline curve/Decurve/Undo/eXit <X>:
PEDIT	
PEDIT	"Close" will be replaced by "Open" if the Polyline is currently closed.
PEDIT	
PLINE	The ADE-3 package supports entities called Polylines, connected sequences
PLINE	of line and arc segments treated as a single entity. The PLINE command
PLINE	draws Polylines.
PLINE	
PLINE	Format: PLINE From point: (select)
PLINE	Current line width is nnn
PLINE	
PLINE	Line mode: Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>:
PLINE	
PLINE	Arc mode: Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/
PLINE	Second pt/Undo/Width/<Endpoint of arc>:
PLINE	
PLINE	To alter an existing Polyline, use the PEDIT command.
PLINE	
PLINE	See also: Section 4.6 of the Autocad Reference Manual.
PLOT	The PLOT command sends your drawing to your plotter or to a specified
PLOT	file. Chapter 13 of the Autocad Reference Manual fully documents plotting and
PLOT	PLOT and PRPLOT commands. Plotting can also be initiated from the Main
PLOT	Menu.
PLOT	
PLOT	Format: PLOT
PLOT	What to plot -- Display, Extents, Limits, View, or Window <D>:
PLOT	
PLOT	In order to plot to a file, you must first configure the target plotter,
PLOT	just as if you were going to send plot output directly to the plotter.
PLOT	During this configuration, you will be asked if you want to write the
PLOT	plot to a file and the plot file name.
PLOT	
PLOT	See also: Chapter 13 of the Autocad Reference Manual.
POINT	The POINT command permits you to place a Point entity in the drawing.

POINT	Points are useful as "nodes" for object snap purposes (ADE-2 feature).
POINT	
POINT	Format: POINT Point: (designate point)
POINT	
POINT	The appearance of Points in your drawing is governed by the PDMODE
POINT	system variable. A "slide" file is provided to illustrate the
POINT	various forms a point can take. To view it, enter "VSLIDE points".
POINT	
POINT	See also: Section 4.2 of the Autocad Reference Manual.
POINT	
POINT	For help on formats for entering points, use "HELP POINTS".
POINTS	You can enter points, or coordinates, in any of the following ways:
POINTS	
POINTS	Absolute: x,y
POINTS	Relative: @deltax,deltay
POINTS	Polar: @dist<angle
POINTS	
POINTS	Normally, distances, points, and angles are entered as decimal numbers,
POINTS	or in scientific notation. If you have the ADE=1 package, you can use
POINTS	the UNITS command to specify linear values in terms of feet and inches,
POINTS	or angles in terms of degrees/minutes/seconds, grads, radians, or
POINTS	surveyor's units. For the commands that accept 3D points, you can
POINTS	include a Z coordinate in the absolute and relative formats:
POINTS	
POINTS	Absolute: x,y,z
POINTS	Relative: @deltax,deltay,deltaz
POINTS	
POINTS	If you omit the Z coordinate, the current elevation is used.
POINTS	
POLYGON	The POLYGON command (ADE-3) allows you to draw regular polygons with
POLYGON	anywhere from 3 to 1024 sides. The size of the polygon may be specified
POLYGON	by the radius of a circle in which it is inscribed or about which it is
POLYGON	circumscribed, or by the length of an edge.
POLYGON	
POLYGON	Format: POLYGON Number of sides:
POLYGON	Edge/<Center of polygon>: (enter a point)
POLYGON	Inscribed in circle/Circumscribed about circle (I/C):
POLYGON	Radius of circle:
POLYGON	
POLYGON	If you reply with "Inscribed", you should then enter the radius of a
POLYGON	circle on which all the vertices of the polygon will lie. You may
POLYGON	enter the radius numerically, or pick a point relative to the center
POLYGON	of the polygon. If you pick a point, a vertex of the polygon will be
POLYGON	drawn at that point.
POLYGON	
POLYGON	If you reply with "Circumscribed", you should then enter the radius of
POLYGON	a circle on which the midpoint of each edge of the polygon will lie.
POLYGON	You may enter a number or pick a point relative to the center of the
POLYGON	polygon. If you pick a point, an edge midpoint will be drawn at that

POLYGON	point.
POLYGON	
PRPLOT	The PRPLOT command causes a hard copy of the drawing to be produced on a
PRPLOT	printer/plotter - a printer with graphics capability. It also has the
PRPLOT	option to send the print plot to a file for later printer plotting.
PRPLOT	Chapter 13 of the Autocad Reference Manual fully documents plotting and the PL
PRPLOT	PRPLOT commands. Printer plotting can also be initiated from the Main
PRPLOT	Menu.
PRPLOT	
PRPLOT	Format: PRPLOT
PRPLOT	What to plot -- Display, Extents, Limits, View, or Window <D>:
PRPLOT	
PRPLOT	In order to printer plot to a file, you must first configure the target
PRPLOT	printer/plotter just as if you were going to send printer plot output
PRPLOT	directly to the printer plotter. During this configuration, you will be
PRPLOT	asked if you want to write the plot to a file and the plot file name.
PRPLOT	
PRPLOT	See also: Chapter 13 of the Autocad Reference Manual.
PURGE	During the course of editing a drawing, you may define Blocks, layers,
PURGE	linetypes, Shape files, and Text styles that subsequently are left
PURGE	unused. The PURGE command allows you to discard these unused objects.
PURGE	
PURGE	Format: PURGE
PURGE	Purge unused Blocks/Layers/Ltypes/SHapes/STyles/All:
PURGE	
PURGE	Reply with the object type you want to purge. PURGE responds with the name
PURGE	of each such object that is unused, and asks whether you want to purge it.
PURGE	
PURGE	NOTE: PURGE only works if it is the first command you use after entering
PURGE	the Drawing Editor to edit an existing drawing.
PURGE	
PURGE	See also: Section 3.12 of the Autocad Reference Manual.
QTEXT	The QTEXT command governs "quick text" mode. If QTEXT mode is off
QTEXT	(the normal case), text items are fully drawn. If QTEXT mode is on,
QTEXT	only a rectangle is drawn enclosing the area of each text item.
QTEXT	
QTEXT	Format: QTEXT ON/OFF <current>:
QTEXT	
QTEXT	See also: Section 6.7 of the Autocad Reference Manual.
QUIT	The QUIT command exits from the Drawing Editor, discarding all updates
QUIT	to the current drawing, and returns you to the Main Menu. If you then
QUIT	wish to exit entirely, select item 0 from the Main Menu.
QUIT	
QUIT	Format: QUIT Really want to discard all changes to drawing?
QUIT	
QUIT	If you reply with anything other than "Y" or "YES", the QUIT command
QUIT	is ignored, and you can continue editing.
QUIT	
QUIT	See also: Section 3.2 of the Autocad Reference Manual.

REDEFINE	The UNDEFINE and REDEFINE commands (ADE-3) let you override
REDEFINE	AutoCAD's built-in commands with versions implemented in AutoLISP or via
REDEFINE	external programs listed in the ACAD.PGP file. For instance, to
REDEFINE	undefine AutoCAD's QUIT command, you would enter:
REDEFINE	
REDEFINE	Command: UNDEFINE Command name: QUIT
REDEFINE	
REDEFINE	and to redefine it, you would enter:
REDEFINE	
REDEFINE	Command: REDEFINE Command name: QUIT
REDEFINE	
REDEFINE	Even if a command is undefined, you can still use it if you precede the
REDEFINE	command name with a period, as in ".QUIT".
REDEFINE	
REDEFINE	See also: The Release 9 reference manual supplement.
REDO	If REDO is entered immediately after a command that undoes something
REDO	(U, UNDO Back, or UNDO nnn), it will undo the Undo. An UNDO after the
REDO	REDO will redo the original Undo.
REDO	
REDO	See also: Section 5.5 of the Autocad Reference Manual.
REDRAW	The REDRAW command causes the display screen to be redrawn,
REDRAW	eliminating any point entry "blips" from the display. Setting
REDRAW	BLIPMODE (q.v.) to OFF can suppress the drawing of "blips".
REDRAW	
REDRAW	Format: REDRAW
REDRAW	
REDRAW	See also: Section 6.4 of the Autocad Reference Manual.
REGEN	The REGEN command regenerates the entire drawing and redraws it on the
REGEN	screen.
REGEN	
REGEN	Format: REGEN
REGEN	
REGEN	See also: Section 6.5 of the Autocad Reference Manual.
REGENAUT	Some commands can change many entities at once. The drawing must be
REGENAUT	regenerated to reflect such a change, so some commands perform this
REGENAUT	regeneration automatically. The REGENAUTO command lets you control
REGENAUT	whether such automatic regens are performed.
REGENAUT	
REGENAUT	Format: REGENAUTO ON/OFF <current>:
REGENAUT	
REGENAUT	If REGENAUTO is OFF and a ZOOM or PAN needs to regenerate the drawing,
REGENAUT	you will be prompt:
REGENAUT	
REGENAUT	About to regen, proceed? <Y>
REGENAUT	
REGENAUT	A "No" response aborts the PAN or ZOOM.
REGENAUT	
REGENAUT	This message does not appear if input is coming from a menu item or a script.
REGENAUT	

REGENAUT	See also: Section 6.11 of the Autocad Reference Manual.
RENAME	The RENAME command lets you change the names of Blocks, layers, linetypes,
RENAME	Text styles, and Named Views in your drawing. (Named Views are an ADE-2
RENAME	feature.)
RENAME	
RENAME	Format: RENAME Block/Layer/Ltype/Style/View: (select one)
RENAME	Old (object) name: (old name)
RENAME	New (object) name: (new name)
RENAME	
RENAME	See also: Section 3.12 of the Autocad Reference Manual.
REPEAT	The REPEAT and ENDREP commands are no longer supported.
REPEAT	You can use the ARRAY and MINSERT commands to achieve the
REPEAT	same results.
REPEAT	
REPEAT	Old drawings containing REPEAT/ENDREP entities must be converted
REPEAT	via Main Menu task 8 before they can be edited by this version
REPEAT	of AutoCAD.
REPEAT	
REPEAT	See also: Sections 5.2 and 9.1 and Appendix E of the Autocad Reference Manua
RESUME	The RESUME command may be used to return to a command script
RESUME	that has been interrupted due to an error or keyboard input.
RESUME	
RESUME	Format: RESUME
RESUME	
RESUME	See also: Command scripts, Section 11.1 of the Autocad Reference Manual.
ROTATE	The ROTATE command (ADE-3) can be used to rotate existing entities.
ROTATE	Format: ROTATE Select objects: (Do so)
ROTATE	Base point: (point)
ROTATE	<Rotation angle>/Autocad Reference:
ROTATE	
ROTATE	If you respond to the last prompt with a numeric angle, this is taken as
ROTATE	a relative angle (number of degrees) by which the selected objects will be
ROTATE	rotated from their current orientation, around the specified base point.
ROTATE	A positive angle causes counterclockwise rotation, and a negative angle
ROTATE	produces clockwise rotation.
ROTATE	
ROTATE	If you respond to the last prompt with "Autocad Reference", you can specify th
ROTATE	current rotation and the new rotation you desire. AutoCAD prompts:
ROTATE	
ROTATE	Rotation angle <0>:
ROTATE	New angle:
ROTATE	
ROTATE	You can even "show" AutoCAD the reference angle (by pointing to the two
ROTATE	endpoints of a line to be rotated), and then specify the new angle. You
ROTATE	can specify the new angle by pointing or by dragging the object.
ROTATE	
ROTATE	See also: Section 5.2 of the Autocad Reference Manual.
RSCRIPT	If a script file has been invoked using the SCRIPT command from the
RSCRIPT	Drawing Editor, an RSCRIPT command encountered in the script file causes

RSCRIPT	the script to be restarted from the beginning.
RSCRIPT	
RSCRIPT	Format: RSCRIPT
RSCRIPT	
RSCRIPT	See also: Section 11.1 of the Autocad Reference Manual.
SAVE	The SAVE command allows you to update your drawing on disk periodically
SAVE	without exiting the Drawing Editor.
SAVE	
SAVE	Format: SAVE File name: (name or RETURN)
SAVE	
SAVE	The current drawing file is the default output file, but you can specify
SAVE	another file name explicitly. Do not include a file type; ".dwg" is assumed.
SAVE	
SAVE	See also: Section 3.3 of the Autocad Reference Manual.
SCALE	The SCALE command (ADE-3) lets you change the size of existing entities.
SCALE	The same scale factor is applied to X and Y dimensions.
SCALE	Format: SCALE Select objects: (Do so)
SCALE	Base point: (point)
SCALE	<Scale factor>/Autocad Reference:
SCALE	
SCALE	If you respond to the last prompt with a number, this is taken as a
SCALE	relative scale factor by which all dimensions of the selected objects
SCALE	will be multiplied. To enlarge an object, enter a scale factor greater
SCALE	than 1. To shrink an object, use a scale factor between 0 and 1.
SCALE	
SCALE	If you respond to the last prompt with "Autocad Reference", you can specify th
SCALE	current length and the new length you desire. AutoCAD prompts:
SCALE	
SCALE	Autocad Reference length <1>:
SCALE	New length:
SCALE	
SCALE	You can "show" AutoCAD the reference length (by pointing to the two endpoints
SCALE	of a line to be scaled), and then specify the new length. You can specify
SCALE	the new length by pointing, or by dragging the object.
SCALE	
SCALE	See also: Section 5.2 of the Autocad Reference Manual.
SCRIPT	The SCRIPT command causes commands to be read from the specified
SCRIPT	script file.
SCRIPT	
SCRIPT	Format: SCRIPT Script file: (name)
SCRIPT	
SCRIPT	Commands are read from the script file until the end of the file
SCRIPT	is reached, a character (preferably Backspace) is entered from the
SCRIPT	keyboard, or a command error occurs. If the script is terminated
SCRIPT	early due to a command error or by keyboard entry, it may be resumed
SCRIPT	using the RESUME command.
SCRIPT	
SCRIPT	The RSCRIPT command can be inserted in the script file to restart the
SCRIPT	script from the beginning.

SCRIPT	
SCRIPT	See also: Section 11.1 of the Autocad Reference Manual.
SELECT	The SELECT command lets you designate a group of objects
SELECT	as the current selection-set. This group can be referenced
SELECT	as the "Previous" selection-set in subsequent commands.
SELECT	
SELECT	Format: SELECT Select objects: (do so)
SELECT	
SELECT	See also: Chapter 5 of the Autocad Reference Manual.
SETVAR	Many AutoCAD commands set various modes, sizes, and limits that then remain
SETVAR	in effect until you change them. AutoCAD remembers these values by storing
SETVAR	them in a collection of "system variables". The SETVAR command allows you
SETVAR	to examine and change these variables directly.
SETVAR	
SETVAR	Format: SETVAR Variable name or ?:
SETVAR	
SETVAR	If you answer with "?", AutoCAD flips to the text screen and displays the
SETVAR	names and current values of all system variables. Some system variables
SETVAR	cannot be changed; these will be flagged in the output by the legend
SETVAR	"(read only)" following the value. If you enter the name of a variable
SETVAR	that is not read-only, you will receive the prompt:
SETVAR	
SETVAR	New value for varname <current>:
SETVAR	
SETVAR	where "varname" is replaced by the variable name, and "current" is the
SETVAR	current value of the variable. If you respond to this prompt by
SETVAR	pressing RETURN or CTRL C, the variable will be left unchanged.
SETVAR	
SETVAR	See also: Section 3.10 of the Autocad Reference Manual.
SH	The ADE-3 package's SHELL command allows you to execute utility
SH	programs or user-supplied programs while still running AutoCAD.
SH	The SH command is similar, but allows only internal DOS commands
SH	to be executed.
SH	
SH	Format: SHELL
SH	DOS command: (enter desired program name, or RETURN)
SH	
SH	When the utility program is done, you can enter another AutoCAD command.
SH	
SH	If you reply to the "DOS command:" prompt with RETURN, a prompt such
SH	as "C>>" (a normal DOS prompt with an extra ">" appended) appears. You
SH	can now enter multiple DOS commands, just as you would at the normal
SH	DOS prompt. To return to AutoCAD from this mode, enter "EXIT".
SH	
SH	NOTE: There are some restrictions on the programs you can run from AutoCAD.
SH	
SH	See also: Section 3.11 of the Autocad Reference Manual.
SHAPE	The SHAPE command inserts a defined shape into the drawing, provided that
SHAPE	the shape definitions have been loaded using the LOAD command.

SHAPE	
SHAPE	Format: SHAPE Shape name (or ?) <default>: (shape name)
SHAPE	Starting point: (point)
SHAPE	Height <1.0>: (value)
SHAPE	Angle <0>: (angle)
SHAPE	
SHAPE	If you reply to the first prompt with "?", AutoCAD will list the
SHAPE	names of all Shapes currently loaded in the drawing.
SHAPE	
SHAPE	See also: Section 4.11 of the Autocad Reference Manual.
SKETCH	The SKETCH command allows you to do freehand drawings. This is an ADE-1
SKETCH	feature, and requires a pointing device such as a digitizing tablet or
SKETCH	mouse.
SKETCH	
SKETCH	Format: SKETCH Record increment: (value)
SKETCH	Sketch. Pen eXit Quit Record Erase Connect .
SKETCH	
SKETCH	Subcommands:
SKETCH	
SKETCH	P - Raise/lower sketching pen
SKETCH	X - Record temporary lines, and exit Sketch
SKETCH	Q - Discard temporary lines, and exit Sketch
SKETCH	R - Record temporary lines, but remain in Sketch
SKETCH	E - Erase temporary lines from a specified point to the end
SKETCH	C - Connect: restart sketch at last end point
SKETCH	. - Draw line from end to current point (pen up)
SKETCH	
SKETCH	See also: Section 12.5 of the Autocad Reference Manual.
SNAP	The "snap resolution" is the spacing of an imaginary grid of dots with which
SNAP	newly designated points must align. The SNAP command allows you to change
SNAP	the snap resolution or to turn it off entirely for free-style drawing.
SNAP	
SNAP	Format: SNAP Snap spacing or ON/OFF/Aspect?Rotate/Style <current>:
SNAP	
SNAP	The meaning of each option is described below.
SNAP	
SNAP	NUMBER - Set alignment spacing
SNAP	ON - Align designated points
SNAP	OFF - Do not align designated points
SNAP	ROTATE - Rotate snap grid by specified angle, and
SNAP	set a specified base point for the grid
SNAP	ASPECT - Set different X/Y snap resolution
SNAP	STYLE ISO - Set isometric snap style
SNAP	STYLE STANDARD - Set normal snap style
SNAP	
SNAP	(The ROTATE, ASPECT, and STYLE options are ADE-2 features.)
SNAP	
SNAP	See also: Section 8.1 of the Autocad Reference Manual.
SOLID	The SOLID command allows you to draw solid filled regions by

SOLID	entering them as quadrilateral or triangular sections.
SOLID	
SOLID	Format: SOLID First point: (point)
SOLID	Second point: (point)
SOLID	Third point: (point)
SOLID	Fourth point: (point, or RETURN for triangular section)
SOLID	Third point: (point, or RETURN to end solid)
SOLID	
SOLID	See also: Section 4.7 of the Autocad Reference Manual.
STATUS	The STATUS command produces a report describing the current drawing
STATUS	extents and the current settings of various drawing modes and parameters.
STATUS	
STATUS	Format: STATUS
STATUS	
STATUS	NOTE: In dimensioning mode (ADE-1 feature), the STATUS command lists
STATUS	the dimensioning variables and their current values.
STATUS	
STATUS	See also: Section 3.4 of the Autocad Reference Manual.
STRETCH	The STRETCH command (ADE-3) allows you to move a selected portion of a
STRETCH	drawing, preserving connections to parts of the drawing left in place.
STRETCH	Connections made with lines, arcs, traces, solids, polylines, 3D lines,
STRETCH	and 3D faces may be STRETCHed.
STRETCH	
STRETCH	Format: STRETCH Select objects to stretch by window...
STRETCH	Select objects:
STRETCH	
STRETCH	While you may use any of AutoCAD's forms of object selection in the
STRETCH	STRETCH command, you must use a window-style selection (either Crosses
STRETCH	or Window) at least once. The last window specified will be the window
STRETCH	moved by STRETCH. Objects may be freely added and removed from the
STRETCH	selection set.
STRETCH	
STYLE	The STYLE command lets you create new Text styles and modify existing
STYLE	ones. Each Text style uses a particular font, to which you can apply
STYLE	a fixed height, an expansion/compression width factor, and an obliquing
STYLE	(slant) angle. You can also select backwards (mirrored right to left)
STYLE	or upside-down (mirrored top to bottom) text generation.
STYLE	
STYLE	Format: STYLE Text style name (or ?): (name)
STYLE	Font file <default>: (file name)
STYLE	Height <default>: (value)
STYLE	Width factor <default>: (scale factor)
STYLE	Obliquing angle <default>: (angle)
STYLE	Backwards? <Y/N>
STYLE	Upside-down? <Y/N>
STYLE	Vertical? <Y/N>
STYLE	(name) is now the current text style.
STYLE	
STYLE	The style you create or modify becomes the current text style used

STYLE	for newly drawn Text entities.
STYLE	
STYLE	See also: Section 4.10 of the Autocad Reference Manual.
TABLET	The TABLET command is used when an existing hard copy drawing is to be
TABLET	"copied" with a digitizing tablet. You can also use the TABLET command to
TABLET	designate tablet menu areas and the portion of the tablet to be used as the
TABLET	screen pointing area.
TABLET	
TABLET	Formats: TABLET ON - Turn tablet mode on
TABLET	TABLET OFF - Turn tablet mode off
TABLET	TABLET CAL - Calibrate tablet to existing drawing
TABLET	TABLET CFG - Configure tablet menus and screen pointing area
TABLET	
TABLET	See also: Section 12.4 of the Autocad Reference Manual.
TEXT	The TEXT command draws text of any desired size and angle.
TEXT	
TEXT	Format: TEXT Start point or Align/Center/Fit/Mid/Right/Style: (point)
TEXT	Height <default>: (value or two points)
TEXT	Angle <default>: (angle or point)
TEXT	Text: (text string to be drawn)
TEXT	
TEXT	If you enter a point for the "Starting point", the text is drawn
TEXT	left-justified at that point. Alternatively, you can reply:
TEXT	
TEXT	A - To align the text between two designated end points.
TEXT	Height and Angle are not requested in this case.
TEXT	C - To center the text around a specified point.
TEXT	F - To align the text between two designated end points with
TEXT	a specified height that varies only in its X scale factor.
TEXT	M - To center text both horizontally and vertically around a
TEXT	specified point.
TEXT	R - To right-justify the text at a designated end point.
TEXT	S - To select a different Text style.
TEXT	
TIME	When you enter the TIME command, the current status of AutoCAD's time
TIME	variables is displayed, as shown below.
TIME	
TIME	Command: TIME
TIME	
TIME	Current time: 08 NOV 1985 at 09:10:44.005
TIME	Drawing created: 23 JUL 1985 at 07:21:30.648
TIME	Drawing last updated: 18 SEP 1985 at 15:33:59.771
TIME	Time in drawing editor: 0 days 00:02:54.520
TIME	Elapsed timer: 0 days 00:00:30.772
TIME	Timer on.
TIME	
TIME	All times are displayed to the nearest millisecond using 24-hour "military"
TIME	format, where 15:31:00 means 3:31 in the afternoon. The TIME command
TIME	next prompts:

TIME	
TRACE	The TRACE command allows you to draw traces (solid-filled lines
TRACE	of specified width).
TRACE	
TRACE	Format: TRACE From point: (point)
TRACE	To point: (point)
TRACE	To point: (point)
TRACE	To point: (RETURN to end trace entry)
TRACE	
TRACE	Traces may be constrained to horizontal or vertical by the ORTHO command.
TRACE	
TRACE	See also: Section 4.5 of the Autocad Reference Manual.
TRIM	The TRIM command (ADE-3) allows you to trim objects in a drawing so they
TRIM	end precisely at a "cutting edge" defined by one or more other objects
TRIM	in the drawing.
TRIM	
TRIM	Format: TRIM Select cutting edges(s)...
TRIM	Select objects:
TRIM	
TRIM	Lines, Arcs, Circles, and Polylines (center line of Polyline) may serve
TRIM	as boundary objects. All the selected edges are highlighted and will
TRIM	remain highlighted for the rest of the TRIM command. Next the prompt:
TRIM	
TRIM	Select object to TRIM:
TRIM	
TRIM	appears. Select the objects to be trimmed at the previously selected
TRIM	cutting edges by pointing to the part of the object to be trimmed.
TRIM	Answer with RETURN to end the command.
TRIM	
TRIM	If the selected point is between two intersections, the entity will be deleted
TRIM	between the two intersection points. Polylines are trimmed at their center
TRIM	line.
TRIM	
TRIM	See also: Section 5.3 of the Autocad Reference Manual.
U	The U command causes the most recent operation to be undone. The name
U	of the command being undone will be displayed. You can enter the U
U	command as many times as you wish, backing up one step at a time, until
U	the drawing is in its original state.
U	
U	See also: Section 5.5 of the Autocad Reference Manual.
UNDO	The UNDO command allows you to undo several commands at once and to perform
UNDO	several special operations, such as marking a point to which you want to
UNDO	return if things go wrong. When you enter UNDO, you get the prompt:
UNDO	
UNDO	Format: UNDO Auto/Back/Control/End/Group/Mark/<Number>:
UNDO	
UNDO	The default response is just to enter a number; this number of preceding
UNDO	operations will be undone.
UNDO	

UNDO	Mark - The Mark subcommand makes a special mark in the undo information,
UNDO	to which you can later back up with the Back subcommand.
UNDO	
UNDO	Group - The Group and End subcommands cause a group of commands to be
UNDO	End treated as a single command for the purposes of U and UNDO.
UNDO	A Group, once Ended, is always treated as a single, indivisible
UNDO	operation.
UNDO	
UNDO	Auto - The Auto subcommand requires an additional specification of ON or
UNDO	OFF. When UNDO Auto is ON, any operation taken from the menu, no
UNDO	matter how complicated, will be treated as a single command,
UNDO	reversible by a single U command.
UNDO	
UNITS	The UNITS command governs the display and input formats for coordinates,
UNITS	distances, and angles. UNITS is an ADE-1 feature.
UNITS	
UNITS	Format: UNITS
UNITS	
UNITS	You can then select one of the following display/input formats for
UNITS	coordinates and distances:
UNITS	
UNITS	Scientific 1.55E+01 (15.5 drawing units)
UNITS	Decimal 15.5000 "
UNITS	Engineering 1'-3.5" "
UNITS	Architectural 1'-3 1/2" "
UNITS	Fractional 15 1/2 "
UNITS	
UNITS	You can also specify the precision (the number of digits after the decimal
UNITS	point, or the smallest fraction of an inch to display).
UNITS	
VIEW	The VIEW command can be used to associate a name with the current view
VIEW	of the drawing, and to retrieve such named views. This is an ADE-2 feature.
VIEW	
VIEW	Format: VIEW ?/Delete/Restore/Save/Window: (select one)
VIEW	View name: (name)
VIEW	
VIEW	? - List the named views for this drawing
VIEW	Delete - Delete the named view
VIEW	Restore - Display the specified view
VIEW	Save - Attach "name" to current view of drawing
VIEW	Window - Attach "name" to specified window
VIEW	
VIEW	See also: Section 6.3 of the Autocad Reference Manual.
VIEWRES	The VIEWRES command controls "fast zoom" mode and sets the resolution
VIEWRES	for circle and arc generation.
VIEWRES	
VIEWRES	Format: VIEWRES Do you want fast zooms? <Y>
VIEWRES	Enter circle zoom percent (1-20000) <100>:
VIEWRES	

VIEWRES	If you respond to the first prompt with "N", all ZOOMs, PANs, and "VIEW
VIEWRES	Restores" will perform regenerations, and thus run at the same speed as in
VIEWRES	previous versions of AutoCAD. If you respond "Y", AutoCAD will maintain a
VIEWRES	large virtual screen for your drawing, and will perform ZOOMs, PANs, and
VIEWRES	"VIEW Restores" at REDRAW speed whenever possible.
VIEWRES	
VPOINT	The VPOINT command is part of 3D Level 1(tm), and is included in
VPOINT	the ADE-3 package. It selects a 3D view point, and regenerates the
VPOINT	drawing as if you were viewing it from that point. All entities are
VPOINT	drawn with the correct elevation and thickness, and projected as you
VPOINT	would see them from the specified view point.
VPOINT	
VPOINT	Format: VPOINT
VPOINT	Rotate/<View point> <current X, Y, Z>:
VPOINT	
VPOINT	If you enter RETURN in response to the "Enter view point" prompt, a
VPOINT	compass and axes tripod will be displayed to assist you in selecting a
VPOINT	view point. A view point of 0,0,1 will return you to the normal 2D
VPOINT	"top" or "plan" view. You can also use either "VIEW RESTORE" or
VPOINT	"ZOOM PREVIOUS" to restore a prior view to the screen.
VPOINT	
VPOINT	The "Rotate" option lets you specify the new viewpoint in terms of two
VPOINT	angles; one with respect to the X axis (in the XY plane) and another
VPOINT	with respect to the Z axis.
VPOINT	
VPOINT	See also: Section 14.2 of the Autocad Reference Manual, and the Release 9 su
VSLIDE	The VSLIDE command permits viewing of a slide made by the MSLIDE command,
VSLIDE	or of a slide from a library constructed by the SLIDELIB utility program.
VSLIDE	
VSLIDE	Formats: VSLIDE Slide file: name
VSLIDE	VSLIDE Slide file: libraryname(slidename)
VSLIDE	
VSLIDE	The current display is replaced by the named slide. To retrieve the
VSLIDE	display of the current drawing, use the REDRAW command.
VSLIDE	
VSLIDE	If you are using a command script to display a series of slides in
VSLIDE	succession, you can overlap the time necessary to load the next slide
VSLIDE	from disk with the viewing time of the current slide. This is done by
VSLIDE	preceding the file name with an asterisk.
VSLIDE	
VSLIDE	Format: VSLIDE Slide file: *name
VSLIDE	
VSLIDE	The named slide is read from disk into memory, but is not displayed
VSLIDE	until the next VSLIDE command is executed.
VSLIDE	
VSLIDE	See also: Section 11.2 of the Autocad Reference Manual, and the Release 9 su
WBLOCK	The WBLOCK command writes all or part of a drawing out to a disk file.
WBLOCK	
WBLOCK	Format: WBLOCK File name: (output file name)

WBLOCK	Block name: (see below)
WBLOCK	
WBLOCK	No file type should be specified; type ".dwg" is assumed.
WBLOCK	The different responses to the "Block name" prompt are:
WBLOCK	
WBLOCK	name - The named Block is written to the disk file.
WBLOCK	= - Same as above, but the Block name is the same
WBLOCK	as the file name.
WBLOCK	* - The entire drawing is written to disk, except for
WBLOCK	unreferenced Block Definitions.
WBLOCK	(blank) - Permits selection of individual objects to be written
WBLOCK	to disk. Also requests an insertion base point.
WBLOCK	
WBLOCK	See also: Section 9.1 of the Autocad Reference Manual.
ZOOM	The ZOOM command magnifies the drawing on the display screen (to see more
ZOOM	detail) or shrinks it (to view more or the drawing with less detail).
ZOOM	
ZOOM	ZOOM number - Magnification relative to ZOOM All display
ZOOM	(ZOOM All = ZOOM 1). Higher numbers (like 2.5)
ZOOM	magnify, lower numbers (like 0.5) shrink.
ZOOM	ZOOM numberX - Magnification relative to current display (1X).
ZOOM	ZOOM All - Place entire drawing (all visible layers) on
ZOOM	display at once.
ZOOM	ZOOM Center - Specify center point and new display height.
ZOOM	ZOOM Dynamic - Permits you to pan a box representing the viewing screen
ZOOM	around the entire generated portion of the drawing and
ZOOM	enlarge or shrink it in a dynamic, graphic manner.
ZOOM	ZOOM Extents - Displays current drawing content as large as possible.
ZOOM	ZOOM Left - Specify lower left corner and new display height.
ZOOM	ZOOM Previous - Restores previous view.
ZOOM	ZOOM Window - Designate rectangular area to be drawn as large
ZOOM	as possible.
ZOOM	
ZOOM	See also: Section 6.1 of the Autocad Reference Manual.
DIR	DIR allows you to show a directory of user specified files.
DIR	
DIR	
DIR	Format: DIR
DIR	File specification: (Enter files to show) - ie *.DWG
TYPE	TYPE allows you to show the ASCI (text, contents, etc.) of a specified DOS
TYPE	file.
TYPE	
TYPE	
TYPE	
TYPE	File to list: (Enter file name) ie SOME.TXT
EDIT	EDIT allows you to edit a DOS file with the DOS program EDLIN, for help with
EDIT	EDLIN see your DOS Reference Manual.
EDIT	

EDIT	
EDIT	Format: EDIT
EDIT	File to edit: (Enter file name) ie SOME.TXT
WRITE	*TEXT* allows you to enter any combination of characters from the keyboard
WRITE	directly into the command string. As a result, whatever you type will be
WRITE	executed by Autocad without adding any additional programming commands.
WRITE	
WRITE	
WRITE	
WRITE	NOTE that "*TEXT*" is NOT an Autocad command it is a Manumaker function.
COMMAND	This allows you to execute a command from inside your new menu. The command
COMMAND	must be one that will be recognized within your drawing when used. An example
COMMAND	might be the command to execute a LISP program.
COMMAND	
COMMAND	NOTE that "COMMAMD" is NOT an Autocad command it is a Menumaker function.
RETURN	This allows you to place a carriage return at the end of the string you are
RETURN	working with. Normally this would be appended to a string at the end where,
RETURN	because of your menu requirements, an additional <RETURN> is required.
RETURN	
RETURN	NOTE that "RETURN" is NOT an Autocad command it is a Menumaker function.
LISP	*LISP* allows you to load an Autolisp program and then run that program.
LISP	
LISP	NOTE that "*LISP*" is not an Autocad command it is a Menumaker function.
?	Autocad commands (Realese 9 ADE +3)
?	3DFACE DDATTE FILES MEASURE REDEFINE STYLE
?	3DLINE DDEMODES FILL MENU REDO TABLET
?	APERTURE DDLMODES FILLET MINSERT REDRAW TEXT
?	ARC DDRMODES FILMROLL MIRROR REGEN TEXTSCR
?	AREA DELAY GRAPHSCR MOVE REGENAUTO TIME
?	ARRAY DIM/DIM1 GRID MSLIDE RENAME TRACE
?	ATTDEF DIST HATCH MULTIPLE RESUME TRIM
?	ATTDISP DIVIDE HELP/? OFFSET ROTATE U
?	ATTEDIT DONUT HIDE OOPS RSCRIPT UNDEFINE
?	ATTEXT DOUGHNUT ID ORTHO SAVE UNDO
?	AXIS DRAGMODE IGESIN OSNAP SCALE UNITS
?	BASE DTEXT IGESOUT PAN SCRIPT VIEW
?	BLIPMODE DXBIN INSERT PEDIT SELECT VIEWRES
?	BLOCK DXFIN ISOPLANE PLINE SETVAR VPOINT
?	BREAK DXFOUT LAYER PLOT SHAPE VSLIDE
?	CHAMFER ELEV LIMITS POINT SHELL/SH WBLOCK
?	CHANGE ELLIPSE LINE POLYGON SKETCH ZOOM
?	CIRCLE END LINETYPE PRPLOT SNAP
?	COLOR ERASE LIST PURGE SOLID
?	COPY EXPLODE LOAD QTEXT STATUS
?	DBLIST EXTEND LTSCALE QUIT STRETCH
SUBMENU	
SUBMENU	This allows you to change from one submenu to another. If you select SCREEN
SUBMENU	the menu will change the submenu on the right side of your drawing editer. If
SUBMENU	you select PULL-DOWN the menu will change the Pull-Down submenus. (NOTE: Pull-

SUBMENU	Down windows require Autocad Release 9)
SUBMENU	
SUBMENU	NOTE: "-SUBMENU-" is Not an Autocad command it is a Menumaker function.