

MGA

Matrox Graphics Architecture



CAD Drivers

User Guide

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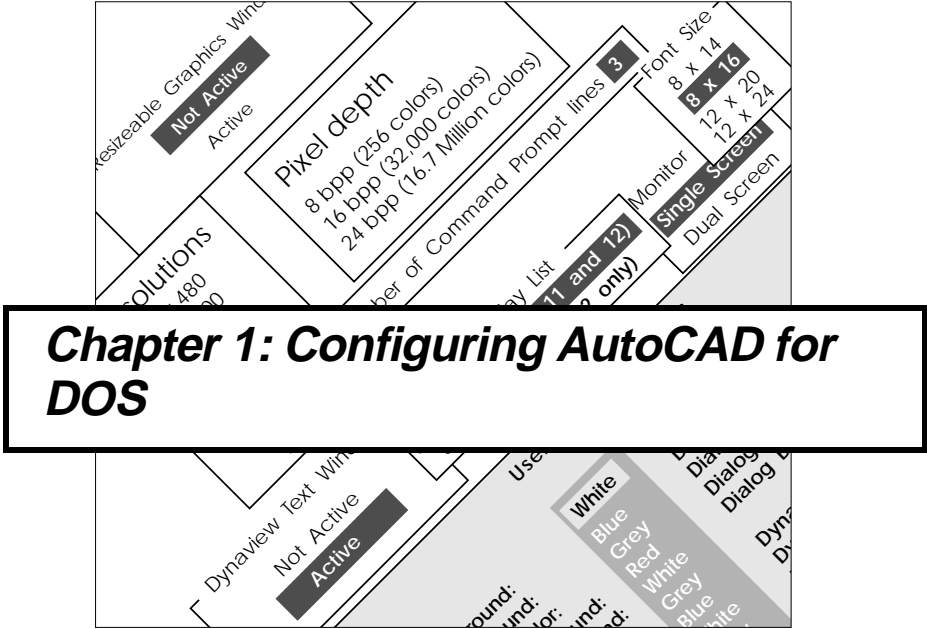
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Overview

This manual covers the MGA CAD drivers which include:

- AutoCAD for DOS driver;
supports AutoCAD 11, 12, 13 and 3D Studio
- AutoCAD for Windows driver;
supports AutoCAD 12 and 13 for Window 3.1
- MicroStation DOS driver;
supports MicroStation PC. 5.0 and 95

These drivers are included on your MGA CD-ROM. Refer to the *Installation Guide* for the installation of these drivers as well as for customer support information, BBS information, etc.



Chapter 1: Configuring AutoCAD for DOS

AutoCAD for DOS driver configuration

After you have installed the MGA driver for AutoCAD for DOS, you must tell AutoCAD to use it. At this point, you will have the opportunity to go through the MGA driver configuration utility.

First time AutoCAD 12 or 13 configuration

1. If you have never run AutoCAD on your system before, simply start it now. AutoCAD will detect that it is not configured, and will force you to configure all devices, including the Video Display.
2. When you get to Video Display, AutoCAD will show you a list of available drivers. Select:

Matrox MGA, version 1.10 (or higher)
3. The Matrox configuration utility will then start automatically.

Driver not listed?

If you do not see the Matrox driver listed, first verify that the AutoCAD environment variable, *ACADDRV*, is set.

AutoCAD 12 or 13 reconfiguration

Release 12 or 13 users who have already configured AutoCAD can go quickly to the Video Display configuration using the following procedure:

1. Start AutoCAD by typing: *acad -r*
2. You are presented with a list showing your present configuration. Press Enter.
3. At this point, you get the configuration menu; select:
 3. *Configure Video Display*
4. From the list that appears, select:
 - Matrox MGA, version 1.10* (or higher)
5. The Matrox configuration utility will start automatically. The following section describes the configuration process in detail.

AutoCAD 11 configuration

From the AutoCAD 11 Video Display configuration menu, select:

ADI P386 v4.014.1 display

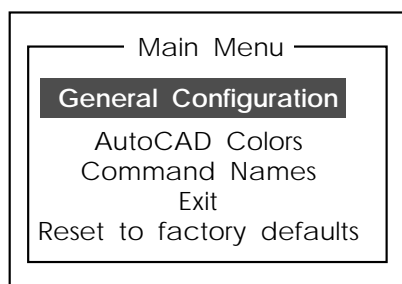
The Matrox configuration utility will start automatically.

Matrox driver configuration utility

The Matrox driver configuration utility is a set of menu-driven screens which will let you select your mode (resolution, pixel depth, 3D), and configure a number of driver parameters. The configuration utility creates a file called *acadrv.cfg*, in the directory pointed to by the Matrox *ACADR*V environment variable (normally *\mga\acadrv*). This *ACADR*V environment variable is set by the *acadrv.bat* batch file. On subsequent AutoCAD sessions, the driver will retrieve the *acadrv.cfg* file, and use the settings stored there.

Even though the driver comes with hard-coded defaults that you may want to retain, we recommend that you go through the configuration at least once, to see the available parameters.

General Configuration screens



Mode Selection

This step lets you select the resolution and number of colors (bpp is short for *bits per pixel*) that your MGA board will use for AutoCAD.

Depending on the board type and memory you have, the driver will automatically list the available modes in a window with scroll bars similar to the example on the next page.

Select a 3D mode only if you plan to use Matrox DynaView /3D features (the resolutions available with 3D are limited).

Also note that many modes list a ‘Smooth’ feature (this refers to ‘Smooth animation’). Smooth animation, which uses ‘double buffering’, makes DynaView /2D and /3D commands look much smoother. When Smooth is combined with 3D, the resolutions available may be limited (they both use the on-board RAM).

If you don’t plan to use the DynaView /2D Bird’s Eye and Spy Glass, or DynaView /3D’s MATDVIEW and MATDWALK commands, it’s not necessary to select a mode with the ‘Smooth’ feature.

Mode Selection			
800 x 600	24 bpp	3D	
1024 x 768	8 bpp	2D, Smooth	
1024 x 768	8 bpp	3D, Smooth	
1024 x 768	16 bpp	2D, Smooth	
1024 x 768	16 bpp	3D,	
1024 x 768	24 bpp	2D,	
1152 x 864	8 bpp	2D, Smooth	
1152 x 864	8 bpp	3D, Smooth	

Color Map

8-bit driver This screen applies only when you select a 256 color (8bpp), 2D mode. It allows you to work with either 16 or 256 colors within AutoCAD.

256 colors If you stay with the default 256 color map, the colors over 15 will change while you use any animation commands (see page 3-18 for more details). It is probably desirable to stay with the 256 color map if you use colors over index number 15 extensively.

16 colors If you select 16 colors, the driver will operate as if only 16 colors are allowed (all colors over 15 will be white).

- The main benefit from this is that when using real-time commands, all the colors will remain the same during the animation. As well, you will avoid viewport redraws at the conclusion of these commands.
- One drawback to using only 16 colors is that the AutoCAD shade command cannot really be used with SHADEDGE set to 0 or 1.

DynaView text window

The DynaView Text Window pops up on top of the graphic screen when the F1 key is pressed.

If you select **Not Active**, when you press F1 (in an AutoCAD session), the board switches to VGA text mode, which requires more time.

❖ Note that the Text Window feature is also available to dual screen users.

If you configure the Text Window as **Active**, the next menu will appear so that you can set its parameters. The Text Window buffer is a circular one. When it becomes full, the first lines entered will be overwritten. Trailing blanks are not saved in the buffer. This means that with the 8K default buffer, you will usually have enough space for more than 200 lines.

Resizable AutoCAD Graphic window

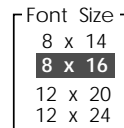
Select Active if you want the AutoCAD Graphic window to be resizable. *You must enable this if you want to have scroll bars for the Graphic window.*

Command prompt

This menu is presented with the default number filled in. To change this, press a number key, then press Enter to confirm (maximum = 9).

Font size

Use this menu to select your font size for menus in AutoCAD. You might find the default 8 × 16 font too small for high resolutions. We suggest the 12 × 24 font for resolutions over 1024 × 768.



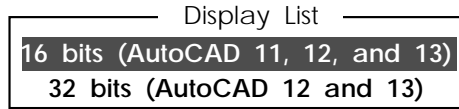
Mouse double-click

This menu allows you to change the software's interpretation of the speed of a mouse double-click. (DynaView allows you to assign single and double-click commands to your mouse or digitizer buttons.)

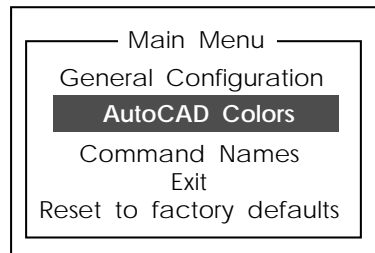
Display list type

This screen allows you to select the type of display list you want the driver to use.

AutoCAD 12 and 13 users have the choice between 16 or 32-bit display lists. The 32-bit display list almost eliminates regens, but it uses twice as much memory.



AutoCAD colors



AutoCAD background color

Select your background color and press Enter to confirm.



AutoShade background color



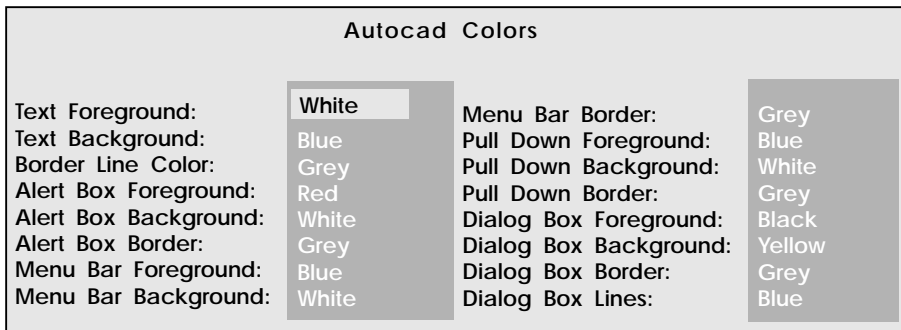
Select your background color and press Enter to confirm.

AutoCAD menu colors

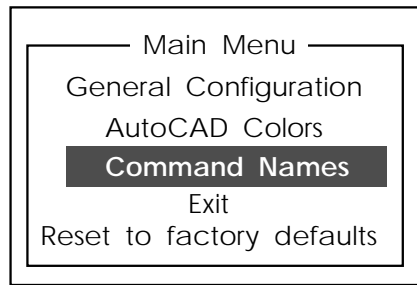
This menu is shown on the following page. To change the color of an element, use the ↑ and ↓ keys to move the highlight box onto the present color for that element, press Enter, then position the highlight box over the color you want for that element, and press Enter again. When you're finished with this menu, press Page Down to confirm your selections and exit.

DynaView colors

The DynaView colors are used by features such as the Text window, Bird's Eye, user menus, etc. DynaView also uses AutoCAD's text foreground and background colors within the Text Window and its title bar. AutoCAD Release 11 users may wish to change the Dialog box background to the more familiar 'yellow'.



Command Names



The **Command Names** menus are for changing the internal commands used by AutoCAD, to prevent conflicts with any other third party programs you may be using, and to adapt the driver to international versions of AutoCAD. The first menu presents the Matrox extended command set, and the second menu presents AutoCAD commands used by the driver.

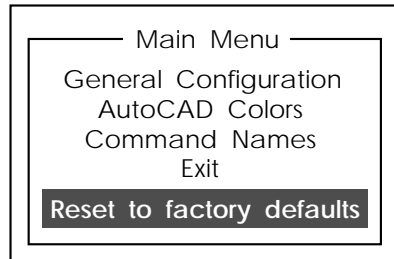
In addition, if you have selected a mode with 3D, commands used by DynaView /3D can be tailored to international versions of AutoCAD as well.

To change an internal command name, use the ↑ and ↓ keys and the Backspace key to position the flashing cursor, and enter the new name. The Enter key moves you to the next command, and Page Down will save and move to the next menu.

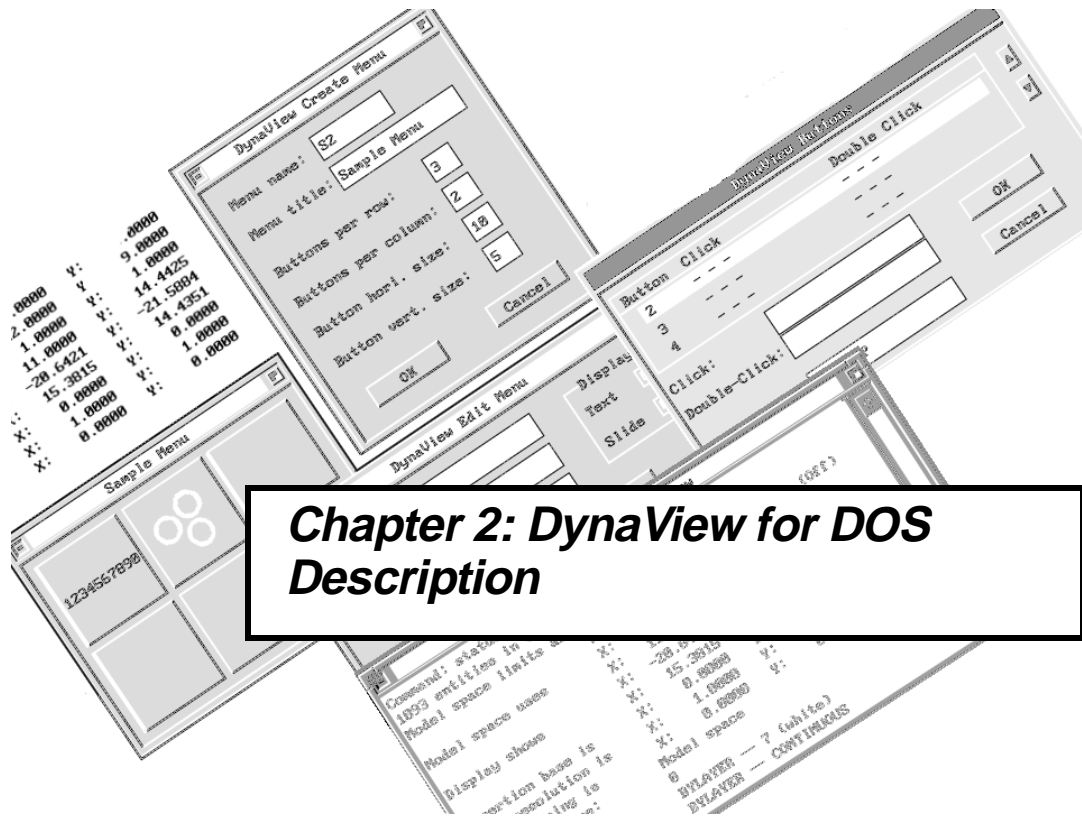
If you change any command names, you will also have to edit our *acad.mnu* and *popup.def* files, if you wish to use them. See the *dynaview.doc* file for customizing *acad.mnu* and the pop-up menu.

Reset to factory defaults

If, at any time, you wish to restore all options to the default settings, select the **Reset to factory defaults** menu item.



This will overwrite any settings you had previously chosen. You are therefore prompted to confirm your decision before the configuration is reset.



Chapter 2: DynaView for DOS Description

Overview

Capabilities

The Matrox MGA DynaView driver, provides the following capabilities:

- **DynaView display lists**
 - Very fast zooms and pans
 - Automatic and manual display list clean-up
 - 16 and 32-bit modes
- **DynaView windowing environment**
 - A movable/resizable AutoCAD graphic window
 - An overlay Text window that can be pinned
 - User-defined iconic menus
 - Digitizer button assignment
- **DynaView/2D command set**
 - Dynamic Bird's Eye
 - Dynamic Spy Glass
 - Dynamic scroll bars
- **DynaView/3D command set**
 - Fast Gouraud shading
 - Light sources and material properties
 - Real-time viewing, walk-throughs, and more

In addition Some more advanced DynaView customization topics, such as creating user menus off-line, modifying the color palette, modifying AutoCAD *.mnu* files, and more, are described in the *dynaview.doc* file located in your `\mga\acadrv` directory.

DynaView display lists

The purpose of a display list

The Matrox DynaView driver is display-list based. This means that the driver keeps a 'copy' of the AutoCAD database in a format which allows faster AutoCAD zooms, pans, and redraws than a standard driver.

One list per viewport

The display list is essential to the operation of most of the DynaView commands. The driver keeps one separate display list per AutoCAD viewport. A display list is created when a drawing is loaded, and the list is recreated every time you use the AutoCAD *regen* command.

Regenerating lists

One important note about a display list is that it does not contain the complete AutoCAD database, but rather just what was contained within the extents of the viewport at the last REGEN. As a result of this, there are cases (most often when you pan the drawing) where the display list has to be recreated to take into account the new extents of the viewport. This can take some time, and will be indicated by the AutoCAD message *Regenerating the drawing*.

Regen warning

Although zooming can usually be performed very quickly with a display list, there are cases where the zoom factor requested is too large, which again would cause a regeneration of the display list. All of the DynaView/2D commands help you avoid regeneration as much as possible. Some commands will not allow regeneration, and the rest will warn you graphically if your zoom or pan requires a regen.

Regular AutoCAD zoom and pan commands are more risky in that they do not warn of a possible regen unless you set the AutoCAD REGENMODE flag to get notified. Even with the REGENMODE flag set, it is more time consuming to be warned of a condition, cancel, and modify your request than to be simply warned graphically that your present zoom area will require a regeneration.

Note that if you do zooms and pans while in paper space, you will cause a regeneration. This is an AutoCAD limitation.

Automatic erased-objects cleanup

Typical erasure

When objects are erased in the AutoCAD database, AutoCAD sends 'black' vectors to the display list driver. These black vectors are annoying because when you make a redraw, you can often see the flashing effect produced by the display of the colored vectors, followed by the erasing action of the black vectors.

Immediate cleanup

DynaView display lists feature immediate automatic display list cleanup. When AutoCAD sends a black vector to the display list, our driver intercepts it, and removes the corresponding vector (to be deleted) from the display list immediately. This prevents the display list from growing in size as you perform edits and erasures. If you prefer, you can disable immediate display list cleanup. See the *dynaview.doc* file for details on the *blackvec* utility.

Manual cleanup

Note that the display list can be compressed even more by calling the MFLUSH or MFLUSHALL command. These commands will perform a manual cleanup of the display list. They will do the job quite a bit faster than a drastic REGEN would.

MFLUSH command

This function operates like AutoCAD's **Redraw** command. The difference is that, after you have performed edits and erasures, MFLUSH also 'compacts' the display list. This prevents the momentary appearance of erased objects when you perform subsequent zooms and redraws.

MFLUSHALL

This function is identical to the MFLUSH command, except that it operates on *all* viewports simultaneously.

For your convenience, both MFLUSH and MFLUSHALL commands are included in the DynaView/2D pop-up menu, which is described in Chapter 3.

AutoCAD Release 12 with 32-bit display lists

AutoCAD Release 12 introduces 32-bit display lists. The advantage of this new feature is that regens are virtually eliminated. The drawback is that display lists are twice as big. Therefore, depending on the memory available and the type of work you do, either the standard 16-bit or the new 32-bit display list may be more appropriate. For these reasons, the DynaView display lists support both sizes of display lists. Selection of the display list size is made in the driver configuration program (see Chapter 1).

- △ *Note that if you are running AutoCAD Rel 11, the 32-bit display list feature is not available; a 16-bit display list will be used regardless of your choice in the configuration program. This is an AutoCAD limitation.*

DynaView Windowing Environment

Window properties

The DynaView Windowing Environment is a modern environment which puts all of the following user interface elements in windows and dialog boxes:

- AutoCAD graphic window
- AutoCAD text window
- DynaView/2D Bird's Eye and Spy Glass
- User-defined iconic menus
- Many dialog boxes (menu creation, menu editing, button assignment, DynaView/2D flag settings)

General

All of the Matrox windows and dialog boxes have title bars. You can move any one of them by simply clicking in its title bar and dragging. Some windows are resizable. Some can be 'pinned', that is, they can be made to stay on screen permanently, by clicking in the small rectangle on the left side of the title bar. Other windows are modal (mostly the dialog boxes), and require immediate user action. The characteristics of each window and dialog box will be mentioned in their detailed descriptions later in this chapter and in Chapter 3.

Pinned windows

Pinned windows must first be brought 'in focus' before interacting with them. This can be done by simply clicking in them. Another way to get the focus to circulate from window to window is to click in the small icon on the right side of the title bar, which puts the current window in the background and the previous window in the foreground.

Focus indicator

The window currently in focus has its title bar colored differently. Windows that are pinned when you exit AutoCAD are automatically restored when you re-enter it (except if you reconfigure the driver with a new resolution or text font).

AutoCAD elements

Although it is redundant (with the Text Window), you can still have an AutoCAD **command line** area. You can also have a **status line** and a **side bar menu**. However, these three optional screen areas cannot be accessed by the Matrox windowing environment. It is not possible to have a window overlapping the command line area, side bar menu, or status line. The figure below illustrates a typical DynaView environment.

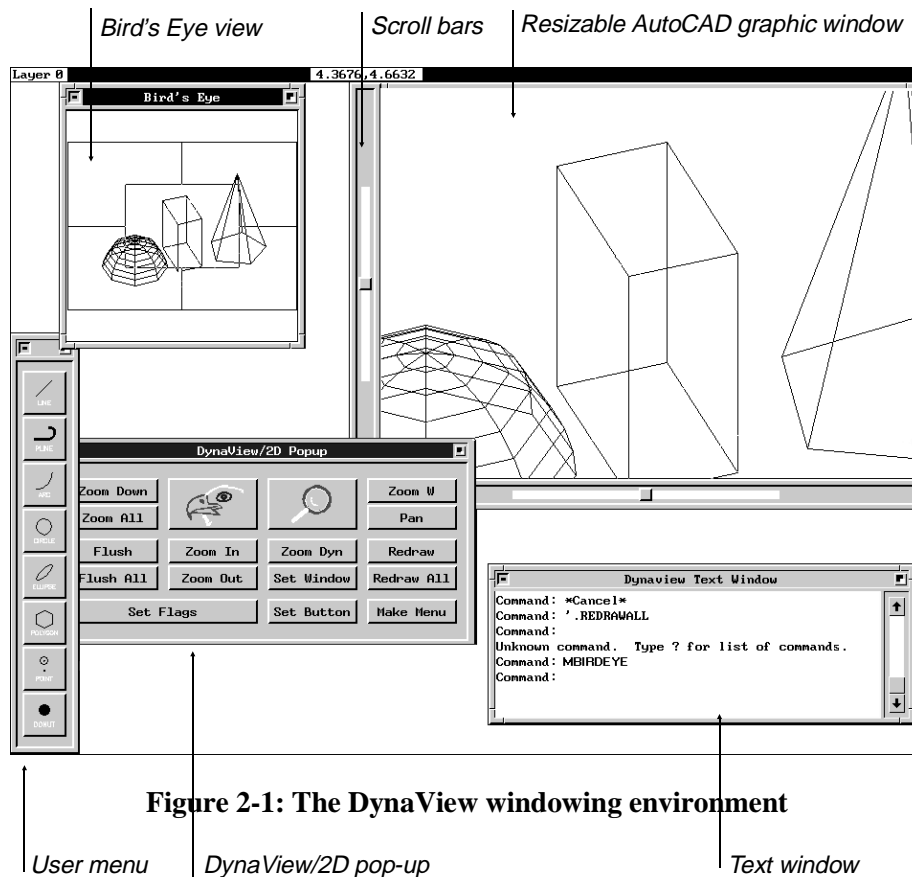


Figure 2-1: The DynaView windowing environment

AutoCAD Graphic Window

A window you can't close

The AutoCAD graphic window has certain particular properties:

- It can't be closed
- It has no title bar (in order to preserve screen space)
- It has the lowest priority, so it never obscures other windows

Moving and resizing

The AutoCAD graphic window can be moved and resized; click on the top border to move, or on the corners to resize. You will notice that the aspect ratio of your drawing is preserved, and that the minimum size is limited to about 50% of the screen.

When you enter AutoCAD for the first time, the AutoCAD graphic window occupies all the available space. It must first be resized before you can move it. The new size and position is saved for subsequent AutoCAD sessions.

- ❖ It is important to understand that if pinned windows (Bird's Eye and/or user menus) overlap the AutoCAD graphic window, the redraw speed will suffer. This is why it is preferable to resize the AutoCAD graphic window so that no overlap occurs.

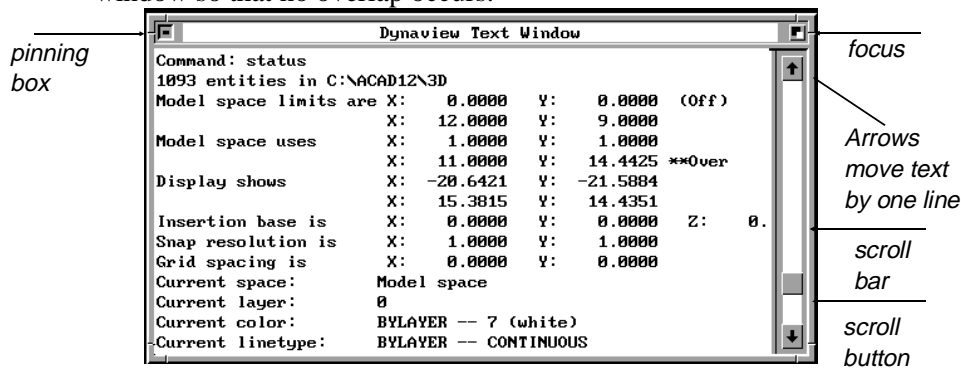


Figure 2-2: The text window

The DynaView Text Window

F1 to call up Text Window

The DynaView Text Window (Figure 2-2) is a resizable, movable, and pinnable window that pops up on top of the graphic screen when F1 is pressed. This is much faster than saving the graphic screen and switching to the text screen. The Text Window is enabled in the driver configuration utility.

Linked to text mode

If the Text Window is enabled, it will appear when AutoCAD switches to text mode, and disappear when AutoCAD switches back to graphic mode. For example, F1, HELP, STATUS, and LIST will bring up the Text Window, while F1 (again), REGEN, REDRAW, POINT, and other commands will remove it.

When AutoCAD switches to text mode, you cannot select in any other window. The cursor will change to an 'X', if you move it over other windows. When the Text Window is enabled, the driver traps all text output from AutoCAD, even in graphic mode. The only exception to this rule occurs when you enter the DOS shell.

Components

The Text Window has a title bar, scroll bar, and borders. Its colors are configurable. (The text background and foreground colors are the same as in the AutoCAD command area, and the rest of the window uses the Matrox colors; see 'Driver configuration' in Chapter 1).

Move and resize

The Text Window can be moved and resized. To move it, place the cursor in the title bar and click once; then move the outline of the window to a new location, and click again to confirm. To resize the window, click on a border or lower corner of the window, resize the outline of the box and click again to confirm.

Default size, location

The first time the driver is used, the Text Window size is 80×25 , and it appears in the upper-left portion of the screen. However, if you move or resize the window, it will subsequently return to the new location at the new size. The position and size of the Text Window is saved in the configuration file, so that between sessions, it will return to the previous location, at the last size you set.

Pinning

The Text Window can be pinned by clicking on the box at the left end of the title bar. Once it is pinned, all messages destined for the command area will also appear in the Text Window. If you press F1 while the Text Window is pinned, the *program focus* is transferred to the Text Window.

Scroll bar

The scroll bar at the right side of the window provides an efficient way to look at previous messages. To scroll up or down by one line, click in the \uparrow or \downarrow boxes at either end of the bar. To scroll to a particular area of the text, click once on the **scroll button** (it now moves with the mouse), move it, and click again to confirm the location. By clicking in the **scroll bar** above or below the button, you will move up or down in the text by one page.

Size of buffer

The size of the text buffer is determined in the configuration program. With the 8K default buffer, you will usually have enough space for more than 200 lines. You may notice that upon exiting AutoCAD, the Text Window flashes on the screen for an instant. This is because AutoCAD switches to text mode just before exiting.

DynaView commands

DynaView commands cannot be called while AutoCAD is in text mode. However, you can have the Text Window pinned, and call these commands while you are in AutoCAD graphic mode.

User Menus

User menus are movable/pinnable windows which contain a title bar and a variable number of buttons. Each button has an associated command string which DynaView sends to AutoCAD when you click on the button. The button can display either text or an AutoCAD slide.

Sample menus

Some sample menus are provided with the distribution disk. The description of a menu is kept in a *.bin* file. The name of the *.bin* file becomes the command name you use to call the menu from within AutoCAD.

Conflicts to avoid

Since AutoCAD first looks at the command before sending it to the driver, a menu can not use the name of an AutoCAD command or a function loaded by AutoLisp or the ADS program. As well, it can not use the name of a DynaView command.

When you call a menu, the driver first looks for the menu file name in the current directory, then in the directory pointed by the *ACADRV* environment variable.

Menu size

Since the overall menu size is defined in characters, the driver may refuse to load a menu if not enough room is available for it. In such a case, you will have to reduce the font size you are using.

Once a menu has been called during an AutoCAD session, its description is kept in memory, so it will come back faster on subsequent requests.

Pinned / Unpinned

When a menu is pinned, it will stay on screen and it will return to the same location automatically in subsequent AutoCAD sessions, until you explicitly unpin it. If a menu is not pinned, it is removed from the screen as soon as you click on one of its buttons or click outside the menu.

A menu button can call another menu, so it is possible to create a menu hierarchy. There is no limit to the number of menus that you can create and load, or to the number of buttons you can have in each menu.

Menu Creation

Menus can be created with the help of the DynaView MCREATE command, or externally with a text editor and the *MENU_CMP* utility.

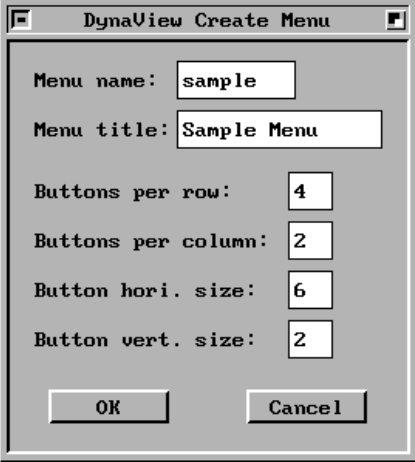
With MCREATE, menus are very easy to create, but button layout is restrained to simple arrangements. For your convenience, the MCREATE command is included in the DynaView/2D pop-up menu, as the **Make Menu** button. See the *dynaview.doc* file or the on-line documentation if you want to create more complex menu layouts.

MCREATE

Make Menu

The MCREATE command brings the modal dialog box, shown below.

This command lets you create a menu by supplying its file name, title name, number of buttons, and button size. All buttons have the same size, and they are automatically equally spaced. You can only have one line of text per button. The button size is defined by the number of text characters. This means that your button size should be no smaller than the longest text label you want to use.



DynaView Create Menu

Menu name: sample

Menu title: Sample Menu

Buttons per row: 4

Buttons per column: 2

Button hori. size: 6

Button vert. size: 2

OK Cancel

- ❖ Note that, as character fonts are usually twice as high as they are large (e.g.: 8→ × 16↑), in order to get square buttons you must specify twice the number of characters in the horizontal direction.

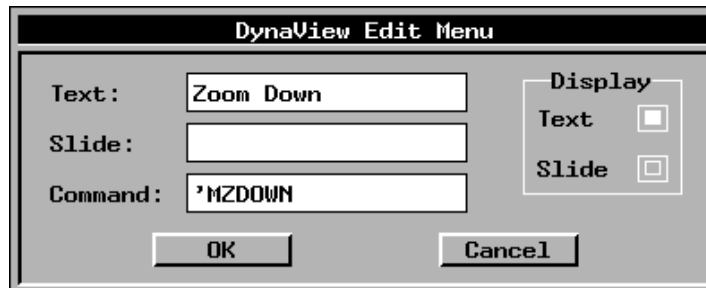
When you confirm your menu by clicking in the **OK** button, the new menu is shown immediately on the screen. It will be a 'naked' menu, with no display (text or other) in the buttons, and no associated commands. The following sections tell you how to edit menus, and add behavior to each button of a menu.

Menu editing

The text (or slide image) on a menu button can be edited 'on the fly'. You will certainly use this feature immediately after creating a new menu, but probably also at other times.

Uses

For instance, you may decide that a button you seldom use should be reassigned, that a button label should be changed, or that you want to switch from a text label to a slide icon.



To start editing

To edit a button, simply **hold down the Shift key and click on the desired button**. You will get the modal dialog box shown above. This dialog box allows you to change the button label (text or slide) and/or the command called. If you want to select a slide, that slide file must be located in the current directory or in the directory pointed by the `ACADRV` variable. When you confirm your change by clicking in the **OK** button, the menu is immediately updated, as well as its external `.bin` file.

- ❖ To change a menu layout, either recreate the menu with `MCREATE`, or use the external menu compiler described in the `dynaview.doc` file.

User Menu rules and tips

Text labels

If a text label is too long to fit in a button, it will be clipped. If a text label is shorter, it will be centered.

Slide files

Don't provide the file name extension (*.sld*) when you specify a slide file for a button label. Slides can be taken from a slide library. Specify a slide in the same way as you would in AutoCAD:

slide_lib(slide_name).

Slide colors

In slides, colors over 15 will be replaced by color 7 (white). You should not use the background color of the button (normally gray) in your slide. Any lines in that color will not be visible.

Slide Scaling

Slides are automatically scaled to fit in the button, and their aspect ratio is preserved. If your slide is roughly square, try to keep your button roughly square as well, otherwise, to preserve the aspect ratio, the slide may end up being rather small.

Command strings

The maximum number of characters for the command string associated with a button is 80. This means that complex commands and multiple commands can be sent.

The table following presents special characters that you can use in a command string. They have the same meaning here as in AutoCAD *.mnu* files.

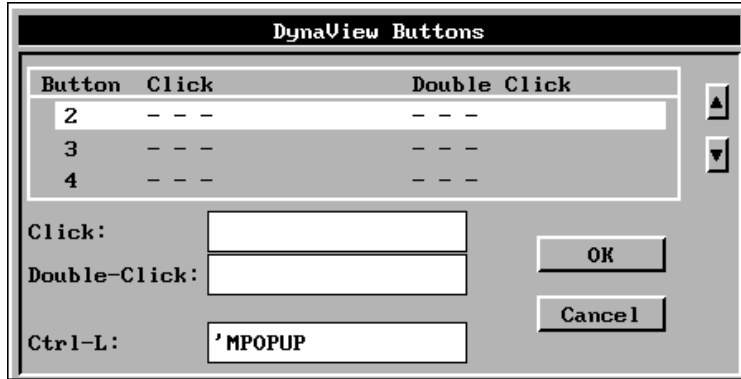
Character	Purpose
'	The single quotation mark is used to call a command transparently. This can be used with many AutoCAD commands (i.e.: 'redraw', zoom...) as well as for all the DynaView/2D commands.
;	The semicolon is used to insert a space. This is useful at the end of a string to insert multiple spaces, for instance, to confirm multiple defaults in an AutoCAD command.
^C	This will insert a Ctrl+C. It is useful at the beginning of a string to make sure AutoCAD exits the previous command before continuing with this new one. Actually, two Ctrl+C's are recommended, because this insures that you will completely exit the current command.
^O	Ctrl+O will toggle Ortho mode on/off.
As well, all the other mode toggle control keys such as Ctrl+B and Ctrl+G are supported.	

When a command string ends with one of the above special characters, the driver does **not** append a space (which would trigger a carriage return).

To terminate a 'normal' command without a carriage return, add a dummy character, followed by the special character '^H' (backspace).

Digitizer button assignment

The DynaView MBUTTON command is a quick and efficient way to overwrite the AutoCAD button assignment defined in the *acad.mnu* file. For your convenience, the MBUTTON command is included in the DynaView/2D pop-up menu, as the **Set Button** command. It presents the modal dialog box shown below.



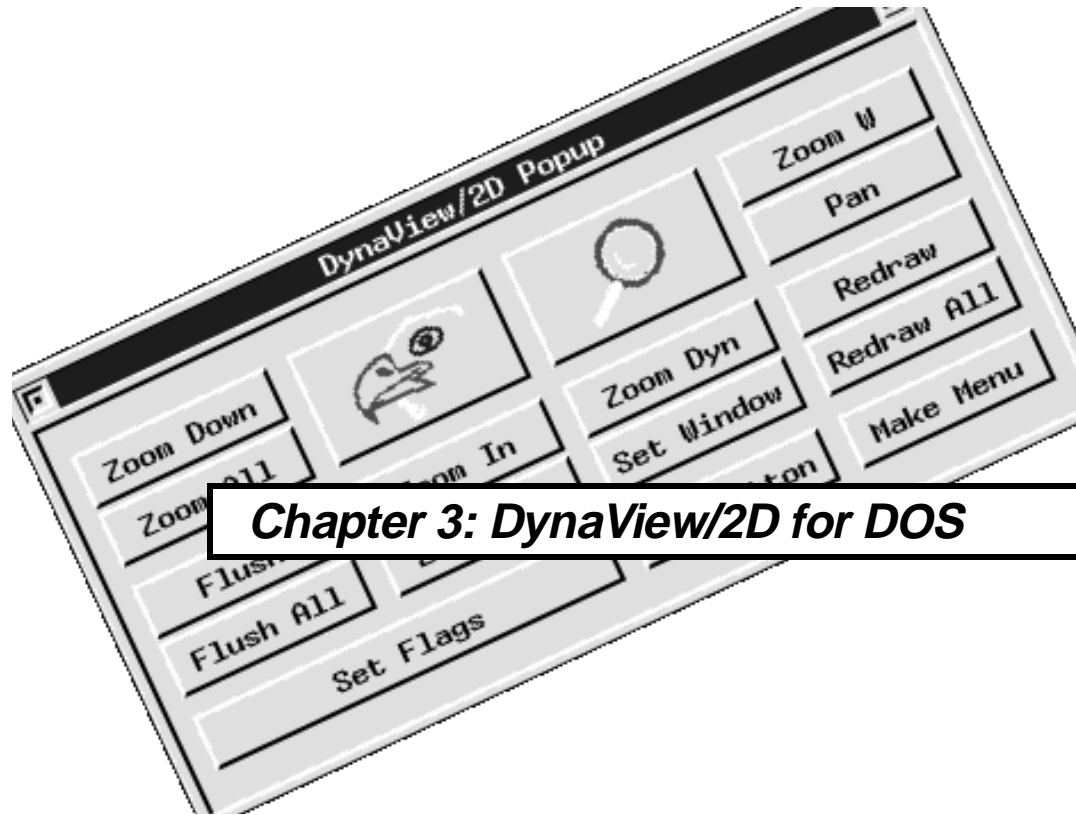
This command lets you redefine a digitizer puck containing up to 16 buttons. As in AutoCAD *.mnu* files, button #1 is not definable; it is reserved for picking. Buttons #2 to #15 can be defined, with different commands for a single and a double-click, if desired. The double-click speed can be controlled in the driver configuration (see Chapter 1).

Ctrl+L

You can also assign another command to the special Ctrl+L key combination. By default, the Ctrl+L key is set to call the DynaView/2D pop-up menu (see Chapter 3), and no digitizer button is defined for this purpose. This means that the setup (for digitizer buttons) contained in the *acad.mnu* file is valid.

As soon as a particular button is assigned, it takes precedence over the *acad.mnu* definition. Button assignments are retained for future AutoCAD sessions.

More than a simple command can be assigned to a button. The rules applying to a command string associated with a button are the same as for user menus (see page 2-14).



Chapter 3: DynaView/2D for DOS

Introduction



The DynaView/2D command set

This chapter introduces you to, and describes the DynaView/2D command set.

DynaView/2D is a comprehensive set of commands formulated to make your work easier and more productive. Built on top of the DynaView display lists and windowing environment, DynaView/2D includes commands such as Bird's Eye and Spy Glass, and a configuration dialog box (**Set Flags**) to control the use of scroll bars, Zoom factor, Bird's Eye (static/dynamic), etc.

The following table lists the DynaView/2D commands. In this table, the command name used in the pop-up menu (and throughout this manual) is given first, followed by the keyboard command name, a brief command description, and the page number.

Transparent All these commands can be called transparently with the (') prefix. This means they can be used while executing another drawing command. For example, you can begin a line command, zoom in for more precision with MBIRDEYE, and continue drawing with the line command.

The DynaView/2D commands			
Pop-up Name	Keyboard Entry	Description	Page
 Bird's Eye	<i>mbirdeye</i>	A small window containing a full view of the AutoCAD drawing. Zooms are specified in the Bird's Eye window.	3-5
 Spy Glass	<i>mspyglass</i>	A small window containing a magnified portion of the AutoCAD drawing. Zooms are specified in the AutoCAD Graphic window.	3-8
Zoom Dyn	<i>mzoomdyn</i>	Matrox version of Zoom Dynamic, it uses the display list.	3-10
Zoom In	<i>mzin</i>	Quick zoom-in by your pre-set zoom factor.	3-11
Zoom Out	<i>mzout</i>	Quick zoom-out by your pre-set zoom factor.	3-11
Zoom Down	<i>mzdown</i>	Zoom down to the display extents (or the Set Window).	3-11
Zoom All	<i>zoom v</i>	AutoCAD command to make a complete display list extents zoom all. This command overrides the constraints of Set Window .	3-12
Set Window	<i>mbirdwin</i>	Overwrite the default window size used by the Bird's Eye, Spy Glass, and Zoom Down .	3-12
Set Flags	<i>mconfig</i>	Dialog box to set the DynaView/2D flags.	3-13

The DynaView/2D pop-up menu

Calling the Popup

The easiest way to access the DynaView/2D commands is through the pop-up menu. To call it, type: *mpopup*.

Ctrl+L keys

By default, pressing the Ctrl+L key combination will call MPOPUP, giving you an even quicker way to bring it to the screen. If you use our pop-up extensively, you can assign it to a digitizer button, with the **Set Button** command.

Menu properties

The DynaView/2D pop-up menu is actually a standard user menu, as described in the previous chapter. It can be moved and pinned, and its position is remembered in future sessions. As well, the buttons can be edited with the 'Shift+Click' method described in Chapter 2. Even its layout can be changed, by using the stand-alone menu compiler we provide (described in the *dynaview.doc* file).

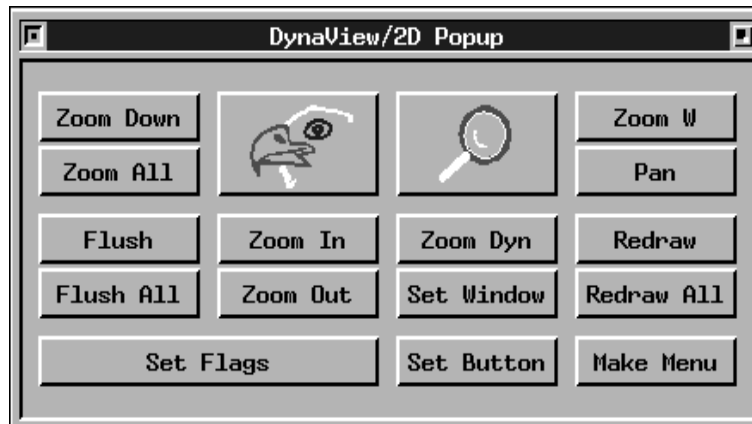


Figure 3-1: The Pop-up menu

Using the DynaView/2D commands

Bird's Eye

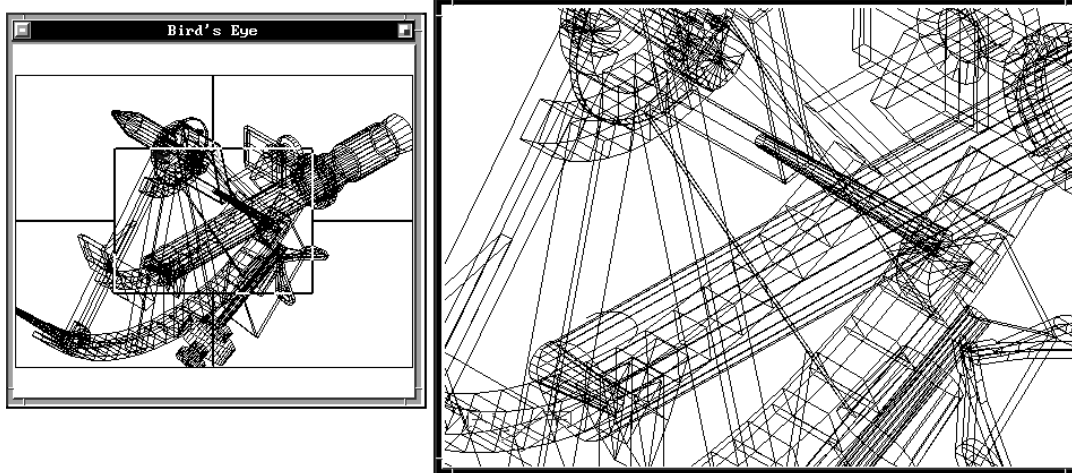


Figure 3-2: Bird's Eye view

This command (*mbirdeye* from the command line) gives you greater flexibility when zooming.



The Bird's Eye window can be moved and resized. In addition, it can be pinned, in which case it will remain on your screen after execution of the command.

Enabling

When you call the Bird's Eye, a window will be opened, with a full view of the display list of your active viewport. The current view of this viewport is outlined by a green rectangle with crosshairs extending outward from the rectangle. At this point the Bird's Eye command will intercept all mouse input, so you will not be able to move or resize other windows.

Operation

The operation of the Bird's Eye command is similar to the AutoCAD ZOOM Dynamic command:

- Click*** ■ First, you click in the inside portion of the Bird's Eye window to activate the command. You will see a zoom window that you can move inside the Bird's Eye.

- Toggle*** ■ By clicking again on the mouse pick button you can toggle between moving and resizing the zoom window. If you size the window too small, a red rectangle will appear in the Bird's Eye window, indicating that a zoom by that amount would cause a regen.

- Confirm*** ■ To confirm the zoom window, either click on the right mouse button or press the Enter key or the Space Bar. At this point the current AutoCAD viewport will be redrawn with the new zoom area.

- Cancel*** ■ If you want to cancel the operation, press the Esc or Ctrl+C keys.

'Static' and 'Dynamic' modes

The Bird's Eye command can operate in **Static** or **Dynamic** mode. The mode is specified through the **Set Flags** dialog box, described later in this chapter.

Static In Static mode, the AutoCAD viewport will not change until you confirm the zoom window. This is the default setting.

Dynamic In Dynamic mode, the AutoCAD viewport is redrawn in real-time as you move or resize the zoom window in the Bird's Eye.

Refreshing the Bird's Eye

When the Bird's Eye is pinned, it will sometimes get 'out of sync' when you make changes to your drawing, or when you change viewports. In some cases, the update is done automatically by the driver, for instance, when a redraw of the viewport is being made. At other times, ***you can force the Bird's Eye window to update by simply clicking in it.***

Set Window and Bird's Eye

The Bird's Eye command can be constrained by the **Set Window** command. If you used **Set Window** to limit the extents of the default window, the Bird's Eye will subsequently show only that desired portion, instead of displaying the complete display list extents. In this case it will be impossible to zoom outside of the set window range. You can use **Zoom All** to override this setting.

Additional points

Disabled

- The Bird's Eye window will not work if the current AutoCAD viewport is in **perspective**, while a **HIDE** is in effect or if AutoCAD is in **paper space** mode. When disabled, a large 'X' is displayed in the Bird's Eye window.

Aspect ratio

- When you resize the Bird's Eye, the aspect ratio of the current viewport is preserved in the Bird's Eye. In many cases, this will result in only part of the graphics area in the Bird's Eye being used.
- The position and size of the Bird's Eye is saved for the next time you call the command. This is also true for subsequent AutoCAD sessions.

See also

- You should also read the 'Hints and Tips' section near the end of this chapter.

Spy Glass

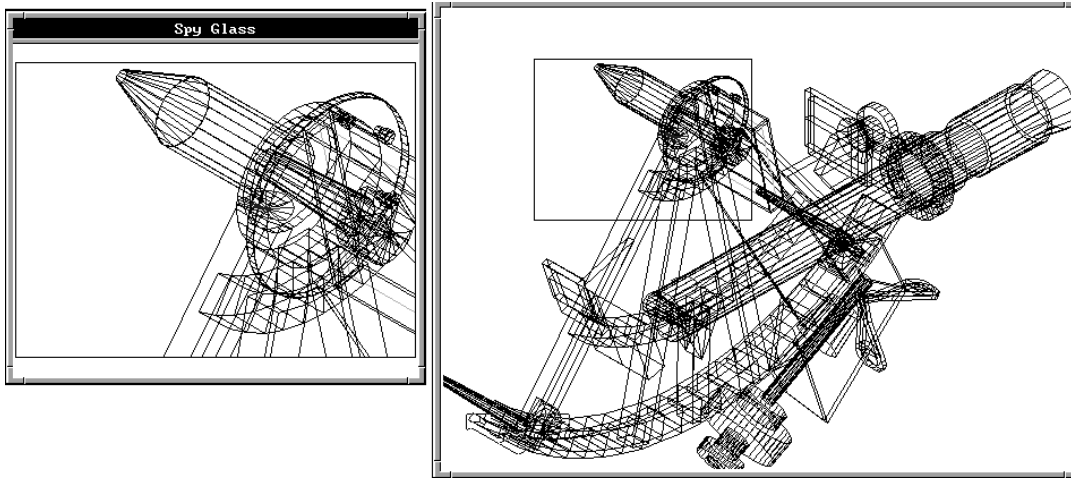


Figure 3-3: The Spy Glass command

This command (*mspyglass* from the command line, or click on the Spy Glass icon) is another useful way to zoom into your drawing.

Its operation is like the inverse of the Dynamic Bird's Eye: you specify your zoom area in the AutoCAD viewport, and it is the Spy Glass window which is redrawn in real-time (to show the zoom area) as you move and pan the zoom window.

**Can't be
pinned**

The Spy Glass window can be moved and resized, but unlike the Bird's Eye, it cannot be pinned. The Spy Glass works in Dynamic mode only.

Operation



You operate the Spy Glass command like the AutoCAD ZOOM Dynamic command:

- Click in the current AutoCAD viewport to activate the command. As soon as the Spy Glass command is invoked, it will intercept all mouse input, so you won't be able to move or resize other windows.

***Click again
to toggle***

- You will see the ‘zoom window’ which can be moved inside the AutoCAD graphic window. By clicking again on the left mouse button, you can toggle between moving and resizing the zoom window.

***Regen
warning***

- If you size the window too small, a red rectangle will appear in the viewport, indicating that a zoom by that amount would cause a REGEN.

Confirm

- To confirm the zoom window, click on the right mouse button or press Enter or the Space Bar. At this point the current AutoCAD viewport will be redrawn with the new zoom area.

Cancel

- If you want to cancel the operation, press the Esc or Ctrl+C keys.

Spy Glass Opening view

You can select between two types of ‘opening views’ for Spy Glass. Your choice is between **Current** or **Window** view, and you select it through the **Set Flags** dialog box, described later in this chapter. ***The default opening view is Current.*** (This is faster than Window view.)

Current view

With the **Current** opening view, the AutoCAD viewport is left as is when the Spy Glass command starts. With Current view, it is impossible to zoom outside of that area. You use this view to zoom in more.

Window view

With the **Window** opening view, the AutoCAD viewport is first redrawn to the full display list extents when the Spy Glass command starts. To help you locate your current view, a ***green rectangle*** with crosshairs extending outward from the edges ***outlines your present viewing area.*** With window view, you can zoom anywhere in the display list. With a large drawing, it can take a few seconds to start-up this command.

- ☞ *The Window view mode is subject to the **Set Window** command. Therefore, if **Set Window** was used before calling *Spy Glass*, your AutoCAD viewport will be redrawn to the *Set window* rather than the full display list extents.*

Additional points

- Disabled**
- The *Spy Glass* command can't be called if the current AutoCAD viewport is in **perspective**, if a **HIDE** is in effect, or if AutoCAD is in **paper space** mode.
- Aspect ratio**
- When you resize the *Spy Glass*, the aspect ratio of the current viewport is preserved. In many cases, this will result in only part of the graphics area in the *Spy Glass* being used.
 - The position and size of the *Spy Glass* is saved for the next time you use the command, and for subsequent AutoCAD sessions.

See also

- You should also read the 'Hints and tips' section near the end of this chapter.

Zoom Dynamic

This command (*mzoomdyn* from the command line, or the **Zoom Dyn** button in the pop-up menu) is similar to the AutoCAD ZOOM Dynamic command, but it operates faster, because it displays the view of the drawing at display list speed.

Subject to Set Window

The Matrox **Zoom Dyn** is also more flexible, because it is tied to the **Set Window** command. The AutoCAD ZOOM Dynamic command always displays a full view of the drawing. If **Set Window** was used, the Matrox **Zoom Dyn** command will display only that portion of the display list outlined by the **Set Window** command. Otherwise, the Matrox **Zoom Dyn** will work in the same fashion as the AutoCAD ZOOM Dynamic.

Regen warning

If the zoom you select might cause a regeneration, you are warned by a red square icon in the lower-left corner of the screen.

Zoom In

This command (*mzin* from the command line, or the **Zoom In** button in the pop-up menu) zooms into the image by the factor currently set in the **Zoom Factor** window of the **Set Flags** dialog box. (See the **Set Flags** command that follows for information on how to set the Zoom Factor.)

No regens

If the Zoom Factor you requested requires a regeneration, your display will zoom to the highest value possible that avoids the regeneration.

Zoom Out

This command (*mzout* from the command line, or the **Zoom Out** button in the pop-up menu) zooms out from your drawing by the zoom factor currently defined in the **Zoom factor** window of the **Set Flags** dialog box.

Zoom Down

This command (*mzdown* from the command line, or the **Zoom Down** button in the pop-up menu) provides a quick zoom out to a full view of the drawing. When you start out, the zoomed down view is set to the view you have when loading the drawing. Later on this view could be modified as you cause drawing regenerations.

Subject to Set Window

In addition, the **Zoom Down** command is constrained by the **Set Window** command, when **Set Window** is used to set a specific window, it temporarily overwrites the default window extents.

Zoom All

Override

If a **Set Window** limit is active and you really need to zoom down to the full extents of the display list, use the **Zoom All** pop-up button, which calls the AutoCAD ZOOM Vmax command.

- ❖ Note, however, that this will cancel your **Set Window** limit, as explained in the **Set Window** command description.

Set Window

This command (*mbirdwin* from the command line, or the **Set Window** button in the pop-up menu), lets you define a *temporary window* into the display list. This window is, of course, *smaller than the display list extents*.

Once set, this limit will be observed by:

- The Bird's Eye (from which the keyboard name comes)
- The Spy Glass
- The Matrox **Zoom Dyn** command
- The **Zoom Down** command

Purpose

The **Set Window** command is useful when you have to work for some time in a specific portion of a very large drawing. It is particularly useful in the 32-bit display mode, where the difference between the display list full extents and your working window can be huge.

Operation

There is no user interaction, as such, with the **Set Window** command. To Set a window, simply zoom in the drawing (with any command) to your desired view, and then call the **Set Window** command. The current window will be saved as the default for those Matrox commands previously listed.

Independent

The **Set Window** command works independently on each viewport; you can switch viewports and set a different limit for each one.

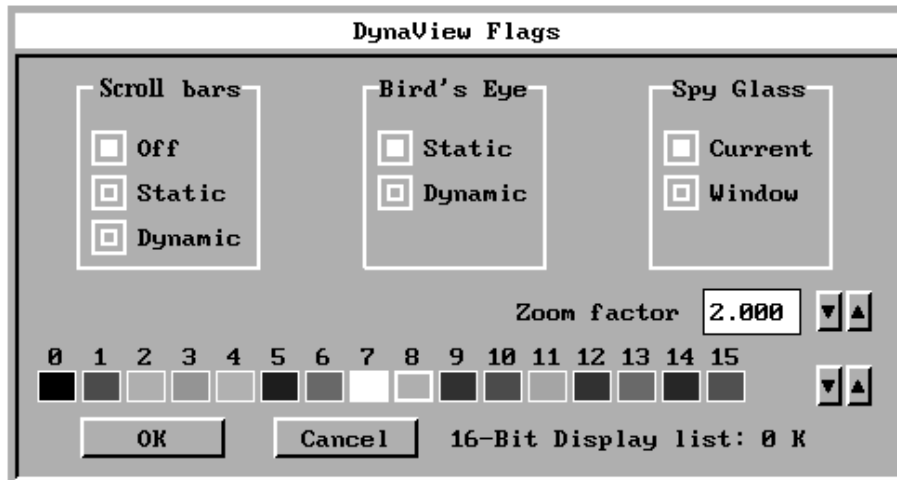
Cancel

A **Set Window** is cancelled when you REGEN the viewport, or if you zoom to the display list extents with the **Zoom All** button of the pop-up menu (*Zoom V* on the keyboard). As well, *all* set windows are cancelled if you use the AutoCAD VIEWPORTS command.

Auto adjusting

Some commands, such as **Zoom Out**, AutoCAD PAN, and AutoCAD ZOOM Dynamic can modify the displayed view such that it won't match the Set window anymore. If this happens, the Set window will be automatically adjusted (enlarged and/or translated). Note, however, that this situation can't happen when you use the Bird's Eye, Spy Glass, Matrox Zoom Dynamic, or the Matrox scroll bar.

Set Flags



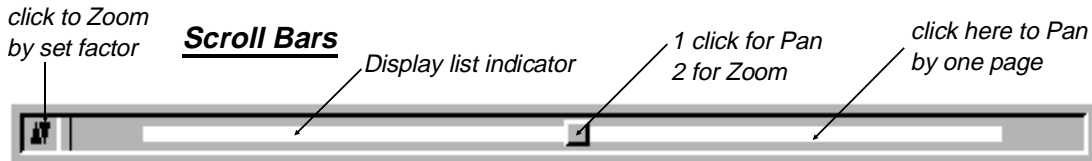
This command (*mconfig* from the command line, or the **Set Flags** button in the pop-up menu) lets you configure the DynaView/2D flags and modes, through an easy to use dialog box. In the lower right corner, you will see the current display list type (16 or 32-bit) and size.

Flag controls

Using **Set Flags**, you can :

- Configure Scroll bars
- Set the Zoom factor
- Configure the Bird's Eye
- Configure the Spy Glass

❖ Note that when you end an AutoCAD session, the current settings of all the parameters of **Set Flags** are retained for your next session.



Options

The **Scroll Bars** feature can be set to **Off**, **Static**, or **Dynamic**. The default is **Off**.

❖ In order to have scroll bars available, you must select the Resizable graphic window in the configuration program (see page 1-6).

The scroll bars allow quick panning and zooming of your AutoCAD drawing. When enabled (either Static or Dynamic), the scroll bars appear on the left and bottom sides of the AutoCAD graphic window. If necessary, the graphic window will be automatically resized to make room for the scroll bars.

Components

Each scroll bar contains a **scroll button**, used to pan and zoom, and an underlying **white bar**, which represents the portion of the display list currently shown on the screen. In a drawing displayed to its full extents, the white bar fills the scroll bar entirely. Panning will then be impossible, as Matrox scroll bars do not allow regens. As you zoom into your drawing (with any AutoCAD or DynaView/2D command), the white bar will decrease in size, and panning becomes possible.

In a multiple viewport environment, the scroll bars always apply to the current viewport. When switching from one viewport to another, you will notice that the size and/or position of the white bars is updated.

Panning

If you click once on the scroll bar button, you enter Panning mode. At this point, you can move the button to pan to the desired location, and then click the right mouse button to confirm and exit from this mode. To cancel, press Esc; to switch to Zoom mode, click on the left mouse button again.

Zooming

To enter Zoom mode directly, click twice on the scroll bar button. In Zoom mode, as you move the mouse horizontally (for the horizontal scroll bar) or vertically (for the vertical scroll bar), the white display list bar gets smaller or larger. To confirm the zoom, press the right mouse button. To cancel, press Esc or click the left mouse button again. Note that it is not possible to toggle back to Pan mode from Zoom mode.

Dynamic mode

If the scroll bars are set to Dynamic rather than Static, the viewport will be panned or zoomed in real-time as the scroll button is moved. See the notes about animation commands on page 3-18.

Page mode

If you click in the scroll bar above or below the scroll button, the drawing is panned by one 'page'. That is, the area just beside the current view is displayed. When there is not enough room to do a full 'page', without regenerating the drawing, a partial page is applied to move the window in that direction as much as possible.

Zoom buttons

You will also notice two small arrows at the intersection of the two scroll bars. These arrows activate the **Zoom In** and **Zoom Out** commands, which cause a zoom by the current **Zoom Factor** (described on the next page).



Bird's Eye mode flag

The Bird's Eye can be set to either **Static** or **Dynamic** mode. The default is **Static** mode. The Bird's Eye command description explains these two modes.



Spy Glass opening view flag

The Spy Glass opening view can be toggled to be either the **Current** view or the **Window** view. The default is the **Current** view. See the Spy Glass command for further explanation of this control.

Zoom Factor flag



The **Zoom Factor** is used by the **Zoom In** and **Zoom Out** commands, as well as by the zoom in/zoom out button of the scroll bars.

Default

The *default Zoom Factor is 2.0*. Other pre-set values can be selected by simply clicking the 'up' and 'down' arrows. You can also click in the **Zoom Factor** box and enter your own value (between 1.0 and 50.0).

AutoCAD Color Map Viewer



The **Color Map** viewer allows you to scroll through the range of available AutoCAD colors. Simply click on the up or down arrows at the right to display a different row of available colors.

Display List Size and Type

16-Bit Display list: 0 K

The **Set Flags** dialog box displays the current display list size, giving you a good idea of the percentage of your RAM memory being used at any time.

Total displayed

Note that if multiple viewports are being used, the value displayed is actually the total memory space allocated for **all** the display lists.

Display list type

The current display list type is also shown in the dialog box: 16-bit or 32-bit.

Other DynaView and AutoCAD commands

In addition to all of the DynaView/2D commands, the Matrox pop-up menu contains other DynaView commands and some AutoCAD display commands. These additional commands are listed in the following table.

Other DynaView commands		
Pop-up Name	Keyboard Entry	Description
Flush	<i>mflush</i>	DynaView command to recreate the display list of the current viewport
Flush All	<i>mflushall</i>	DynaView command to recreate the display list of all viewports
Zoom W	<i>zoom w</i>	AutoCAD Zoom Window command
Pan	<i>pan</i>	AutoCAD Pan command
Redraw	<i>redraw</i>	AutoCAD Redraw command
Redraw All	<i>redrawall</i>	AutoCAD Redrawall command
Set Button	<i>mbutton</i>	DynaView button assignment dialog box
Make Menu	<i>mcreate</i>	DynaView menu creation utility

Hints and tips

This section addresses some points which can help you make the most effective use of the DynaView/2D features.

Response time during animation

Animation commands

Animation techniques are used in the following DynaView/2D commands:

- Bird's Eye in Dynamic mode
- Scroll Bars in Dynamic mode
- Spy Glass.

In these cases, to ensure a smooth response from the digitizer, the commands are implemented in a manner similar to AutoCAD's 'dragging outline' method. That is, if the drawing is too complex to be displayed in real-time (particularly if you move the digitizer rapidly), only portions of the display list will be shown while you are dragging the cursor. When you stop moving the digitizer, the display of the drawing will catch up.

Animation in 8 bits per pixel mode

Less colors

In 8 bpp mode, animation is always 'smooth', being performed using 'double buffering' techniques. In this pixel mode, double-buffering is performed by reducing the number of colors. (You can only have 16 colors rather than 256 during animation.)

16 color map

If you elected to run the driver using the 16 color map (General Configuration, screen 3) you will not be affected by this limitation. If you are using the default 256-color map, colors over 15 will get temporarily remapped into the 0-15 range.

Remapping algorithm

The remapping algorithm uses modulo 16 arithmetic (the remainder of division by 16) to select the color. For example, color 17 will become color 1, color 34 will become color 2, and so on. Note that with this method, color numbers that are a multiple of 16 (16, 32, 48, etc.) will get remapped to 0 (black), which is the default background color.

**Ending
animation**

Another smart reason to run the driver in 16 color mode is that, in 256-color mode, all viewports (as well as the Bird's Eye, if pinned) have to be re drawn at the conclusion of the animation command. This can take some 'long' seconds if you have many viewports or a very large drawing.

Animation in 16 and 24 bit per pixel modes

In 16 and 24 bit per pixel modes (or in 8-bit per pixel with 3D), no color change occurs when switching into animation. However, 'Smooth' animation is not available in all cases, therefore animation could be more jerky. To have smooth animation, go back to the driver configuration utility and select a mode with 2D, Smooth or 3D, Smooth.

Efficient window layout for animation

**Avoid
overlap**

Best performance (faster redraws) with the Dynamic Bird's Eye and scroll bars is achieved by preventing windows (user menus, Text Window, Bird's Eye itself) from overlapping with the AutoCAD viewport. If you don't do this, the viewport is redrawn in multiple steps, which will slow down the animation.

Avoid overlapping frames by either resizing your AutoCAD window smaller, or by sizing the other windows as small as possible.

DynaView/2D commands with 32-bit display lists

With a 32-bit display list, regens almost never occur. You will find that the **Set Window** command becomes very useful, because at some point, the drawing in the Bird's Eye will be too small.

**Larger
extents**

In 32-bit display list mode, you will notice that the display list extents are larger than the AutoCAD drawing extents, so the **Zoom All** command (AutoCAD Zoom V) is not very practical, because it presents a very small view of the drawing. Use the Matrox **Zoom Down** command instead; this will zoom you out to your **Set window**.

No regen indicator

In 32-bit display list mode, there is no regen indicator with the Bird's Eye, Spy Glass, and **Zoom Dynamic** commands. When you get too close to a REGEN, the zoom window will simply be locked at that point.

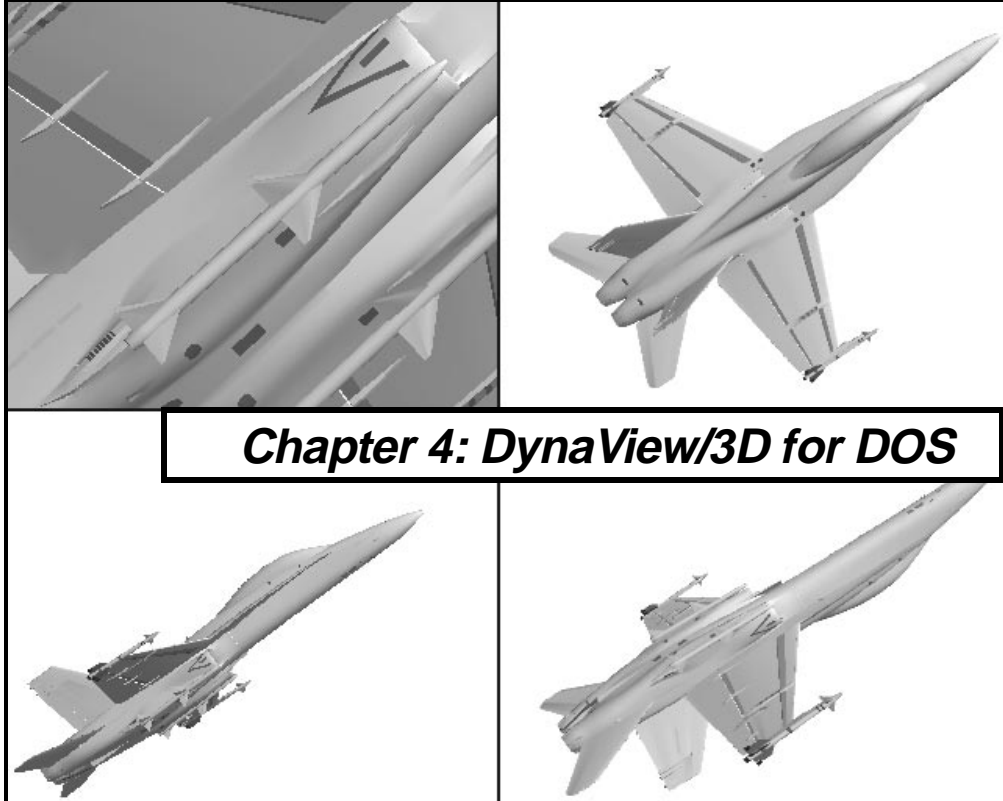
Display list indicator

Due to the very large display list extents, the white bar display list indicator in the Scroll bars will often become smaller than the scroll bar button itself. In this case, the Zoom mode is still accessible, but not practical, especially when the Scroll bar is in Static mode.

Shorter command names

If you like to use the keyboard to call the DynaView/2D commands, you will certainly find that some names are quite long. If this is a problem, use the configuration utility to rename your favorite commands to shorter names (remember to avoid conflicts with other AutoCAD, ADS, or Auto Lisp commands).

- After renaming a DynaView command, you will have to update the DynaView menus (in particular, the pop-up) that include these modified commands.
- Also, if you customized your side-bar or pull-down menus to include the DynaView commands, any changes you make to the command strings will have to be reflected in your *acad.mnu* file, or these menus will cease to work for those particular command strings you have modified.



Overview

3D Rendering within AutoCAD for DOS

The Matrox DynaView/3D command set gives you the power to render a 3D model inside AutoCAD for DOS. It features a set of viewing commands which can greatly help in visually confirming the accuracy of your model.

With the Matrox DynaView/3D command set you can render, manipulate, and move around inside a 3D model without leaving AutoCAD. The MATRD command scans the AutoCAD database and displays the model in various display modes from Wireframe to Gouraud shaded.

DynaView/3D can be loaded only if a mode with '3D' was selected in the driver configuration utility (see Chapter 1).

Rendering mode

MATRD initiates Rendering (RD) Mode. The other 3D commands allow you to:

- Modify the viewpoint
- Set your own lighting
- Assign material properties
- Perform real-time viewing

The MATEXIT command returns you to your normal AutoCAD display, with the last viewing position you had in RD mode.

RD mode allows several viewports to be active at any given time. You must bear in mind, though, that many AutoCAD commands and DynaView/2D commands can't be used in RD mode. Refer to the 'About Rendering mode' section on page 4-49 for more details.

Loading the 3D command set

Four ways to load

The DynaView/3D command set must be loaded before it can be accessed. This can be done in four different ways:

1. Loading from the pop-up menu

Type *mpop3d* to bring up the 3D menu. Then, simply click on the **Load** button. This menu provides the easiest access to the 3D commands.

If you installed the driver in a subdirectory different than the default one (*\mga\acadrv*), you will have to edit the menu button and enter the real path. To do this press Shift and click on the **Load** button.

2. Loading from the command line

To load the 3D command set manually from the command line, type the following line, exactly as shown:

```
(xload "/mga/acadrv/dv3dads")
```

The '*/mga/acadrv*' indication is the directory where the Matrox driver and support files are loaded by default. If you installed the driver in a different directory, you must substitute the path to that directory.

3. Load from the Matrox .mnu file




Special menu files called *acad13.mnu*, *acad12.mnu*, and *add-on.mnu* are provided on your Matrox diskette. Each of these files has a pull-down menu and a side-bar menu, each menu containing all the 3D commands, including one to load the command set. If you install one of these menus, you can simply click on the **Load** button.

If you installed the driver in a subdirectory different than the default *c:\mga\acadrv*, you will have to change four lines in the *acad.mnu* file. Using an ASCII text editor, search for the phrase 'MGA'; at each occurrence of this phrase, enter the correct path to the driver files.

4. Loading from acad.ads

The Matrox 3D command set can also be automatically loaded by adding the proper instruction to the optional *acad.ads* file. See the *ADS Programmer's Manual* for more details.

The DynaView/3D command set

Pop-up Button	Command	Description	Page
 Render	<i>matrd</i>	Enter RD mode.	4-6
Display	<i>matdisplay</i>	Change display option.	4-8
View Point	<i>matvpoint</i>	Change viewpoint.	4-9
 Dyn View	<i>matdview</i>	Dynamically manipulate the model.	4-12
 Dyn Walk	<i>matdwalk</i>	Dynamically walk around in the model.	4-16
Rec/Play	<i>matrecord</i>	Record/Playback of walk-throughs.	4-18
Lights	<i>matlight</i>	Change or create light sources.	4-21
Materials	<i>matmaterial</i>	Define material properties.	4-33
Save Image	<i>matimage</i>	Save rendered viewport to a <i>.bmp</i> file	4-39
Config	<i>matcfg</i>	Configure RD mode.	4-40
Exit	<i>matexit</i>	Leave RD mode.	4-47

The DynaView/3D pop-up menu

MPOP3D

The easiest way to access the DynaView/3D commands is through the 3D pop-up menu. To call it, type: *mpop3d*.

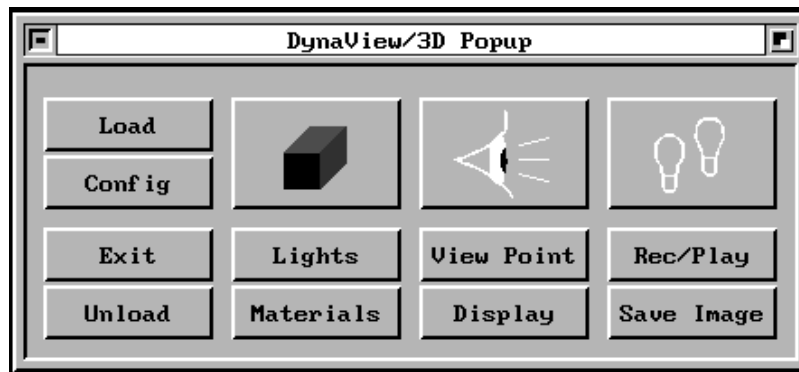
If you use the 3D pop-up menu extensively, you can assign it to a digitizer button, using the **Set Button** command (in the DynaView/2D pop-up).

Menu properties

The DynaView/3D pop-up menu is actually a standard user menu, as described in Chapter 2. It can be moved and pinned, and its position is remembered in future sessions. As well, the buttons can be edited with the 'Shift+Click' method described in Chapter 2. Even its layout can be changed, by using the stand-alone menu compiler we provide (described in the *dynaview.doc* file).

Load 3D commands

To use the 3D pop-up menu, you must first load the 3D command set. You can use the **Load** button for this. If you wish, you can use the **Unload** button to remove the 3D command set from memory after you leave rendering mode. These commands are explained in detail in this chapter.



MATRD



Entering rendering mode

To enter RD mode, switch to the desired viewport and then call the MATRD command (or you can click on the shaded cube icon button).

By default, everything in the viewport is selected for rendering. If you don't want to render everything, you can configure MATRD to ask you to manually pick which entities to render. (See the MATCFG command.)

Once the selection is made, the MATRD command will start scanning the AutoCAD database in order to *tessellate* the geometry.

In the process, the following entities will be lost:

- 2D lines without thickness
- 3D lines (must be a surface)
- points, text, dimensioning and shapes information

Once the tessellation is completed, the viewport is cleared and replaced by the rendered image, with the same viewpoint as before. The rendering is done in a perspective or orthogonal projection, depending on the current AutoCAD viewing mode for the viewport.

Second rendered viewport

To have a second rendered viewport, simply switch to another viewport and redo the same operation.

Changing defaults

There are many parameters that you can modify to change the way the rendered image is created. See the MATCFG command for details.

3D display list

The content of an RD viewport is saved in a 3D display list. Once this display list is created, you will be able to perform the following operations without having to regenerate the drawing:

- Change the display option with MATDISPLAY.
- Change the viewpoint with MATVPOINT, MATDVIEW, or MATDWALK.
- Change the lighting with MATLIGHT.

You *will* have to run MATRD again to update the display list if you change:

- Material properties (with MATMATERIAL)
- Almost any of the MATCFG parameters.

MATDISPLAY

Change the display option

With MATDISPLAY, you can change the rendering display option without having to regenerate the 3D display list.

You start the command by typing *matdisplay* or by clicking on the **Display** button of the DynaView/3D pop-up menu. You are asked to select your display option:

Wireframe/Hidden/Bordered/Flat/Gouraud <default>

The *<default>* indicates the last display option you used. At startup, the default display is Gouraud. You may choose between:

Wireframe	The display is made with lines only. No light is applied. The AutoCAD color is used for the color of the entity.
Hidden	The display is made as in Wireframe, but with hidden lines removed.
Bordered	The display is made with flat colored polygon fills. The AutoCAD color is used for the color of the entity.
Flat	The display is made with flat colored polygon fills and borders, using the material properties and light sources defined through MATMATERIAL, MATLIGHT, and MATCFG.
Gouraud shaded	The display is made with smooth shaded polygons, using the material properties and light sources defined with MATMATERIAL, MATLIGHT, and MATCFG.

Gouraud shading is generally the best choice, although wireframe display can help speed up redraws in the case of very complex models. Note that the current display mode has no impact on the regen time in MATRD.

The display mode you select is saved in the Matrox configuration block (see the MATCFG command). The next MATRD will be performed with the most recent display mode you selected.

MATVPOINT

Change the viewpoint

With the MATVPOINT command, you can quickly switch from Orthogonal to Perspective mode. This command (**View Point** button in the DynaView/3D menu) can be used as a ‘manual’ way to define a new viewpoint (useful for AutoLISP programs). When you invoke this command, the following prompt appears:

Orthogonal/Perspective <current>

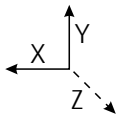
Orthogonal

If you select Orthogonal, you get the following prompt:

?/Center/Direction/Height:

When your viewing mode is Orthogonal, MATVPOINT can be used to define a new **center point, viewing direction, and height**. If you press ‘?’, you get a report of the current settings. Use the *Center* parameter to pan the model; use the *Height* parameter to zoom it. Use the *Direction* parameter to switch to a new viewing angle. See examples below:

Center and Direction examples:



Center: 20,0,0



Center: 40,0,0



Direction:0,0,1

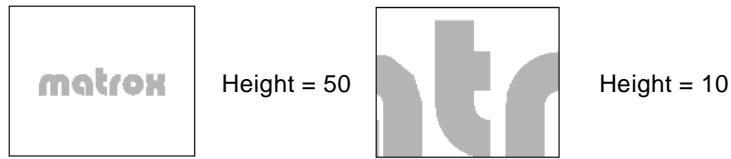


Direction: 0,0,-1



Direction: 0,1,0 (overhead, on black background)

Height examples:



Perspective

If you select Perspective, you get this prompt:

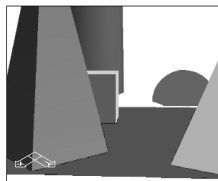
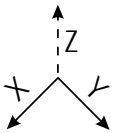
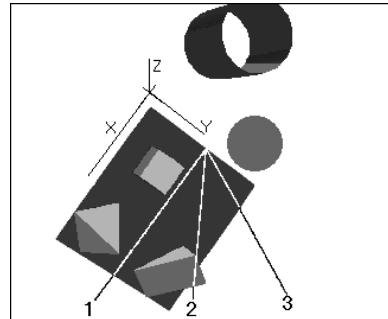
**/Camera/Focallength/Target/TWist:*

If the selected viewing mode is 'Perspective', MATVPOINT can be used to define the camera location, lens focal length, target point, and twist angle. If you press '?', you get a report of the current settings. These perspective parameters default to the ones set via the AutoCAD DVIEW command.

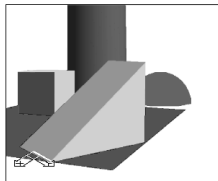
- ❖ Note that, if you have never used the AutoCAD DVIEW command with your drawing, the viewing distance may change by quite a bit when you switch to Perspective view.

Camera examples:

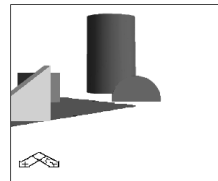
Changing your camera settings is like changing the camera position, but always looking at the same target area. To the right, you see an overhead view of the three different camera positions shown below. If you have user directional light sources, they will move with the camera.



1. Camera:15,5,2



2. Camera:15,10,2



3. Camera:15,17,2

Focal length examples:

Changing this parameter is like changing a camera lens. A wide angle lens will show the usual amount of pincushion distortion.



Focal length = 10



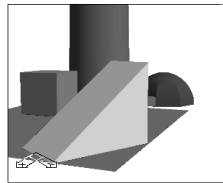
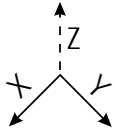
Focal length = 50



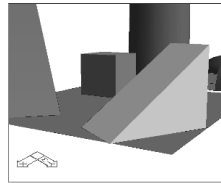
Focal length = 75

Target examples:

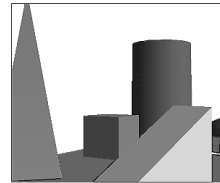
Changing the target location is like standing in the same spot and looking around. The target spot is centered in the rendering window. Below are three different target locations, camera on the same spot.



Target: 0,4,0



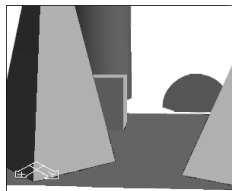
Target: 5,4,0



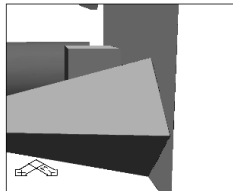
Target: 5,4,2

Twist examples:

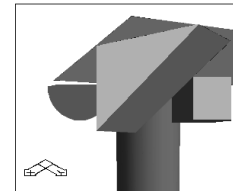
The twist option lets you tilt your view around the line of sight. The twist angle is measured counterclockwise, in radians, with zero radians being to the right at the 3 o'clock position.



Twist = 0



Twist = 1.6



Twist = 3.14

MATDVIEW



Dynamically manipulate the model

The MATDVIEW command is a dynamic viewing utility intended to be used as a tool to view a 3D model as if you were holding it in your hand.

Assume that the Z axis is between you (the camera) and the target point. The MATDVIEW command lets you zoom, pan, and rotate the model around the X, Y, or Z axes.

With MATDVIEW, the model can be constantly moving. The type of motion (rotation, zooming, or panning) depends on the *Operation mode* (these modes are described on the following pages). The apparent speed of the motion (more accurately, the motion increments, since the speed varies for models of different complexity) depends on the position of the cursor on the screen. The closer you are to the center of the screen, the slower the motion appears.

When you call the MATDVIEW command (you can also use the 'eye' icon button of the DynaView/3D pop-up menu), you will notice horizontal and vertical 'bands' passing through the center of the screen.

- If the mouse is positioned at the intersection of these two bands, the model stops moving. This area is referred to as the *dead spot*.
- If the mouse is put within a band, the motion will be *pure*.
- If the mouse is not in a band, the motion will be a combination of the X and Y position of the mouse, with the speed of the motion increasing as the cursor moves away from the center.

MATDVIEW Operation modes

The five operation modes of MATDVIEW are presented in the following pages.

Selecting/changing modes

Click to start moving

You select the mode by pressing the appropriate keypad or keyboard key (1, 2, 3, 4, or 5). Once you select a new mode, the model only starts moving when you press the left mouse button (this avoids unwanted motion during the mode switch).

Status line

A mode-identifying status line is displayed in the AutoCAD command area.

Mode 1	Rotate Y axis and rotate X axis
Move right	Rotate the model to the right around the Y axis.
Move left	Rotate the model to the left around the Y axis.
Move up	Rotate the model towards the top around the X axis.
Move down	Rotate the model towards the bottom around the X axis.

Mode 2	Twist right/left and Zoom in/out
Move right	Rotate (twist) the model clockwise around the Z axis.
Move left	Rotate (twist) the model counter-clockwise around the Z axis.
Move up	Zoom in to the model.
Move down	Zoom out from the model.

Mode 3	Move right/left and Move up/down
Move right	Pan the model right.
Move left	Pan the model left.
Move up	Pan the model up.
Move down	Pan the model down.
Mode 4	Slice in/out
Mouse up	Move the cutting plane inside the model.
Mouse down	Move the cutting plane outside the model.
Mode 5	Perspective in/out
Mouse up	Increase lens focal length. When you increase the focal length of the lens, the model appears <i>closer</i> .
Mouse down	Decrease lens focal length.

❖ **NOTE:** You will not get a response in Mode 5 unless you are in Perspective view.

Special keys

When you are using the MATDVIEW command, you can type the following special keys:

- 'C' Re-center the model.
- 'W' Switch to Wireframe display.
- 'H' Switch to Hidden line removal display.
- 'B' Switch to Bordered display.
- 'F' Switch to Flat shaded display.
- 'G' Switch to Gouraud shaded display.
- 'R' Toggle on/off recording of movements (see MATRECORD).

The **Centering** command can be useful after you've panned the model using Mode 3. Switching display options could be useful for accelerating the animation of complex models. Upon completion of MATDVIEW, the model is redrawn using the display option that was current prior to calling MATDVIEW.

Leaving MATDVIEW

You can exit from this dynamic movement mode in two ways:

- By pressing the Right mouse button (or Enter) to end the MATDVIEW command with the ***new viewing point***.
- By pressing Ctrl+C or Esc to end MATDVIEW and get back to the ***original viewpoint***.

In both cases, the temporary horizontal and vertical bands will disappear, but you are still in RD mode.

Exiting RD mode

If you click on the **Exit** button of the 3D menu, you will exit MATDVIEW and RD mode, and be left with the new viewing point.

Lighting effects

The default MATRD lighting has only one light source, of the type **User Directional**. A **User** light doesn't move with the model, the default position is just above and to the right of you, shining down on the model. With this light, the highlights and shaded areas keep changing as you rotate the model, giving the feeling of rotating the model in your hand. Any **Model** lights that you add will rotate with the model, always lighting the same areas. This gives a rather different effect when you rotate the model.

Animation limits

- The animation speed is affected by the number and type of light sources you have. You can get optimal speed by using only one directional light source, with no specular highlights (the default).

MATDWALK



Dynamically walk around in the model

The MATDWALK command, like MATDVIEW, is a dynamic viewing utility. The difference is that with MATDWALK, you appear to walk around inside the model rather than rotating it in your hand. You can initiate this mode by typing *matdwalk*, or by clicking on the footprints icon in the DynaView/3D pop-up menu.

User interface

The MATDWALK command has a user interface identical to the MATDVIEW command:

- Two ‘bands’
- A ‘dead spot’ (you move straight ahead)
- Different ‘Operation modes’ which you can access by pressing the appropriate number key (1 or 2)
- Special keys to switch display options, Center the scene, and Record (same as on previous page)
- An ‘idle’ mode (click the left mouse button)
- A status line which shows the current mode

We recommend that you become familiar with the MATDVIEW command before starting to use MATDWALK. The two MATDWALK Operation modes are listed on the following page.

- ❖ Note that the MATDWALK command only functions in Perspective view. You can use the MATVPOINT command to switch your rendered display from Orthogonal to Perspective view, before entering MATDWALK. (Remember, your viewing distance could change dramatically).

MATDWALK Operation modes

Mode 1: Walk Around (Use '+', '-', and '*' to control speed)

Move right	Turn the viewer right.
Move left	Turn the viewer left.
Move up	Turn the viewer up.
Move down	Turn the viewer down.

Mode 2 : (Perspective in/out)

Mouse up	Increase lens focal length.
Mouse down	Decrease lens focal length.

Setting walking speed

In Mode 1 (the main mode), you set your direction with the mouse, but you must also specify a speed using the '+', '-', or '*' keys.

There are 20 speed levels available (10 forward, 10 backward). The default speed is no movement.

Straight ahead

If you leave the mouse cursor in the dead spot and specify a speed, the viewer moves in a straight line. You can then turn by moving the mouse left and right or up and down.

Pausing and Stopping

To *pause* for a moment ('idle' mode), click the left mouse button; click again to resume movement at the same speed. To *stop* moving and reset the speed to 0, press the '*' key. You can stop and then look around this way.

User Lighting effects

The default MATDWALK lighting has only one light source, of the type **User Directional**. A **User** light doesn't move with model, it moves with *you*; its default position is just above and to the right of you, shining downwards in front of you. With this light, the highlights and shaded areas keep changing as you walk around in the model, giving the feeling of walking with a flashlight fixed just above your right shoulder.

Model lighting effects

Any **Model** lights that you may add (Directional, Positional, or Spot) will remain fixed in place, always lighting the same areas of your model. Model lights can be used to create an atmosphere as you walk through your architectural design.

Tip

Model lights give you a more natural feeling with MATDWALK.

Animation limits

The animation speed is affected by the number and type of light sources you have. You can get optimal speed by using only one Directional light source, with no specular highlighting.

Exiting

You can exit the MATDWALK command by clicking on the **Exit** button of the 3D pop-up menu, or by typing *matexit*. You will return to your normal AutoCAD display, at the same location.

MATRECORD

Record/Playback of walk-throughs

MATRECORD allows you to record a session made in MATDVIEW or MATDWALK and play it back.

You can start this command by typing *matrecord*, or by clicking on the **Rec/Play** button in the 3D menu. Upon invoking the MATRECORD command, the following prompts will appear:

```
MATRECORD  
Open/Close/Play:  
Filename <default_name>:
```

There is no default action; pressing Enter will exit the command.

- The *Filename* prompt applies only to the *Open* and *Play* option.
- The *Close* option closes the current file, since only one file can be opened at a time.

The first time the command is called, the default file name is the name of your current drawing file, but on successive uses, the default is the last file name you used. The default extension for a recording file is *.wlk*.

Opening a file for recording

Opening

When *Open* is selected, the software will first check to see if a file is already open; if so, a warning will be issued and the command ended. If no file is opened, the program will check to see if the specified file already exists. If a file by the same name already exists, the program will ask whether you want to overwrite or append to it before proceeding.

Recording

To start recording, you have to use MATDVIEW and/or MATDWALK along with the 'R' hotkey; a prompt will remind you to do this.

Press 'R' to start/stop recording

In MATDVIEW/MATDWALK, the 'R' hotkey is used to start and stop recording. This can be useful if you want to stop and test a particular movement, or if you want to stop, move to a new location and then continue recording. When you are recording, every new image will be recorded, but not time spent in 'idle' mode (using the left mouse button to stop temporarily).

Recording status

Status messages will be displayed by MATDVIEW and MATDWALK to let you know if you are recording or not.

How to record without moving

If you go to the 'dead spot' while recording is on, and press the '*' key to stop advancing, the time spent there will be recorded such that during playback, the 'standing' time will be observed. You could, alternately, just press the '*' key at any point to reset movement speed to 0, and still be recording.

- ❖ Note that you are allowed to switch display options 'on the fly' with the 'W', 'H', 'B', and 'G' keys.

Many sequences can be made in the same *.wlk* file, with both MATDVIEW and MATDWALK, and in different views (Perspective or Orthogonal for MATDVIEW). Thus a *.wlk* file can be left open for a

whole session, but if it is not closed before you leave AutoCAD, the recording will be lost.

Saving a .wlk file

When *Close* is selected, the software first checks to see if a file was opened. If not, a warning will be issued and the command ended. If nothing was recorded since the *Open* command, a warning will say so, and no file is created. Otherwise the file will be saved, and you can then view it with the *Play* option. (It is not necessary to *Close* a file before playing it.)

Playing back a .WLK file

When *Play* is selected, the software first checks that the current viewport is in RD mode. If not, a warning will be issued and the command will end. If a file is currently opened, it will be played back; otherwise, the software will prompt for a file, check that it exists, and that it is a valid *.wlk* file. If so, the playback will be done. You can interrupt the playback by pressing Ctrl+C.

- ❖ Note that the *.wlk* file does not contain the geometry data, so you have to make sure to play the *.wlk* file that ***matches*** your drawing file. The drawing name is saved in the *.wlk* file, and if you select files that are mismatched, you will be warned of this.

Keep same viewport arrangement

In addition, it is recommended that you have the same viewport arrangement for playback as you had during recording, or else strange aspect ratio effects could be seen.

Returning from playback

At the end of the playback, you are left at the last 'scene' of the *.wlk* file. If you perform a REDRAW, you will be returned to whatever you had on display before you began the playback.

MATLIGHT

Change/Create light sources

With the MATLIGHT command, multiple light sources can be specified for rendering in Gouraud display mode. There are four types of light sources supported: **Ambient**, **Directional**, **Positional**, and **Spot**. There is no limit to the number of light sources you can have. You can start the command by typing *matlight*, or by clicking on the **Lights** button in the DynaView/3D menu

The end result of the lighting you apply to your models is tempered by the **material properties** of the entities that make up the model. We recommend that you learn the lighting controls first, and when you feel comfortable with these controls, you will understand more easily how the material properties interact with the lighting to give you the final result in MATRD, MATDVIEW, and MATDWALK.

Light layer

Light sources are put on a special layer, named through the MATCFG command. The light sources are saved with your drawing, for future rendering. Each light source is created as an AutoCAD 'insert' entity, and most are displayed (in 2D mode only) in the drawing as an icon. (Ambient light and User Directional lights are not displayed as icons.) Once created, light sources can be modified and deleted with either the MATLIGHT command or through standard AutoCAD commands.

The MATLIGHT command prompt displays the following options:

Create/Modify/Delete/List/Update:

These options are described in detail on the following pages.

Create lights option

This option is used to create new light sources. When selecting the option, the following prompt will appear:

Ambient/Directional/Positional/Spot

Ambient light

Description Ambient light does not come from any particular source and has no direction. It is used to insure a minimum level of lighting for every object. Ambient light alone does not produce very realistic images. At least one other type of light source should also be defined. Ambient light is very useful, but you should keep it at a low intensity level.

Parameters When selecting this light type, the following prompts will appear:

Light name : < >
Ambient r,g,b, <0.25,0.25,0.25>:

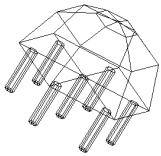
The *Light name* prompt is present only for consistency with other light source types; in fact, only one Ambient light source can be created.

The r,g,b, values are for the red, green, and blue components of the light. Each value must be between 0.0 and 1.0.

Directional light

Description This type of light simulates a light source located at an infinite distance, emitting light rays that are parallel to each other. A good example of a directional light source is the sun.

Two kinds There are two kinds of **Directional** light sources in DynaView/3D: **User** and **Model**. A **User** Directional light source is defined relative to the user, and remains relative to the user. A **Model** Directional light is defined relative to the model, and remains relative to the model.



Directional light
icon

The difference becomes quite obvious when using MATDVIEW ; when you rotate your model, a **Model** light source will keep lighting the same 'area' of the model (it rotates with the model), while a **User** light will light different areas (it stays in the same position, relative to you) .

- A User light source is best for simple lighting of a mechanical part, particularly when MATDVIEW is used extensively.
- Model light sources are more appropriate for architectural models, where MATDWALK is often used.

Model Directional light

Model light Parameters

For a Model Directional light source, the data to provide is:

Light name < >:

Diffuse r,g,b <0.75,0.75,0.75>:

Specular r,g,b <0.0,0.0,0.0>:

Light position <x,y,z>:

Light aim point <x,y,z>:

Diffuse and Specular

The *Diffuse* and *Specular* parameters specify the color and intensity of the light produced by the light source. Your light source can produce diffuse and/or specular light. The diffuse light will be used along with the diffuse reflectivity of the material, while the specular light will be used along with the specular reflectivity of the material. See the MATMATERIAL command for a description of diffuse and specular reflectivity.

The default **intensity** will be (.75, .75, .75). This means that a surface perpendicular to this light source will be rendered at 75% of its diffuse reflectivity. The diffuse reflectivity is set through either MATCFG (for defaults) or MATMATERIAL (for specific materials).

Light position

The *light position* is an X, Y, Z location specified numerically (any mode) or through a mouse pick (in 2D mode only). An icon will be drawn at this location (visible in 2D mode only).

Light aim point

The *light aim point* is an X, Y, Z location, specified numerically or through a mouse pick. It is the point which the directional light faces.

The direction of the light is determined by a straight line going from the *light position* to the *light aim point*.

User Directional light

Parameters For a User Directional light source, the data to provide is:

Light name < >:
Diffuse r,g,b <0.75,0.75,0.75>:
Specular r,g,b <0.0,0.0,0.0>:
Horizontal angle <-43 deg>:
Elevation angle <-44 deg>:

Angles

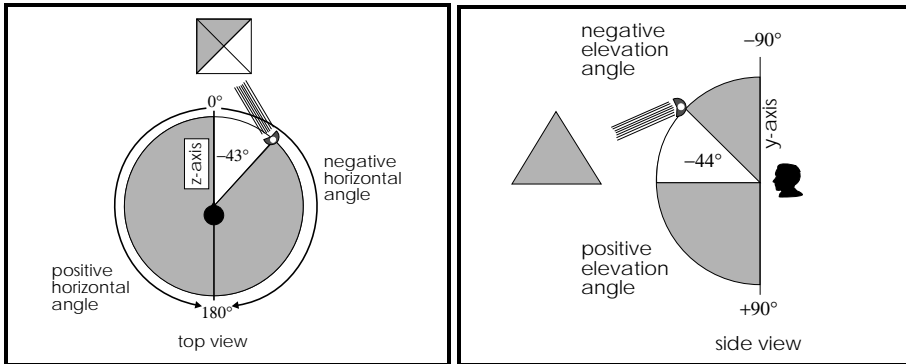
The *Horizontal angle* (default -43°) is the horizontal delta angle between the light direction and the viewer direction. It can range from -179° to +180°.

The *Elevation angle* (default -44°) is the vertical delta angle between the light direction and the viewer direction. It can range from -90° to +90°.

Examples

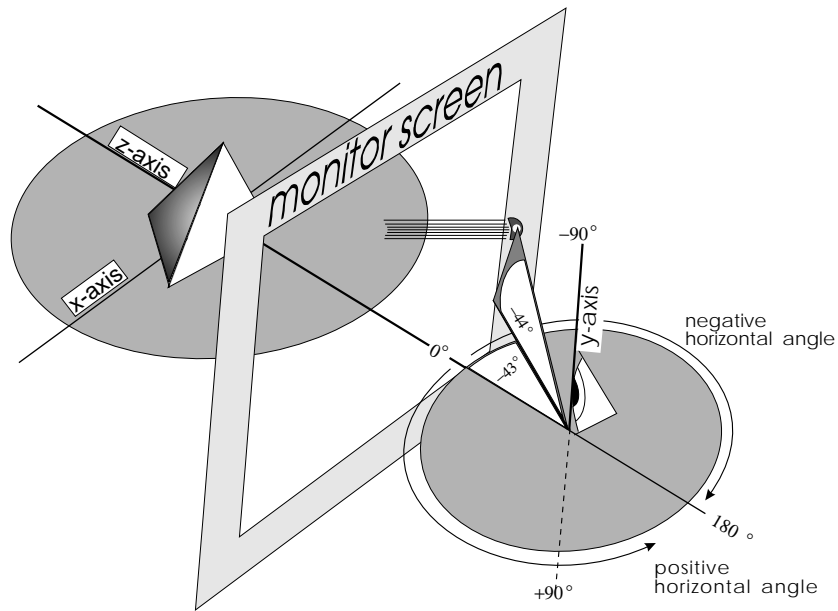
To have a User light source at the viewer position, specify 0° for both angles. To have the light source directly above the viewer, specify a *Horizontal angle* = 0°, and an *Elevation angle* between 0° and -90°.

The default angles have been chosen to insure consistency with the previous version of DynaView/3D. There is no *light aim point* parameter; a User Directional light is always aimed towards the center of your model. The two drawings at the bottom specify the user plane only, from the top, and from the side.



The diagram below shows a User Directional light in 3D space, using

the default values of -43° Horizontal, and -44° Elevation angles.



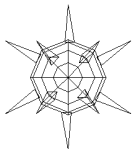
User Directional light horizontal and elevation angles

Positional light

Description A Positional light is much like a light bulb. It is located at a precise point, and radiates light in all directions. There is only one kind of Positional light; it is always a *Model* light.

Parameters For a Positional light source, the data to provide is:

- Light name* < >:
- Diffuse r,g,b* <0.75,0.75,0.75>:
- Specular r,g,b* <0.0,0.0,0.0>:
- Light position* <x,y,z>:
- Reference distance for the residual light* < >:
- Residual light percentage at that distance* <100>:

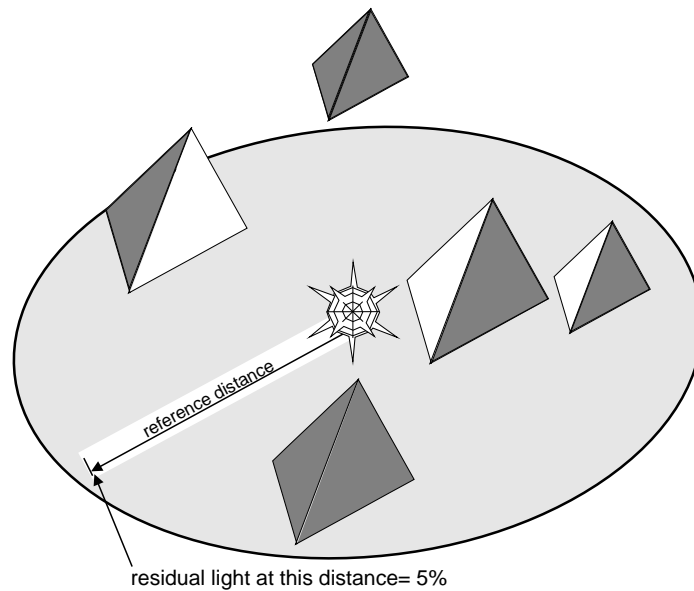


Positional light icon

Light fall-off

It is possible to have a Positional light with or without attenuation (also called 'fall-off'). To provide attenuation, you specify a *Reference distance*, then you specify what percentage (of the original output) of light is left at this distance. The attenuation is calculated linearly from the light position. By default, there is no attenuation, which is indicated by a *Residual light percentage* of 100%.

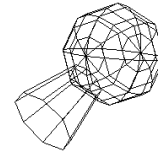
The diagram following illustrates many of the properties of Positional light. In this diagram, the pyramid at the top is beyond the reference distance, and the light at the reference distance is only 5% of the original amount, therefore the pyramid is dark. Note also that the pyramid to the far right is lit, even though it is located behind another object. (All lights pass through faces and do not cast shadows.)



Spot light

This type of light source simulates a spotlight. Like a Positional light source, it is located at a finite distance, and it can have attenuation properties. But unlike a Positional light, it radiates light only in one direction, within an area represented by a cone.

There is only one kind of Spot light; it is always a **Model** light.



Spot light icon

Parameters The Spot light prompts for data are:

Light name < >:
Diffuse r,g,b <0.75,0.75,0.75>:
Specular r,g,b <0.0,0.0,0.0>:
Light position <x,y,z>:
Light aim point <x,y,z>:
Residual light percentage at the aim point <100>:
Cone aperture angle <45>:
Cone aperture fall-off angle <55>:
Light distribution for fall-off region (Linear, Square, Cubic)
<Linear>:

Light fall-off

By default, a Spot light has no distance attenuation. This is indicated by a *Residual light percentage at the aim point* of 100 %. You can specify a value between 0 and 100 for this parameter. The attenuation is calculated *linearly* (explained later) from the light position.

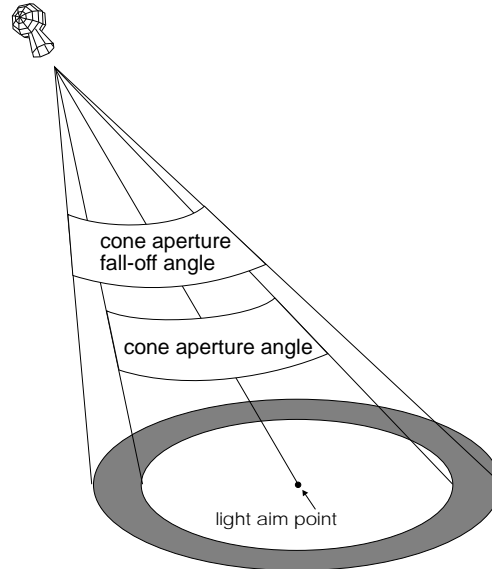
Cone aperture angle The *cone aperture angle* specifies the angle in which the light is at its maximum intensity. The default value of 45° means that the area within 22.5° on each side of the light direction (light aim point) is lit at maximum intensity.

Cone aperture fall-off angle The *cone aperture fall-off angle* specifies the total angle in which any light is present. The default value of 55° means that outside the area 27.5°, on each side of the light direction, no light is received.

Light distribution The *light distribution for fall-off region* parameter specifies how the light attenuation is made between the outer edge of the cone aperture angle and the outer edge of the fall-off angle. The default values define a fall-off region of 5°, in which the fall-off can be *linear*, *square* or *cubic*. *Linear* fall-off is the most gradual, *square* is less gradual, and *cubic* fall-off is the ‘fastest’.

Spotlight tip

If the *cone aperture angle* is equal to the *cone aperture fall-off angle*, the *light distribution* parameter is irrelevant, but the circumference of your 'spot' of light will look very jagged, unless you have very fine meshing. Therefore, to produce a very concentrated, smooth-edged spotlight, we recommend you make the *fall-off angle* a little bit bigger than the *cone aperture angle*, and use *cubic* fall-off.

**Modify lights option:**

This option is used to modify existing light sources. When choosing this option, you must first select one or more lights, by either picking, or naming them. Remember you can't pick in RD mode.

Selecting

The prompts for light selection are:

Selecting light sources by <Pick> or <Name>:

Pick method If you choose *Pick* (by typing *P*), you will enter into the usual AutoCAD selection mechanism. Remember that you can't pick in RD mode:

Select objects:

At this point, you can pick one light source icon at a time, or select by 'corner' or 'window'. You press Enter to terminate the selection set.

Name method If you choose to select by *Name* (by typing 'N'), you simply have to type in the light source name at the prompt:

Light name:

You can enter many light names, one at a time, pressing Enter after each name. ***Selection is terminated by pressing Enter twice.***

Ending selection

You will return to the *Selecting light sources* prompt, and you end the selection process by pressing Enter once more. At this point the MATLIGHT command will ask you to re-input the data for all the selected light sources. Refer to the *Create* option, described previously, for the list of prompts you will get.

Notes

- User Directional lights and Ambient lighting have no icons associated to them, so you must select them by name.
- To get the list of light names, you can use the *List* option of MATLIGHT, described on the following pages.
- When modifying light sources, it is not necessary to call MATRD again. Just perform a REDRAW of the RD viewport.

Delete lights option

This option is used to delete light sources. When choosing this option, you must first select one or more lights, by either picking or naming, the same way as with the modify option. When deleting light sources, it is not necessary to call MATRD again; just perform a REDRAW of the RD viewport.

List lights option

This option will give you a list of the light source names, types, and data.

Update lights option

Update active layer

When light sources are modified through **AutoCAD** commands, the 3D Module does not immediately get notified of the change. You must use the *Update* option to see the effects of the changes at the next REDRAW. Alternately, you can call MATRD again, but this solution usually takes longer.

Neither the *Update* option nor a new MATRD is necessary if light sources are added or modified using any of the MATLIGHT options. You will see the effect of the lighting changes at the next REDRAW.

Hints and Tips on lighting

1. Default lighting

DynaView/3D uses a simple default lighting which is a good starting point for mechanical parts models. The default lighting is:

- Ambient light:
Name: DEFAULT_LIGHT01
Ambient r,g,b: 0.25,0.25,0.25
- User Directional light:
Name: DEFAULT_LIGHT03
Diffuse r,g,b: 0.75,0.75,0.75
Specular r,g,b: 0.0,0.0,0.0
Horizontal angle: -43°
Elevation angle: -44°

With these defaults, surfaces perpendicular to the Directional light source will be lit at 100% of their diffuse reflectivity (0.25 Ambient + 0.75 Directional light). Any other non-perpendicular surfaces will be lit between 25% and 100% of their diffuse reflectivity.

- ❖ Note that when you begin adding light sources, you lose the default lighting setup.

2. Specular highlights

Note that all the default light sources do not have any specular light values. Therefore, make sure to add some specular component if you want to produce specular highlights.

3. Material properties

Remember that all lighting intensities (whether ambient, diffuse or specular) are always multiplied by the object's *material properties*. If the material properties for an object say that it does not reflect a particular type of lighting, the effect of that lighting will be null. See the MATMATERIAL command for more details.

4. Ambient light

The Ambient lighting reduces the range of intensities left for other light sources. As stated earlier, Ambient light should be left at a low level.

5. Highlighting

To better see the effect of Spot lights and specular highlights, make sure that your surfaces are tessellated in small enough faces:

- For 3D surfaces and objects, this means specifying a fairly fine meshing at creation time.
- For entities such as arcs, extruded arcs, and extruded splines, the number of faces can be controlled through MATCFG.
- For an entity such as a large plane, finer tessellation must be performed 'manually'.

6. Animation speed

The speed of redrawing the model is affected by the number and complexity of light sources. For best speed, during MATDVIEW and MATDWALK, it is better to limit yourself to Ambient lighting and a couple of Directional lights without specularly.

- ❖ As already mentioned, when you add new light sources, it is not necessary to call MATRD again; a REDRAW of the RD viewport will be sufficient.

7. Multiple lights

When the resulting color of an entity is calculated, the effect of the addition of each light (including Ambient light) is made. If the resulting R,G,B components total more than 1.0 in intensity, you have over-saturation, so the intensity is set back to 1.0. This can happen quite rapidly when you start adding many light sources. In general, when you have more lights, the intensities of each of them should be smaller.

Different lighting situations can be created by making a few different light source layers for your drawing. Then you can use MATCFG to swap between the different layers. MATCFG can also be used to switch to the default lighting, simply by specifying an invalid layer name (such as NONE). Also, if a light layer is 'frozen' it becomes inactive, and the default lighting setup is used.

MATMATERIAL

Defining material properties

With the MATMATERIAL command, you can define specific material properties for your model, on an entity per entity basis. To get the results you expect, all your light sources should be emitting white light, at least while you're fine tuning the colors of your materials. In particular, this command will let you set:

- Ambient reflectivity
- Diffuse reflectivity
- Specular reflectivity, and highlight area size
- Emission properties

Diffuse reflection

Surfaces such as matte-painted walls exhibit diffuse reflection. Light hitting totally diffuse surfaces is reflected equally in all directions. No matter where your viewpoint is, the reflection of the surface is the same, so the surface looks uniformly bright.

Specular reflection

Unlike diffuse surfaces, specular surfaces reflect light in a narrow cone, creating a shiny spot. Light hitting a perfectly specular surface, such as a mirror, is reflected in one direction only.

For specular reflection, the rendering module must take into account the angle of the surface to both the light source *and* your viewpoint.

Therefore a specular reflection depends not only on the light source position, but also on your viewing position. With an object such as a pool ball, as your viewing position changes, the location and shape of the shiny spot will also change.

specular coefficient

You can control the size of the specular reflection spot with the *Specular coefficient*. This parameter could be visualized as a 'surface roughness' factor; the more the surface is polished, the smaller (and brighter) the spot will be.

Using the MATMATERIAL command

First, you must not be in RD mode. You can start this command by typing *matmaterial*, or by clicking on the **Materials** button in the DynaView/3D pop-up menu. You are first prompted to make an entity selection, using the regular AutoCAD selection mechanism:

Select objects:

If you selected more than one entity, you will see:

Work on individual entities ? <Yes>

Say *No*, or you will work line by line. Next, you get the main prompt:

Change/Reset/<?>:

List If you press Enter to select '?', you will get a list of the material properties associated with the selected entities.

Reset The *Reset* option will erase the material properties associated with the selected entities.

Change The *Change* option allows you to modify the material properties of the selected entities.

When you select *Change*, the following series of prompts will appear:

Material name <>:

Ambient r,g,b <AutoCAD color>:

Diffuse r,g,b <AutoCAD color>:

Emission r,g,b <0.0,0.0,0.0>:

Specular r,g,b <AutoCAD color>:

Specular coefficient <0.5>:

Parameter descriptions

Material name

The *Material name* is simply a text string to help you identify the material you want to represent. It can be left empty if you wish. This name can be retrieved by the LISP version of MATMATERIAL (see 'AutoLISP commands', page 4-50) as a material ID, in a LISP-based home-made material library you could develop.

Ambient reflectivity

The *Ambient r,g,b* parameter determines how an entity reacts to Ambient light. It is a red, green, and blue set of values that can range from 0 (no intensity) to 1 (full intensity).

Example

A value of [0,0,0] means that the entity does not reflect Ambient light. A value of [1,0,0] means that only the red component of your Ambient light is reflected.

Of course, if you don't define any Ambient light, the ambient reflectivity set here is meaningless, and will not affect the image. The *default values* for *Ambient r,g,b* are the R,G,B intensities of the AutoCAD color of that entity. Unlike diffuse and specular, ambient reflectivity has no real physical correspondence, and is provided only for more flexibility. *In most cases, you should leave ambient reflectivity at the default setting.*

Diffuse reflectivity

The *Diffuse r,g,b* parameter determines how much an object will reflect the diffuse light produced by your Directional, Positional and Spot lights (see MATLIGHT command). An object can reflect red, green and blue in different amounts; this is what will determine an object's color under white light. The *Diffuse r,g,b* parameter is a set of red, green, and blue values ranging from 0 to 1.

Example

A value of [0,0,0] means that the entity does not reflect diffuse light. A value of [1,0,0] means that only the red component of your diffuse light will be reflected.

Of course, if you do not define any diffuse light in your light sources, the diffuse reflectivity set here is meaningless, and will not affect the image. The **default values** for diffuse reflectivity are the R,G,B intensities of the AutoCAD color of that entity.

Emission

The *Emission r,g,b* parameter determines if the entity, in addition to (or instead of) reflecting light produced by your light sources, does produce light itself. You could use this quality to simulate neon lights, light bulbs, or status lights on a control panel for example. Note that an object possessing this quality will glow, but does not throw light onto other objects nearby. The *Emission r,g,b* parameter is also a set of red, green, and blue values ranging from 0 to 1.

Example

A material having an emission value of [0,0,1] would glow blue even when there was no light shining on it. The **default values** for *Emission* are [0,0,0] (no emission). This is probably a parameter that you will not use very often.

Specular reflectivity

The *Specular r,g,b* parameter determines how much an object will reflect the specular light produced by Directional, Positional, and Spot lights (see MATLIGHT command). The object can reflect red, green and blue at different levels; this is what determines the color of the specular highlights. The *Specular r,g,b* parameter is also a set of red, green, and blue values ranging from 0 to 1.

Example

A value of [0,0,0] means that the entity does not reflect specular light. A value of [1,0,0] means that only the red component of the specular light is reflected; if you were lighting this object with white specular light, the specular highlights would be red.

Notes

- Of course, if you do not define any specular light in your light sources, the specular reflectivity property is meaningless, and will not affect the image.
- Remember, the default specular light in MATLIGHT is indeed [0,0,0]. The default values for *Specular r,g,b* are the R,G,B intensities of the AutoCAD color of the entity.
- Nothing stops you from setting non-zero values for both diffuse and specular reflectivity at the same time. In fact, this is just normal. It is also possible to have different colors for diffuse and specular reflectivity (for instance red for diffuse and white for specular, in order to get a red surface with white highlights).

Specular coefficient

The specular reflectivity data also includes a *Specular coefficient*, which can range in value from 0 to 1. As described at the beginning of this section, this coefficient controls the *size* of the specular ‘spot’, or *highlight* produced on the object.

Examples

To produce the smallest possible ‘spot’, you need to set the specular coefficient value close to 1. A value of 0.3 will give you a highlight area larger than a value of 0.5. The default value for *Specular coefficient* is 0.5.

MATMATERIAL Notes

Unlike light sources, which can be set and changed freely before or after a MATRD, material properties must be set *before* calling MATRD. *Every time MATMATERIAL is used, MATRD will have to be called again to see the effect of the changes.*

Stored as extended data

On subsequent use of the same drawing, any material properties that you set will be re-used, since the material properties are stored in the drawing as ADS extended data, and saved for your next session.

Summary

The MATMATERIAL command can make your model look more realistic, by allowing you to:

- Overwrite the AutoCAD color
- Differentiate matte and shiny surfaces

Suggested Uses

- By default, DynaView/3D shades entities using their AutoCAD colors. For drafting purposes, you may have drawn many different entities made of the same material (for instance steel) in different colors. By using MATMATERIAL, you can have all these entities (that are made of the same material) rendered with the same color.
- By default, all surfaces are matte. Using the MATMATERIAL and MATLIGHT *Specular r,g,b* parameters, you can make shiny objects.

If you don't use the MATMATERIAL command, all entities will react as you would expect from AutoCAD. This is due to the default material properties specified in MATCFG. See the MATCFG command for more details.

MATIMAGE

Saving a rendered viewport to a .bmp file

MATIMAGE allows you to save the contents of a rendered viewport into a file. The file format supported is a Windows *.bmp* file.

You can invoke the command by typing *matimage* or by clicking on the **Save Image** button in the DynaView/3D pop-up menu. The following prompts will appear:

```
MATIMAGE  
full Screen or current Viewport <Viewport>:  
Filename <default_name>:
```

There is no 'Quit', option; you can press Ctrl+C if you need to exit the command.

- The first prompt lets you save the whole screen (including menu areas) or only the content of the current viewport.
- The default name for the file will be the drawing name. The default extension is *.bmp*. After you enter a name for the output file, the software will check to see if the file already exists. If so, you will be asked to confirm before the file is overwritten.
- ❖ Note that the *.bmp* file created is a 24-bit (true color) file. Depending on the resolution and viewport size, the file can be close to 3 Mbytes in size.

MATCFG

Configuring the rendering mode

The MATCFG command changes the defaults for the DynaView/3D commands. You invoke this command by typing *matcfg*, or by clicking on the **Config** button in the DynaView/3D pop-up menu. The following prompt appears:

~/MaterialDefaults/LightLayer/DisplayOption/Flags/BackgroundColor:

If you press '?', you will see a list of the current settings.

Where settings are saved

The data you input with MATCFG is kept in a special AutoCAD block. When you save your drawing these defaults are saved with your drawing.

The options perform the following functions:

Material Defaults

Material defaults determine the default colors of your objects when they are rendered. The original setting gives you the same AutoCAD colors for your objects when they are rendered.

Important

The material defaults information you input here defines the **default** material reflectivity for entities rendered in Gouraud shading mode. You can, in addition, use the MATMATERIAL command to create specific materials (such as steel, plastic, and even neon lights) with certain colors and qualities (see the MATMATERIAL command description on page 4-33 for additional details). The MATMATERIAL properties assigned to an object will *always* override the material defaults settings.

Summary

In summary, the *Material Defaults* are used to assign a color to entities which have no assigned material properties (through MATMATERIAL).

☞ ***This being said, when we speak of all objects within this section on material defaults, you must understand that we are speaking of all objects that do not have assigned material properties. We will not repeat this in order to keep the explanations simpler.***

When you select the *MaterialDefaults* option (by typing *m*), you will get the following series of prompts:

Define colors as Absolute/Percentage: <Percentage>
Ambient r,g,b <1.0,1.0,1.0>:
Diffuse r,g,b, <1.0,1.0,1.0>:
Emission r,g,b, <0.0,0.0,0.0>:
Specular r,g,b, <1.0,1.0,1.0>:
Specular coefficient <0.5>:

Parameter descriptions

Absolute/Percentage

The *Absolute/Percentage* parameter gives you greater flexibility in choosing the way colors will be determined, as follows:

Absolute color

- If the *Absolute* color method is selected, the driver will directly use the *r,g,b* values you specify in the subsequent prompts for *Ambient*, *Diffuse*, *Emission*, and *Specular* properties. The values you input must be in the range of $0.0 \leq RGB \leq 1.0$. All of the entities will be rendered with the same color.

Percentage color

- If the *Percentage* method is selected, MATRD will first take the AutoCAD color of each entity, and then multiply it by the *r,g,b* values you specify in the subsequent prompts, for *Ambient*, *Diffuse*, *Emission*, and *Specular* properties. In this case, the entities rendered would ***normally*** keep their original colors (example 1), only their brightness would be altered.

Percentage method Examples

1. If $r = g = b$, and the *r,g,b* values are less than 1.0, this gives you the effect of varying the brightness of your scene. The values you input must be in the range of $0.0 \leq RGB \leq 1.0$.

2. If $r = g = 0, b = 1$, this gives you the effect of looking at your scene through a blue 'filter'.

- ❖ In most cases, if you use the *Percentage* method, the *Ambient* and *Diffuse* r,g,b values should be kept to 1.0, which will give you rendered entities in shades of the AutoCAD colors you drew them with. This is the default setting.

You select *Absolute* color by typing a , or *Percentage* by typing p . If you just press Enter, you will get the default method, which is *Percentage*.

Ambient reflectivity

The second prompt is:

Ambient r,g,b: <1.0,1.0,1.0>

Description

The *Ambient* reflectivity determines how objects reflect Ambient light.

The default Ambient light is $[.25,.25,.25]$, which means that an entity with $[1.0,1.0,1.0]$ Ambient reflectivity, which receives **only** Ambient light will have an intensity of $[.25,.25,.25]$.

You can change these values by entering new values, or accept the default by pressing Enter. The values you input must be in the range of $0.0 \leq RGB \leq 1.0$.

- ❖ Note that if there is no Ambient light at all, the Ambient reflectivity parameter is meaningless, and will not affect the rendered image.

Tip

If you are using the *Percentage* color method this parameter should normally be left at its $[1.0,1.0,1.0]$ default.

Diffuse reflectivity

The third prompt is:

Diffuse r,g,b: <1.0,1.0,1.0>

Description The *Diffuse r,g,b* reflectivity parameter determines how much an object reflects diffuse light. An object can reflect red, green, and blue at different levels; this is what determines the color of an object bathed in white light.

The default values for the *Diffuse r,g,b* parameter are [1.0,1.0,1.0]. You can change these values by entering new values, or accept the default by pressing Enter. The values you input must be in the range of $0.0 \leq RGB \leq 1.0$.

Tip If you are using the *Percentage* color method this parameter should normally be left at its [1.0,1.0,1.0] default setting.

Emission

The fourth prompt is:

Emission r,g,b: <0.0,0.0,0.0>

Description The *Emission r,g,b* parameter determines if a material produces light itself. Note that this parameter will make a material appear to ‘glow’ but it will not throw light onto other objects nearby. This means that you can’t use the emission parameter to make a material behave like a light source. It is a parameter you probably won’t use very often.

The default values are [0.0,0.0,0.0], which leaves default materials with no emission properties. You can change these values by entering new values, or accept the default by pressing Enter. The values you input must be in the range of $0.0 \leq RGB \leq 1.0$.

Specular reflectivity

The fifth prompt is:

Specular r,g,b, <1.0,1.0,1.0>:

Description The *Specular r,g,b* parameter determines how much a default material will reflect specular light. A material can reflect red, green, and blue light at different levels, this is what determines the color of the specular highlights. You can read more about this parameter on page 4-36.

The default values are [1.0,1.0,1.0], which means that materials will reflect all specular light, if there is any. You can change these values by

entering new values, or accept the default by pressing Enter. The values you input must be in the range of $0.0 \leq RGB \leq 1.0$.

Tip

If you are using the *Percentage* color method, this parameter should normally be left at its default setting.

Specular coefficient

The last prompt for material defaults is:

Specular coefficient <0.5>:

Description

The *Specular coefficient* parameter controls the size of the specular highlight 'spot' on a surface. This parameter is described in detail on page 4-28 . The default setting is for 0.5; you can change it by entering a new value, or accept it by pressing Enter. The values you input must be in the range of $0.0 \leq RGB \leq 1.0$. A value of 1.0 gives you the smallest spot. A value of 0.0 (not recommended) will give you the largest specular highlight.

Light Layer

This MATCFG option defines the layer used by DynaView/3D to store and read light sources. Using this option, you can create and save alternate lighting situations for your model in rendering mode. See the MATLIGHT command on page 4-32 for more information.

Display Option

This MATCFG option retains the last display option defined through the MATDISPLAY command. This display option will be used the next time MATRD is issued. At start-up, the display option is set to Gouraud. The choices are explained in the MATDISPLAY command.

Background Color

This MATCFG option controls the background color of a viewport in RD mode. The default color is black. You can change it by entering new red, green, and blue intensities. A new MATRD is not necessary to view this change; using the AutoCAD REDRAW command will update the viewport immediately.

Flags

When selecting the *MATCFG Flags* option, the following additional parameters can be set:

ArcSeg/SplineSeg/FullGouraud/Nesting/ThresholdAngle/SelectMode

To select an option, type the first one or two uppercase letters of the flag name, as it is shown in the prompt.

ArcSeg

This parameter controls the minimum and maximum number of faces to be created along the circumference of a circle or an arc.

The maximum value represents the number of faces generated for an arc having an included angle of 360° . The number of faces created for an arc will always be:

$$MIN \leq nfaces \leq MAX$$

Defaults

The *default settings* for this parameter are $MAX = 24$, $MIN = 6$. If you are not satisfied with the rendering of cylinders, change this parameter and call *MATRD* again.

- ❖ Note that for meshes, the number of faces is determined when the surface is created by AutoCAD. If you are not satisfied with the rendering, you must change the *SURFTAB1*, *SURFTAB2*, *SURFU* and/or *SURFV* AutoCAD variables, recreate the surfaces, then call *MATRD* again.

SplineSeg

When you render a spline which is approximated by arcs, a large number of faces might be created. Selecting the *SplineSeg* flag will use straight segments to draw the spline. The **default setting** for this parameter is *Yes*.

FullGouraud

When *FullGouraud* is enabled, even independent entities such as 3D faces will be sorted by *MATRD*, in order to come up with a smooth look. In this case, faces are accumulated and stored in trees. When every entity

is processed, the program traverses the trees, averaging the normals. This can result in a large number of 3D faces, and slow down the rendering process. The **default setting** for this parameter is Yes. Enabling this option can make a big difference (with Gouraud shading) in the appearance of older AutoCAD drawings, where surfaces are not shaded as smoothly as you expected.

Nesting

The *Nesting* flag controls the nesting level when sorting 3D faces. Sorting can be done on global, block, and sub-block bases. The **default setting** for this parameter is 1. The nesting levels (choices) are:

- 0 All 3D faces.
- 1 Global (3D faces not in blocks) and blocks.
- 2 Global (3D faces not in blocks), blocks, and sub-blocks.

Threshold Angle

To achieve a smooth surface in Gouraud shading, the normals between adjacent faces must be averaged. However, with some objects (cubes, for example) averaging the normals is undesirable and would result in cubes with rounded edges. To minimize this effect, a *Threshold angle* is used in the following manner:

- If a face 'A' is adjacent to face 'B', the angle between the normals of the faces is calculated. The calculated angle is compared with the threshold angle, and if it is smaller, the normals are averaged.

The **default setting** for this parameter is 35°.

SelectMode

This option specifies how MATRD selects entities. It can be set so that all entities are automatically selected, or you can manually make a selection-set. The **default setting** for this parameter is *Yes* (to select all entities automatically).

MATEXIT

Leaving RD mode

Apply current viewpoint

To leave RD mode, you should issue the Matrox MATEXIT command from the desired viewport. You can do this by typing *matexit* or by clicking on the **Exit** button in the DynaView/3D pop-up menu. That viewport will be cleared, then a REGEN in normal AutoCAD wireframe mode will occur. *You will keep the same active viewpoint you had* while you were using the MATVPOINT, MATDVIEW, or MATDWALK commands.

Abandon current viewpoint

- Alternately, you can quit RD mode by issuing an AutoCAD REGEN command. The effect will be the same, except that the viewpoint will be as it was *before* you entered RD mode.
- You can also quit by issuing the DynaView MFLUSH command. This command works a bit faster than calling a REGEN.

Redraw without exiting

- ❖ Note that using the AutoCAD REDRAW command simply causes a redraw of the viewport without changing the display mode.

Unloading the 3D command set

If you don't need it anymore, it is possible, but not necessary, to remove the 3D command set from memory. This may help the AutoCAD overall speed.

Manual unload

To unload the 3D command set manually type the following command line exactly as shown:

```
(xunload "/mga/acadv/dv3dads")
```

Using the pop-up menu

To unload the 3D command set through the DynaView/3D pop-up menu, simply click on the **Unload** button.

Using the pull-down or side bar menu

If you used one of the Matrox *acad###.mnu* or *add-on.mnu* files to install the pull-down or the side bar menu, you can simply click on the UNLOAD command in that menu.

- ❖ If you installed the driver in a subdirectory different than the default one (*c:\mga\acadv*), you will have to change four lines in the *acad.mnu* file. Open the file with an ASCII text editor, search for the phrase 'MGA' and replace each occurrence of this phrase with the path to where you installed the driver files.

About Rendering mode

Using AutoCAD commands in RD mode

Rendering Mode (RD mode) is a special mode within AutoCAD and is only intended for display purposes, therefore certain AutoCAD or Matrox commands can't be used in this mode. This section explains which commands you may and may not use in RD mode.

Updating the RD viewport

An RD mode viewport is not automatically updated with AutoCAD wire-frame entities (even if you're rendering in 'wireframe' display mode). Any entity that is added, deleted, or moved by the result of some action in a regular AutoCAD viewport will not be automatically updated in the RD viewports. To update the RD viewport, you must run the MATRD command again. This can be time-consuming with very complex drawings.

- Picking is not allowed in an RD viewport. You should not try to pick entities (in most cases you would not get the desired entity, because of the different nature of the display).
- You should not use any AutoCAD display commands while in RD mode for the following reasons:
 - Many commands (ZOOM, for instance) are invalid in Perspective mode.
 - Even in Orthogonal mode, a 'zoom window', which would normally be handled by the display list, will be ignored.
 - Other commands which may force a REGEN, such as ZOOM Extents, will pull you out of RD mode.

No 2D commands in RD mode

As is the case for AutoCAD display commands, the DynaView/2D commands (Bird's Eye, SpyGlass, Scroll Bars) can't be used within an RD viewport – they are simply ignored.

Matrox AutoLISP commands

Command list

To facilitate the development of AutoLISP applications running in RD mode, Matrox also provides commands which can be called through an AutoLISP program.

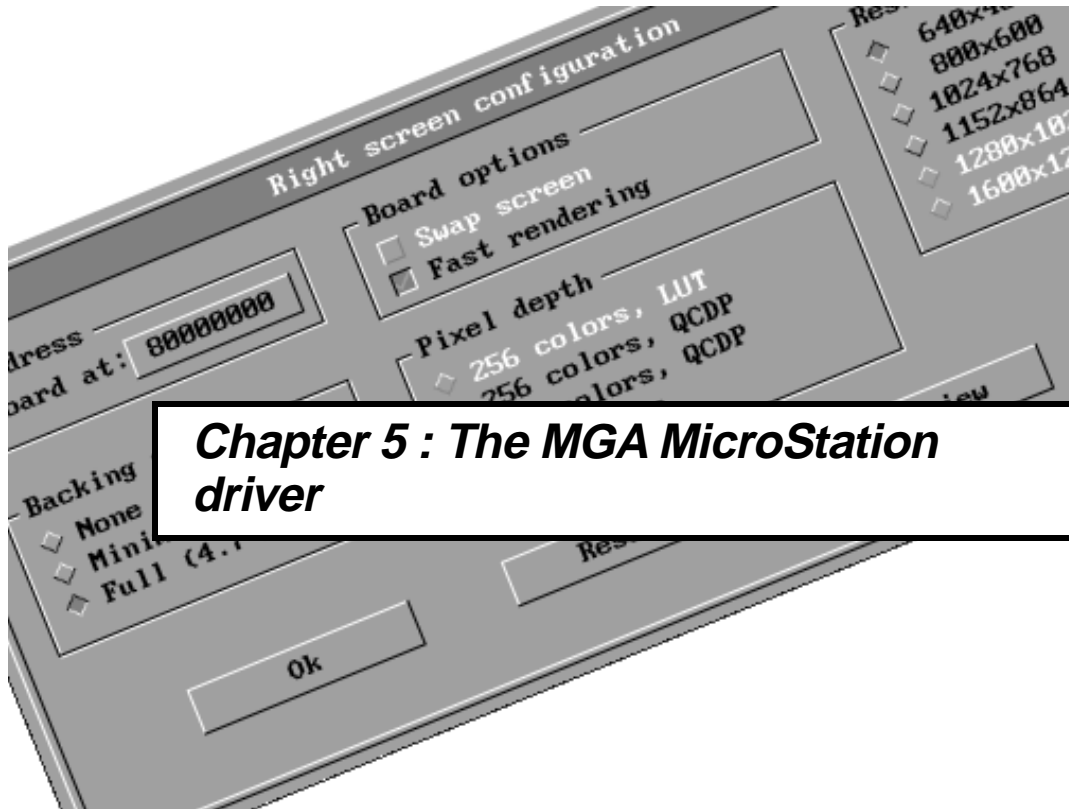
A sample AutoLISP program, called *walk.lsp*, is included on the driver diskette. The functions included in the program will show you how to create very powerful walk throughs. We hope you will find this program a useful starting point for developing your own special AutoLISP functions.

These commands allow you to develop more elaborate Lisp programs. Some potential applications are:

- A program that swaps between light source layers during animation.
- A material database program (using the *ma* material descriptor – see MATMATERIAL in this section).

The Matrox AutoLISP commands are listed in the table below. The description of these commands can be found in the *dynaview.doc* file in your `\mga\acadriv` directory.

Name	Description
MATRDINQ	Inquire if a viewport is in rendering mode.
MATDISPLAY	Change the display option used for rendering.
MATVPOINT	Change or request the viewing parameters of a rendered viewport.
MATDBLBUF	Select double-buffering mode
MATLIGHT	Change or request lighting parameters.
MATMATERIAL	Change or request material properties parameters.
MATCFG	Change or request configuration parameters.



Chapter 5 : The MGA MicroStation driver

Driver features

The MGA MicroStation 5 driver features:

- Support for MicroStation 5.0 and MicroStation 95.
- 8, 16, and 24 bit-per-pixel color modes.
- Supports single screen (with or without swap screen), dual screen and **Windows Connection** (single or dual screen). The Windows Connection for single screen uses only the MGA Windows driver. The Windows Connection for dual screen uses the MGA MicroStation driver for one screen, and the MGA Windows driver for the other screen.
- Window management. The window management is handled by the driver, which makes copies of overlapping windows. Those copies (backing store) are then used to restore the view contents without delay.
- Hardware assisted 3D rendering. With the MicroStation 5.0 API, the driver can receive untransformed elements (lines, quad and triangles). This feature allows the driver to transform and render those elements using Matrox SXCI software 3D engine and the Matrox 3D hardware. This makes rendering at least two times faster for views rendered in Hidden Line, Filled Hidden Line, Constant, and Smooth shading.
- Dynamic Link Module (DLM). The window manager and accelerated 3D features are implemented with the new DLM technology of MicroStation 5.0 and up.
- A configuration interface that is easy to use.

Configuring MicroStation PC

To configure the MicroStation PC driver, take these steps:

1. Make the MicroStation directory current by entering:

```
cd \ustation
```

Start the configuration program, by entering:

```
usconfig
```

2. Select **Generate for DOS** if you will be using MicroStation under DOS. Select **Generate for Windows Connection** if you will be using MicroStation under Windows.
3. From the list presented to you, select:

```
DISPLAY ADAPTERS
```

4. You can use an Matrox board in single or dual screen configuration. In dual screen configuration, the MGA can be either the 'right' or 'left' screen; the other display adapter can be another Matrox board, a VGA board or another high-resolution adapter. The configuration is done separately for each Matrox board when the driver is started. Now skip ahead to the subsection that's appropriate for the number of Matrox boards and monitors in your system

Single screen configuration

1. From the list of Graphics adapters, select **Matrox MGA, v1.10** for the Right screen, and **No Second adapter** for the Left screen.
2. You must now decide if you want to use the MicroStation virtual graphics page swap screen. The prompt will ask:

```
Do you want to disable the virtual graphics page? [Y,N]
```

Type 'n' to enable swapping. Swap screen is supported only in 8-bit mode. In this case only 16 colors will be available for each page.

- You can select 'n', but still have the option of disabling the swap screen feature in the MGA driver configuration menu at a later time.
3. The *usconfig* program will show you the **Current MicroStation Configuration**. If it is satisfactory, select **Exit and Save**.

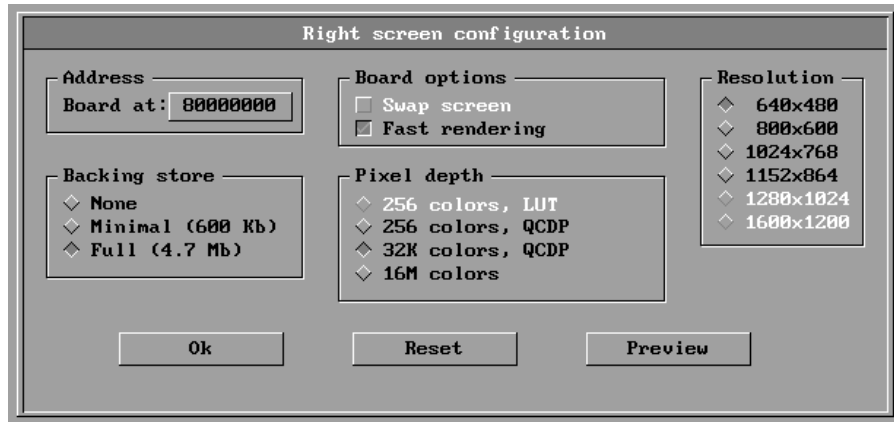
Dual-screen with two boards

Follow this procedure if you have two monitors and want to use two Matrox boards or one Matrox and another graphics adapter with the MicroStation PC program. If you have only one Matrox board, this procedure assumes that you want to select MGA for the **Right** screen, but it is also possible to select VGA for the **Right** screen and MGA for the **Left** screen.

1. From the list of Graphics adapters, select **Matrox MGA, v1.10** for the Right screen.
2. Next, you can select your second Matrox board or another graphics adapter for the **Left** screen.
3. The *usconfig* program will show your **Current MicroStation Configuration**. If all the settings are satisfactory, select **Exit and Save**.

MGA driver configuration

Now you must configure each Matrox board in your system. Start MicroStation and the configuration menu will appear. The configuration starts with the Right screen first. You can use the mouse or keyboard (Tab, Shift-Tab, and arrows) to make your selections. After you have made your choices, you can click on the **Preview** button to see the menu at the resolution you have selected. Pressing the **Reset** button will return the settings to what they were at the last configuration you set. When you click on **Ok**, the MicroStation program starts. The following subsection details your configuration options.



Configuration Menu options

To properly configure each board you have to set these parameters:

- Board address: You can modify this only if you have multiple Matrox boards in your system. This is to allow you to swap the MicroStation Left and Right screens by software (instead of swapping the monitor cables).
- Board options:
 - swap screen: Enables the swap screen feature, which is available only in single screen with 256 colors. This item is selectable only in a single screen configuration, when you have already answered (in MicroStation PC configuration) 'n' to the question: *Do you want to disable the virtual graphics page? [Y,N]*
 - Fast rendering: Enables hardware assisted 3D rendering. The MGA driver will accelerate the rendering of MicroStation views in the following modes: Hidden Line, Filled Hidden Line, Constant Shading, and Smooth Shading. Note that texture mapping is not supported in Constant Shading and Smooth Shading.
- Resolution: up to 1600 × 1200, depending on the model and pixel depth selected.

- Backing store: This option sets the amount of memory to use for view backstore.
 - None: Backing store is disabled.
 - Minimal: Backing store is enabled for the size of the screen (would handle up to 8 tiled views).
 - Full: Backing store is fully enabled (would support up to 8 views in cascade)
- Pixel depth: Sets the number of colors available, up to 16 Million colors.

Windows Connection

When using the Windows Connection with one board, the configuration is straightforward. You just select the Windows Connection as the display adapter for the right screen in the *usconfig* program, and you're done. In this case, the MGA MicroStation driver is not used. To obtain the best speed make sure that you are using an MGA Windows driver.

If you have one Matrox board and any other VGA adapter, there should be no conflict between Windows and MicroStation. Just select a different driver for each screen and you're done.

If you have two Matrox boards in your system and wish to use Windows Connection for one of them, please take note of those important points:

- The MGA Windows driver will use the board with VGA enabled. Refer to the *Dual Screens* section of the MicroStation manual *Introducing MicroStation PC*, page 6-13 for more information.
- Under **Windows Connection**, the MicroStation driver will always start on the first board found *without* VGA enabled.
- To configure the MicroStation driver, you must activate, in Windows, the 'MicroStation Text Window'. You must do this to enable the keyboard and the mouse pointer in the configuration utility. Otherwise they won't work. The mouse has limited

movements when the 'MicroStation Text Window' is not in full screen.

Software switch

In addition, there is a special software switch for the MGA Windows driver that can be set in the *system.ini* file under the section *[MGA.DRV]*. This switch tells the MGA Windows driver which board to use. The syntax of the switch is:

MGASelect = x

Where *x* is a number from 0 to 4. The default setting is 0.

- 0 Use the MGA with VGA enabled. If no MGA's have VGA enabled, use *MGASelect=1*.
- 1 Use the first board **without** VGA enabled. This board is used by the Matrox MicroStation driver. DO NOT use this value if you're running MicroStation dual screen under Windows with two Matrox boards.
- 2 Use the second board **without** VGA enabled.
- 3 Use the third board **without** VGA enabled.
- 4 Use the fourth board **without** VGA enabled.

MGA driver reconfiguration

- If you wish to change certain aspects of your setup, you will have to run the *usconfig* program again and reselect the MGA driver for the desired screen, so that the configuration screen will appear.

Tips and limitations

- The Fast rendering option is currently supported only on **Orthogonal** views (views that have the camera attribute OFF).
- When using fast rendering, the colors are not exactly the same as MicroStation rendering.
- Rendered views with fast rendering are cleared after the operation when the attribute **Delayed display** is ON (**Setting/ Rendering/ Attribute** menu).
- When a window in front of a view is moved off to the side, there may be some lines missing because of drawing modifications. Just update the view to correct this problem.
- When a window in front of a newly rendered view is moved off to the side, the exposed rectangles are drawn in wireframe.

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