COMMAND OPTION	ICON	DESCRIPTION
Zoom All		This option causes AutoCAD to display the whole drawing as far as its drawing limits or drawing extents (whichever is the greater of the two).
Zoom Center		This option requires two things: a point that is to be the center of the new display and a value to be its new height *in drawing units*. The existing height is the default for the new height to allow for panning across the drawing. If the new height value is followed by "X" (eg. 2x), then it is taken as a magnification factor relative to the current height. If followed by "XP", then it is taken as a scale factor relative to paper space and can be used for scaling the contents of paper space viewports.
Zoom Dynamic		This is a very useful ZOOM option once it is understood. It permits very quick movement around the drawing. Once selected, this option redraws the graphics area of the screen and displays two rectangles. The larger box shows the extents of the current drawing. The smaller box shows the current view with an "X" in the middle. This moves with the mouse. This view box should be positioned so that its lower left corner is at the lower left corner of the view required. By pressing the left button on the mouse, the "X" is replaced by an "> " pointing to the right side of the view box. This allows you to change the magnification. As the mouse is moved, the view box shrinks and expands so that the size of the required view can be set. The left mouse button toggles between PAN "X" and ZOOM "> " mode so that fine adjustments can be achieved. When the view required has been selected, press <ENTER> or right click to cause AutoCAD to display it.
Zoom Extents		This option will display all the graphics that are contained in the drawing (referred to as the *drawing extents*) with the largest image possible.
		This option restores the displayed view

Zoom Previous		prior to the current one. For the purpose of this option, up to 10 views are saved so that the last ten views can be recalled. This option includes every time you use the scroll bar, which is one reason to **avoid the scroll bars** for panning a lot in your drawing.
Zoom Scale		This is a 'hidden' default option. You do not have to type "S" to choose this option. It simply requires the entry of a number that represents a magnification factor. Note that the factor is applied to the entire drawing (as defined by the drawing's limits). Numbers less than 1 will reduce the displayed size of the drawing, while numbers greater than 1 will enlarge it. If "X" is inserted after the number (eg. 0.8x) then the factor is applied *to the current view*. If "XP" is inserted after the scale factor, then the view is scaled relative to paper space. This is useful for zooming a view within a paper space viewport to a specific scale, for example, "1/48XP" will produce a view of model space at a scale of ¼" = 1' relative to paper space.
Zoom Window		This option (also a 'hidden' default) prompts the user to pick two corners of a box on the existing view in order to enlarge that area to fill the display.
Zoom Realtime		Zoom Realtime provides interactive zooming capability. Pressing <ENTER> (after entering zoom) on the command line automatically places you in Realtime mode. Hold the left mouse button down at the midpoint of the drawing and move the cursor vertically to the top (positive direction) of the window to zoom in up to 100% (2x magnification). Hold the left mouse button down at the midpoint of the drawing and move the cursor vertically to the bottom (negative direction) of the window to zoom out to100% (.5x magnification). *You cannot zoom out beyond the extents of the current view.*

		When you release the pick button, zooming stops. You can release the pick button, move the cursor to another location in the drawing, and then press the pick button again and continue zooming from that location. To exit Realtime Zoom mode, press <ENTER> or (ESC).
Aerial View command: **DSVIEWER**	None	Aerial View is a zooming tool that displays a view of the drawing in a separate window so that you can quickly move to that area. If you keep the Aerial View window open as you work, you can zoom and pan without choosing a menu option or entering a command. You can change the view by creating a new view box in the Aerial View window. To zoom in to the drawing, make the view box smaller by left clicking a rectangle. To zoom out of the drawing, make the view box larger. As you zoom in or out of the drawing, a real-time view of the current zoom location is displayed in the graphics area. The screenshot shows how the view box looks. Right click in the box and you can move the box to where you want to zoom to.
Zoom		This option asks you to select an object or objects, then press <ENTER> and the screen will zoom to those objects only. This is great for when you want to work on object.
Zoom In		Clicking this icon will zoom in to the drawing by about 50%. This option is only available as an icon and cannot be invoked by the command line.
Zoom Out		Similar to 'Zoom In' - this icon will zoom out of your drawing and allow you to see about 50% more of your drawing space.
Mouse Scroll	-	If you have a scrolling wheel on your mouse, you can use it to zoom in and out of your drawing. Scroll towards you to zoom out and away from you to zoom in. You have the option to change the amount of zoom per wheel click with the Zoomfactor system variable. Keep in mind that you will zoom in and out using your mouse location as a 'centre point'.

PAN		Panning allows you to quickly move around the drawing area at the same magnification you currently have set. Type in **PAN** (or **P**) <ENTER> and a hand will appear on the screen. Left click and hold to move around your drawing.

Command	Keystroke	Icon
Block	**Bmake / B**	
Write Block	**Wblock / W**	None
Insert	**Insert / I**	

Define an attribute	**ATTDEF / ATT**	
Edit attributes	**DDATTE / ATE**	 <
Block	**Block / Bmake / B**	
Display Atts.	**ATTDISP**	None
Extract Attributes	**EATTEXT**	
Polyline	**Pline / PL**	
Polyline Edit	**Pedit / PE**	
		

Boundary Hatch	**Bhatch / H**	
Hatch Edit	**HatchEdit / HE**	
BOX	**BOX**	
SPHERE	**SPHERE**	
CYLINDER	**CYLINDER**	
CONE	**CONE**	
WEDGE	**WEDGE**	
TORUS	**TORUS**	
POLYSOLID	**PSOLID**	

Menu	Result
Draw > Block > Make	Creates a block from separate entities (internal to current drawing)
None	Creates a block and writes it to a file (external)
Insert > Block	Inserts a block (internal or external)

Draw > Block > Define Attribute	Creates an attribute definition
Modify> Object> Attrb.> Single	Edits the contents of an existing attribute
Draw > Block > Make	Creates a block from separate entities and attributes.
None	Hides or shows attributes
Tools > Attribute Extraction...	Extracts attributes using the wizard
Draw > Polyline	Creates a polyline of arcs and/or lines.
Modify > Polyline	Edits polyline objects
	Covers an area with

Draw > Hatch	a predefined pattern
Modify > Object > Hatch...	Edits an existing Hatch
Creates a solid box after you provide 2 opposite corners.	
Creates a solid sphere from a center point and radius.	
Creates a straight cylinder from a center point, radius and height.	
Creates a tapered cone from a center point, radius and height.	
Creates a triangular wedge from 2 opposite points.	
Creates a torus (donut shape) based on center point, radius and tube radius.	
Draws a solid object with width and height as would draw a polyline.	