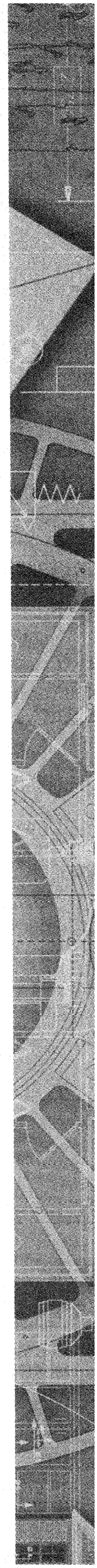
Getting Started Guide

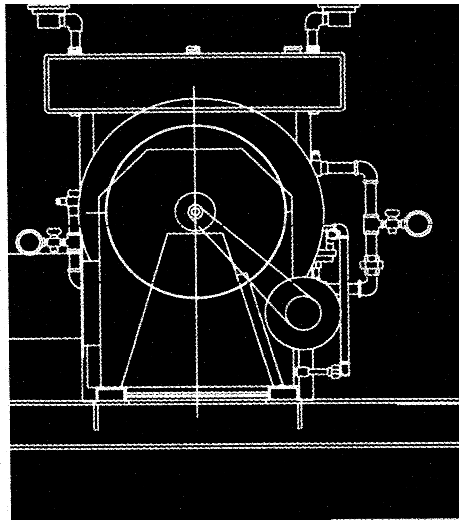
AutoCAD LT® 97

05704-094800-5010 September 5, 1997

Contents



About AutoCAD LT Documentation



This *Getting Started Guide* provides an introduction to the most commonly used features of AutoCAD LT®. Use this guide to familiarize yourself with AutoCAD LT so you can begin working quickly. Then you can use the references that point to online information for in-depth procedures and additional command resources.

This guide is complemented by several online information resources, including Help, a series of tutorial lessons called *Learning Assistance*, and two brief tours called *Quick Tour* and *What's New*. To learn how to navigate through the complete set of documentation, see the inside front cover of this manual.

In this chapter

- Detailed descriptions of the printed guides and the online user assistance
- Learning strategies for new and upgrade users
- Sources of additional help
- Conventions used in this guide

The New AutoCAD LT Manual and Help

This guide and the online Help have been designed to help you learn AutoCAD LT more efficiently, find the help you are looking for as you work, and to make the information easier to understand.

Getting Started Guide

This *Getting Started Guide* provides an introduction to get you working quickly and efficiently with AutoCAD LT. You'll find the organization of the book matches the order of tasks you usually follow in producing a drawing. As you read through the chapters, you'll have an overview of the kind of tools AutoCAD LT provides and a headstart on how to use them to get your work done. The guide doesn't attempt to provide you with every procedure and optional setting available; if you do need more information, use the pointers to the online Help.

A *Quick Reference Card* provided as a pull-out card at the back of the guide includes reference information on commonly used shortcut keys, coordinate entry methods, command aliases, and object selection methods.

Online Help

Previous releases of AutoCAD LT included a printed *User's Guide* with detailed information about how AutoCAD LT works and with step-by-step information showing how to do your work. This release provides all that information online, organized so that you can find the *type* of information you are looking for, whether it's a procedure for inserting a symbol from a Web site into your drawing, or the conceptual intricacies of controlling the display of externally referenced drawings.

The improved online Help provides quick information while you work, helping you to maximize productivity by minimizing the time you spend learning how to use AutoCAD LT. Help topics provide conceptual and overview material, step-by-step instructions, visual examples, and references to relevant commands and system variables.

You access the information by selecting AutoCAD LT Help Topics from the Help menu. Additional methods for getting help are described in "Getting Assistance While You Work" on page 49.

NOTE For information about late changes to AutoCAD LT that are not included in this documentation, refer to the *readme.hlp* file in the Help folder of the main AutoCAD LT folder.

Learning Tools

You access the following learning tools from the Help menu.

- ***Quick Tour*** For new users. A multimedia overview of basic AutoCAD LT concepts and capabilities.
- ***What's New*** For users upgrading from the previous release of AutoCAD LT. A multimedia presentation highlighting the significant new features found in AutoCAD LT.
- ***Learning Assistance*** For all users. An interactive multimedia learning tool designed to help you acquire the skills you need to use AutoCAD LT efficiently. Topics include drawing and editing objects, customizing the user interface, creating and maintaining symbol libraries, dimensioning drawings, creating text, and importing data from other applications.

Where to Start

You probably have one of two backgrounds: you are new to AutoCAD LT and computer-based drafting, or you are upgrading from a previous release of AutoCAD LT. Based on your experience, the following will help you determine the documentation and learning tools that best meet your needs.

New User

If you are new to AutoCAD LT, start by viewing the *Quick Tour* multimedia presentation found on the Help menu. This will give you a good overview of AutoCAD LT capabilities and the concepts of CAD. In addition, the introduction to this guide, "Making the Transition from Paper to CAD," provides a graphical overview of the advantages of CAD over paper-based drafting.

To try out some of the features you saw in the *Quick Tour*, run *Learning Assistance* from the Help menu. Following the lessons will give you some basic hands-on AutoCAD LT experience. Appendix A in this guide provides a simple one-hour tutorial that provides an introduction to drawing objects, adding annotation and dimensions, and printing or plotting your finished drawing.

To learn conceptual information in more depth and get an overview of AutoCAD LT capabilities, use this *Getting Started Guide*. Reading through the manual will introduce you to the most commonly used features and prepare you to start working.

As you gain experience, you can learn more about all of the options of specific commands and procedures by referring to online Help. Expanded and improved, the online Help provides a complete reference with access tools to get you the right information quickly.

Upgrade User

If you are upgrading from the previous version of AutoCAD LT, choose *What's New* from the Help menu. This multimedia presentation highlights the significant changes and improvements made in this release of AutoCAD LT. More detail about the features described in *What's New* can be found in this *Getting Started Guide* and in online Help.

To increase your knowledge and productivity, choose *Learning Assistance* from the Help menu. These tutorials will help you work more efficiently, teach you about new, advanced, and underutilized AutoCAD LT features, and show you how to collaborate with others by sharing your designs across the Internet.

Additional Help Resources

The following resources help you get information about Autodesk products and assistance with your AutoCAD LT questions.

- Autodesk Web site: *http://www.autodesk.com*
- AutoCAD LT 97 Web site: *http//www.autodesk.com/products/autocad/autocadlt*
- AutoCAD LT documentation suggestions, errors, or questions: send email to *acad_docs@autodesk.com*

Use of This Guide

The following conventions are used in this guide.

command name entered at Command Prompt

toolbar name

menu commands or submenus follow right arrow

figures of dialog boxes include text on how to use them

commands, system variables, named objects and keys you press on the keyboard are uppercase

icon shows equivalent toolbar access

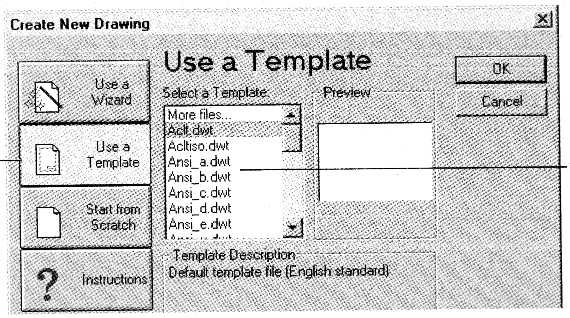
text you enter is bold

Using Templates

All new drawings are based on a template. You can use any drawing as a template drawing. When you use an existing drawing as a template, all information is passed on to the new drawing.

Standard Toolbar	**Menu**	**Command**
	File ➤ New	NEW

Choose Use a Template



select a template from the list or choose More files

To create a template

1. Open a drawing and change any drawing settings to match what you plan to use as defaults in your template. If needed, add a border and title block.
2. Erase all the objects except the border and title block by entering ERASE, then press ENTER at the Command prompt. Enter **all** at the Select Objects Prompt.
3. Enter **r** (Remove) and select the border and title block to remove them from the selection set to be erased.
4. To save your newly created template, in the Save As dialog box, choose Drawing Template file type under Save as Type.
5. Choose OK.

ONLINE HELP To work with templates, see "Starting with a Template File."

Chapter 2 Organizing Your Project

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Making the Transition from Paper to CAD

Now that you've made the switch from paper to computer-aided design (CAD), how can you accomplish your drawing tasks faster and more efficiently? This chapter is designed to show you what AutoCAD LT can do for you. If you're already familiar with paper-based drafting terms, you'll get a visual introduction to the equivalent concepts in CAD, and discover how easy it is to become more productive. This introduction explains the basics of CAD and uses real-life AutoCAD LT drawings to illustrate each topic.

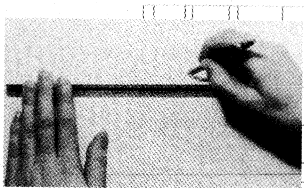
For a multimedia review of AutoCAD LT features, see the online *Quick Tour* on the Help menu.

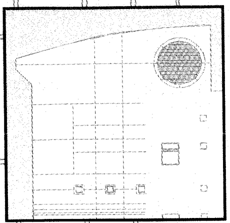
Drawing to Scale

As with any paper-based drafting project, a CAD project requires planning. Establishing the scale of your drawing is one of the first activities you undertake on paper, but in CAD, setting the scale can be saved until last.

A CAD drawing has its own set of preliminary activities.

When you draft on paper, you determine the scale before you start drawing. This scale compares the size of the drawn object to the actual size of the object the drawing represents. For example, in an architectural drawing each quarter inch might equal one foot in the floor plan of a house. The scale you choose must allow the drawing of the object to fit on the paper.

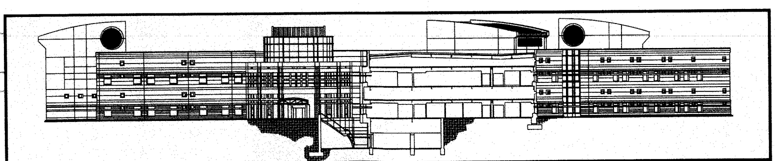




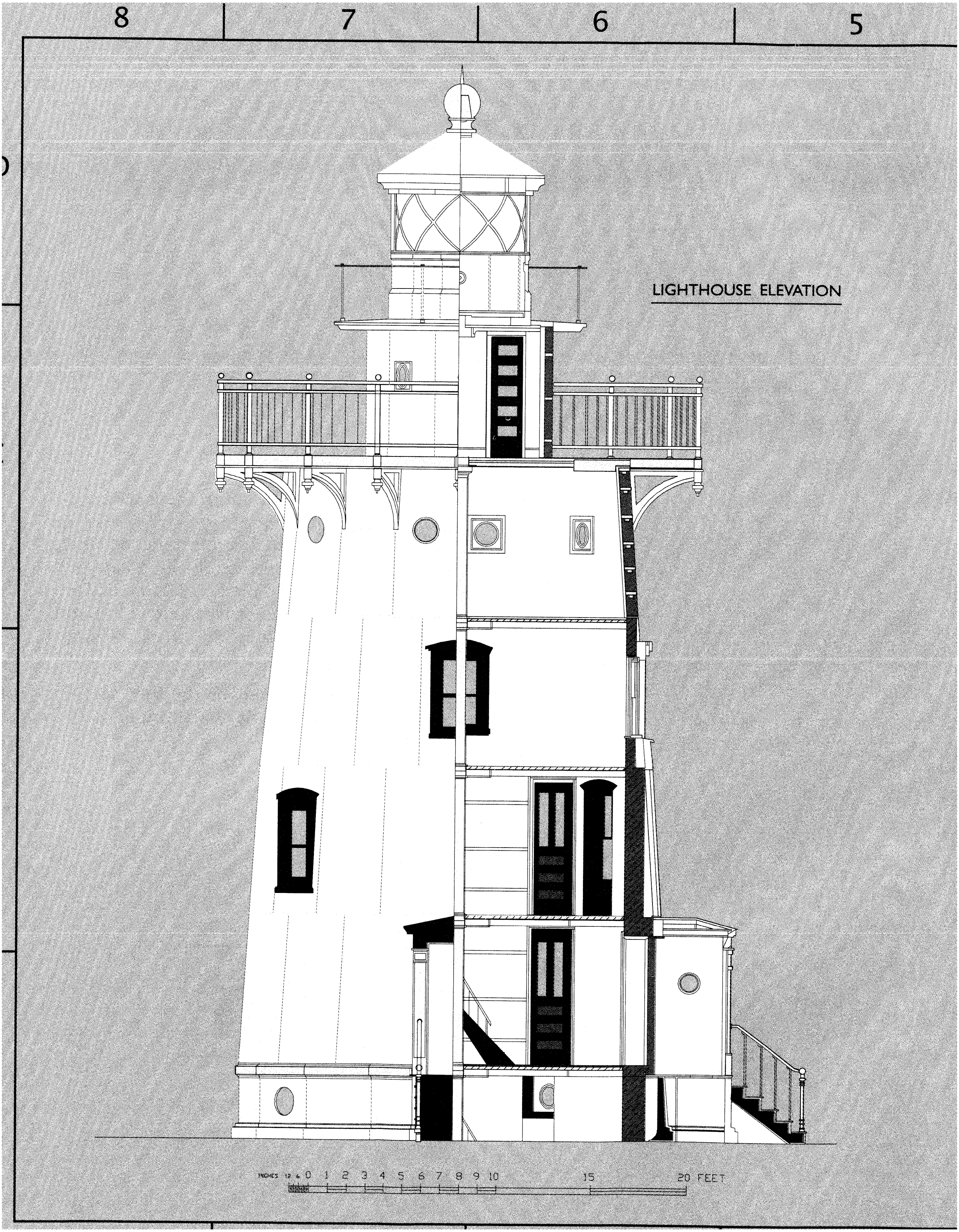
Scale: 1/8"= 1'- 0"

In AutoCAD LT, you don't need to think about setting a scale until you're ready to print or plot your drawing. You draw with a unit type you specify (architectural, decimal, and so on). Each unit on the screen represents whatever you want it to be: an inch, a millimeter, a kilometer. For example, if you are drawing a motor part, one unit might equal one millimeter. If you are drawing a map, one unit might equal one kilometer.

When you're ready to print or plot, you can then set different scales for different sections of a drawing.



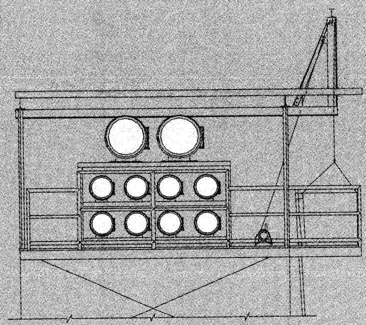
Scale: 1/16"=1'- 0"


8
7
6
5
LIGHTHOUSE ELEVATION
INCHES 12 6 0 1 2 3 4 5 6 7 8 9 10 15 20 FEET

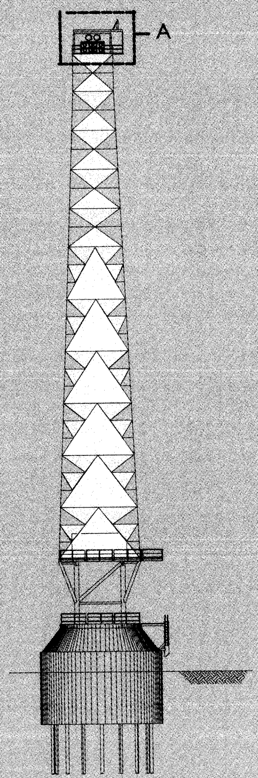
Setting Up Your Drawing Layout

In both paper-based drafting and CAD, the drawing layout is fundamental to creating well-designed drawings. A paper layout is defined by the sheet size you use. In CAD, the layout is not limited by paper size, and is readily changed throughout the drawing process.

When you draft on paper, you establish the final drawing layout—plans, elevations, sections, details, title block, and border—early in the drawing process. Based on a specific sheet size, you assign locations for various scaled views.



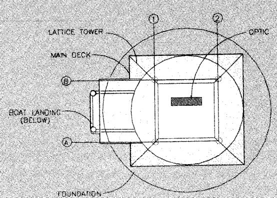
OPTIC DECK ELEVATION A



TOWER ELEVATION B

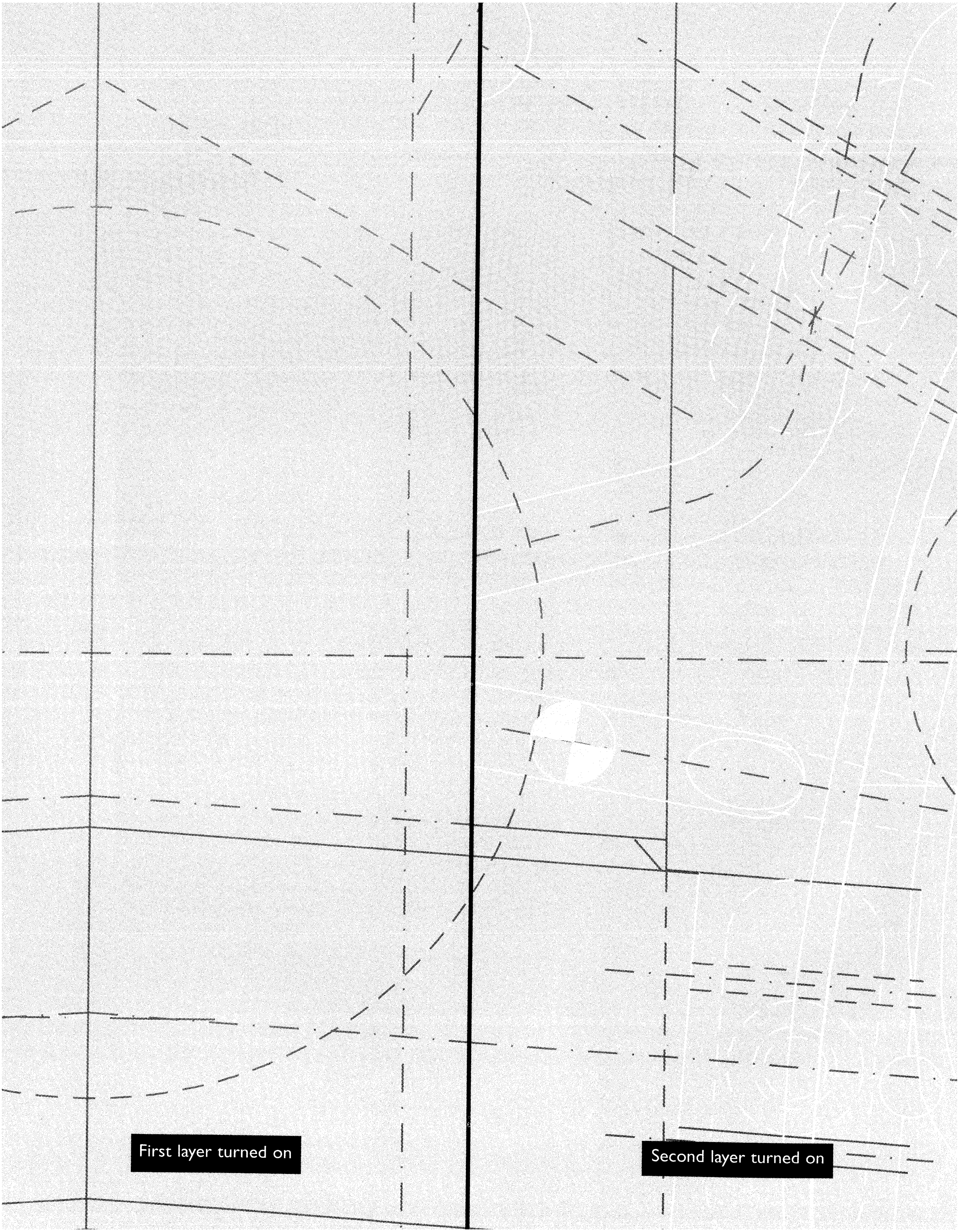
With AutoCAD LT you start with a work space that is virtually unlimited. As with scaling, you can delay designing the final layout until you're ready to print.

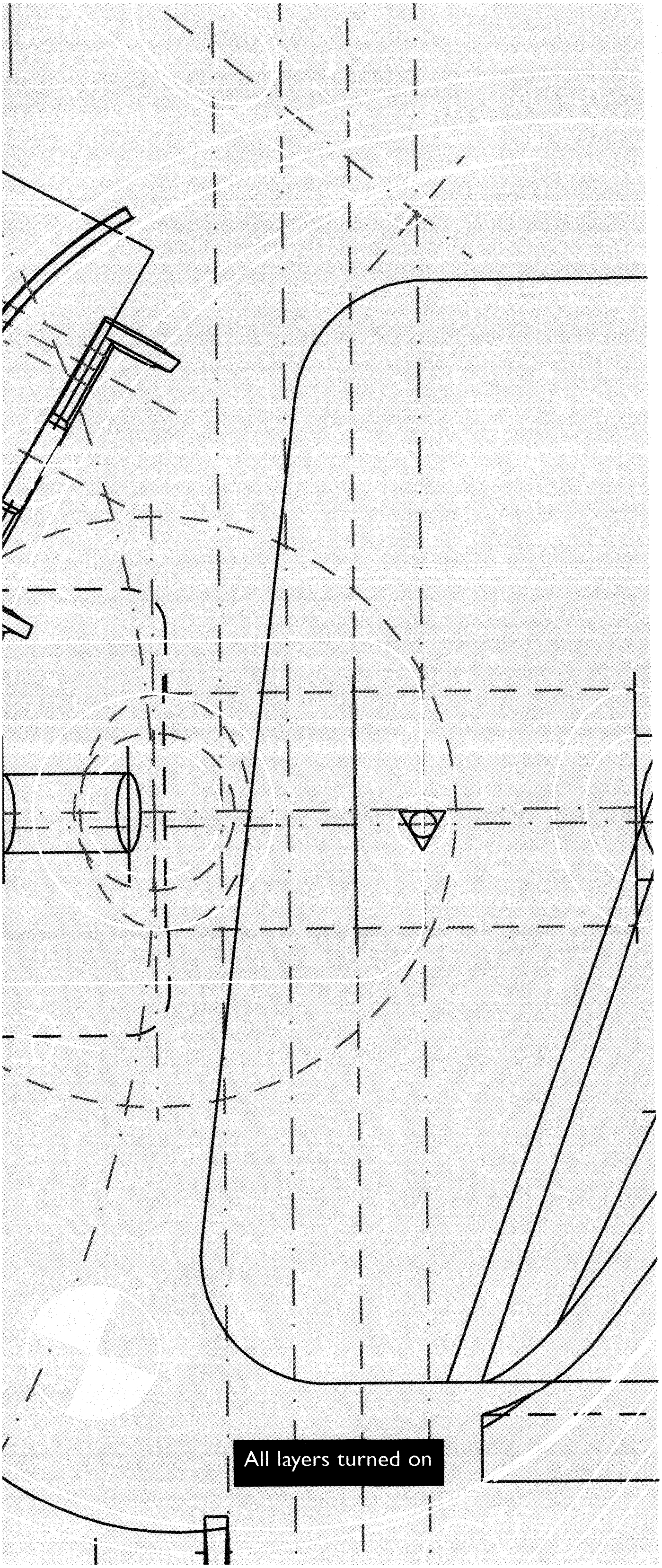
AutoCAD LT provides two drawing environments: model space and paper space. You usually create your drawing, called a *model*, at full scale in an area known as model space without regard to the final layout on paper. Then, when you're ready to arrange elements of the drawing for printing, you can switch to *paper space*. There you can create *floating viewports*, or "windows" into your drawing, which show different scales of your model. Conceptually, paper space represents the paper on which you plot.



TOWER PLAN

U.S. COAST GUARD		
05-9913A	ELEVATIONS	
S.S.	A-1	SCALE: AS SHOWN

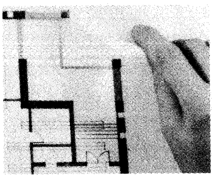

First layer turned on
Second layer turned on



Organizing Drawing Information

Organizing the content of your drawing is a fundamental aspect of drafting, either on paper or in CAD. Combining various types of drawing elements helps you maintain the data easily and lets you control the display of the content selectively.

When you draft on paper, you can separate elements onto transparent overlays. For example, a site plan might contain overlays for foundations, floor plans, heating plans, and piping.



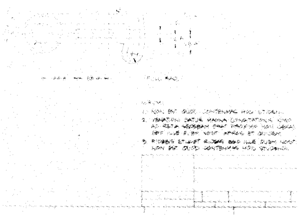
In AutoCAD LT, *layers* are equivalent to overlays. You can name each layer, assign a color and linetype to it as dictated by industry or job standards, and then group drawing elements by assigning similar objects to the same layer.

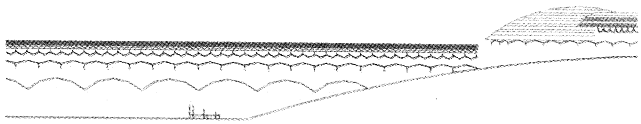
In addition to organizational benefits, layers give you greater control over the graphic display. The illustration shows how you can turn on layers to easily identify the effects of changes you make on objects in all the other layers. Then, you can turn layers off to remove details as you work.

Establishing Drafting Standards

Whether you work as a member of a team on a project or work alone, developing standards is a requirement for efficient communication and organization of your work.

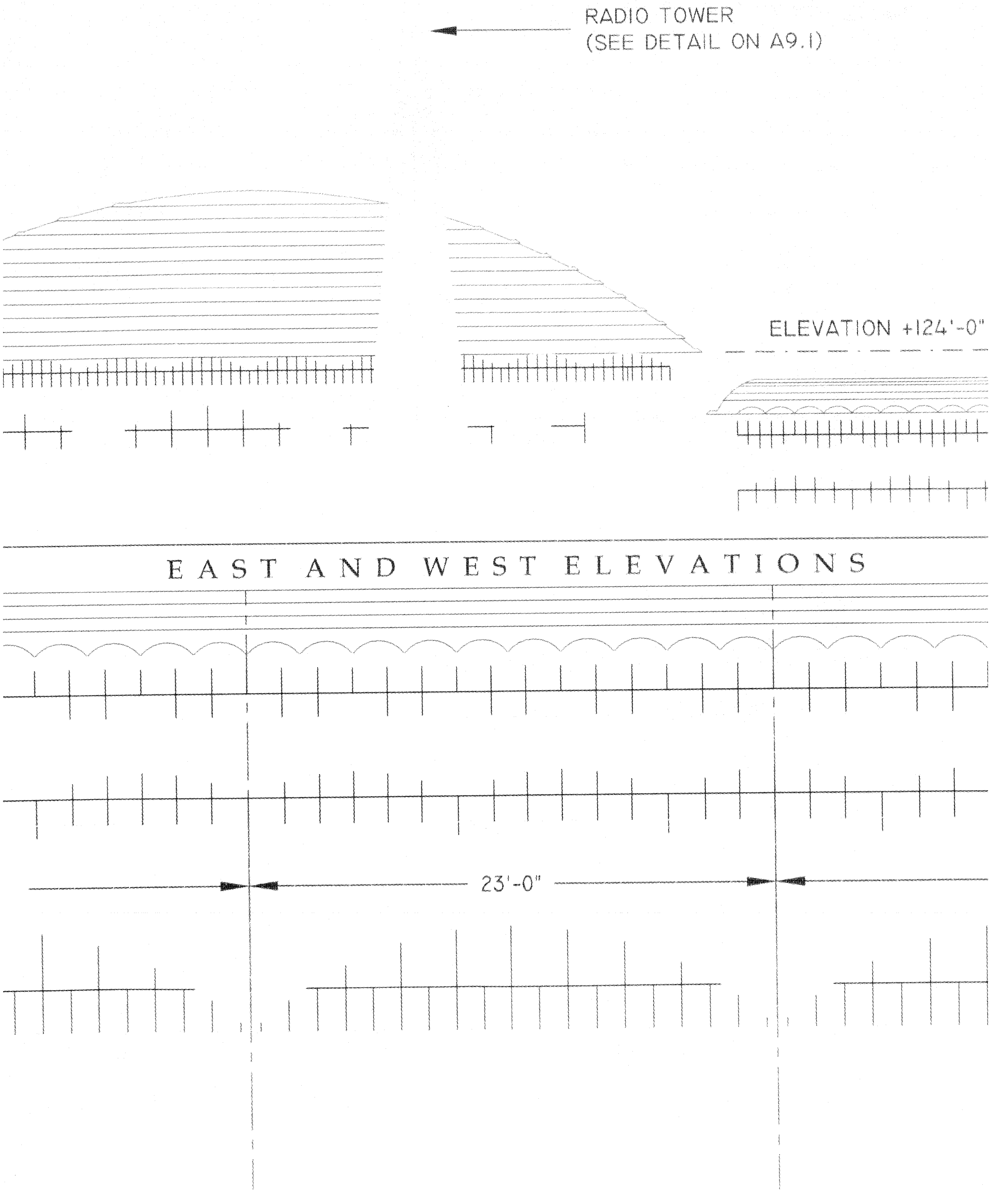
Different jobs have different sets of drafting standards, not only for linetypes and colors, but also for text and dimensioning. Paper-based drafting requires meticulous accuracy and detailed guidelines to maintain company or client standards. For each new project these standards must be applied from the beginning and conform to the guidelines.





With AutoCAD LT, you can ensure conformity to standards by creating reusable *styles*. Styles are groups of settings that you can name and use repeatedly to apply consistency across your project. You can create styles for text, dimensions, and linetypes. A text style, for example, establishes font and typeface characteristics such as height, width, and slant. When you use that style, the text you create assumes those characteristics.

Once styles are established, you can save them in drawing *templates*. Any drawing (*.dwg*) that contains the settings you need can be saved as a template drawing (*.dwt*). Templates may also contain standard layers, viewport configurations, and a customized title block and border.

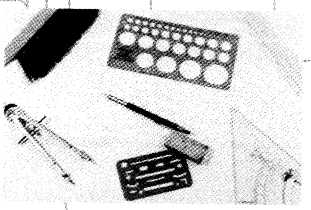


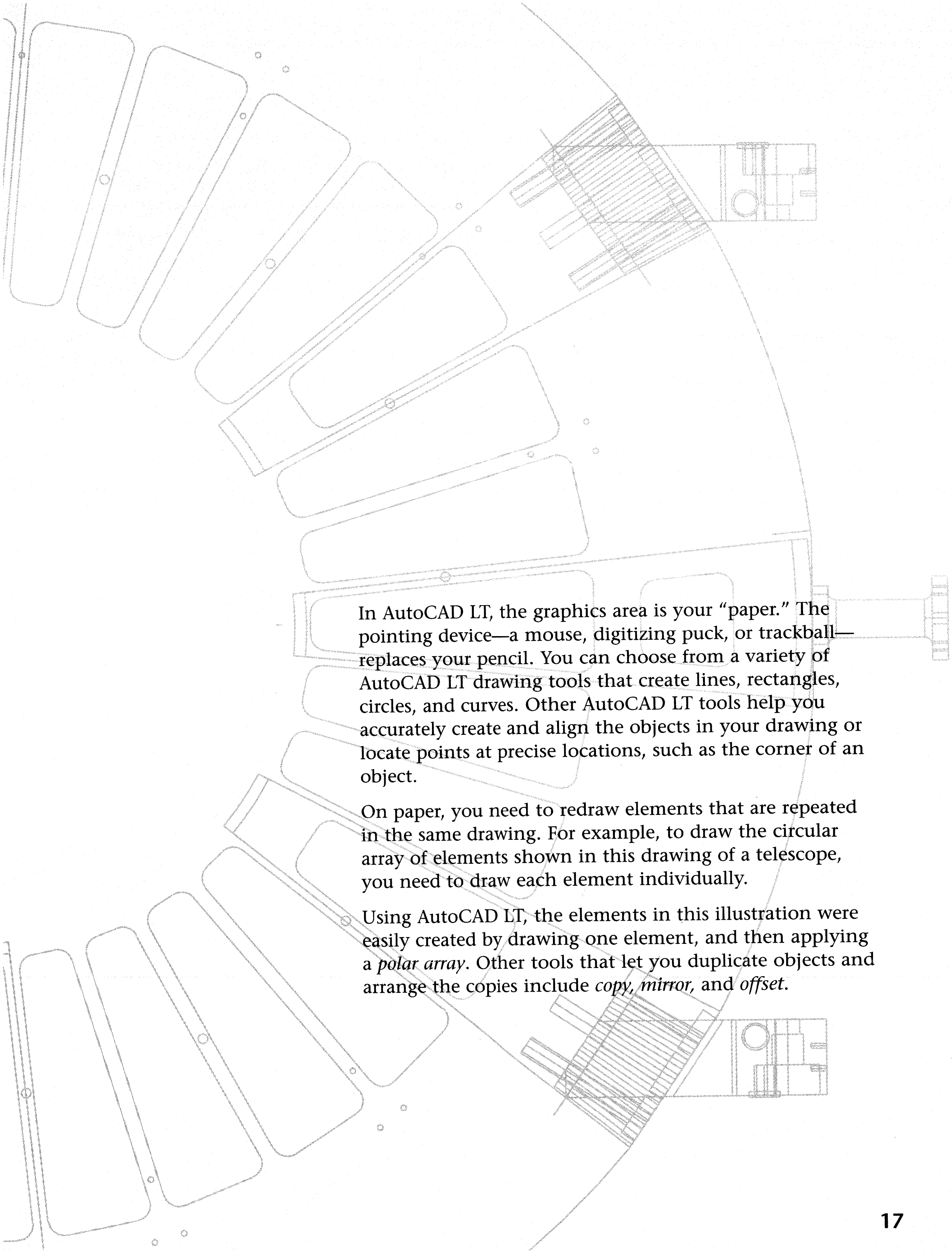
EAST AND WEST ELEVATIONS

Drawing Efficiently

Draw with less effort and revise with more speed: these are the two main reasons you use CAD. With the AutoCAD LT complete set of drawing tools and standard symbols and its ability to update dimensions and text, you can eliminate repetitive, time-consuming tasks.

If you work with paper and a drawing board, your set of drawing tools is likely to include pencils, scales, parallel rules, templates, erasers, various sizes of leads, and more.

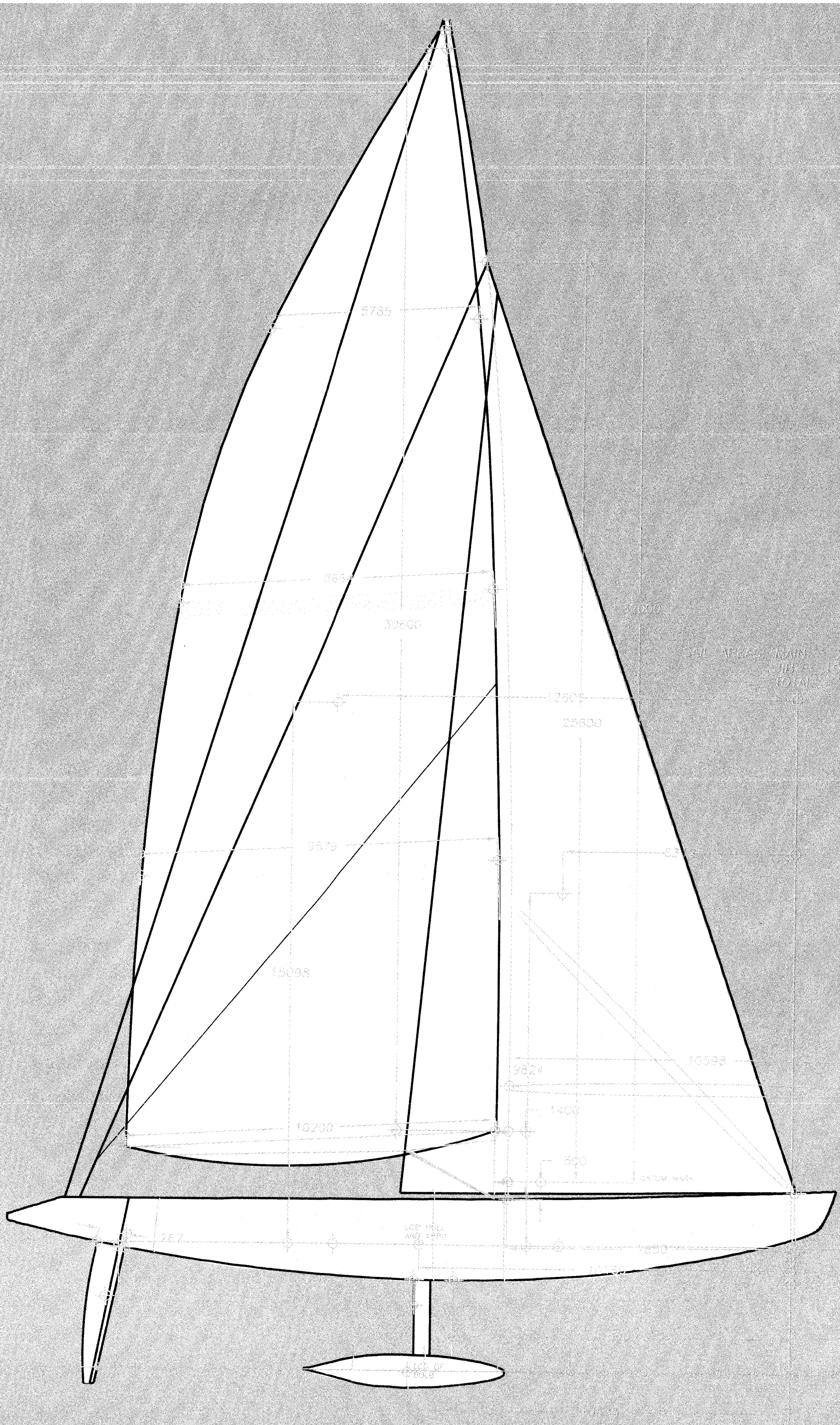




In AutoCAD LT, the graphics area is your "paper." The pointing device—a mouse, digitizing puck, or trackball—replaces your pencil. You can choose from a variety of AutoCAD LT drawing tools that create lines, rectangles, circles, and curves. Other AutoCAD LT tools help you accurately create and align the objects in your drawing or locate points at precise locations, such as the corner of an object.

On paper, you need to redraw elements that are repeated in the same drawing. For example, to draw the circular array of elements shown in this drawing of a telescope, you need to draw each element individually.

Using AutoCAD LT, the elements in this illustration were easily created by drawing one element, and then applying a *polar array*. Other tools that let you duplicate objects and arrange the copies include *copy, mirror,* and *offset.*


5785
8650
30600
9579
15098
10200
12800
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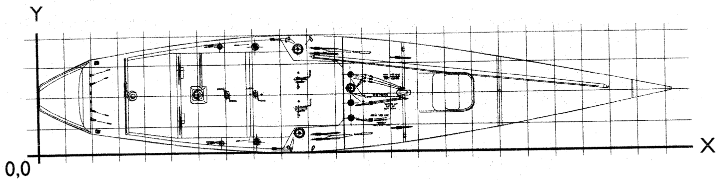
9824
1400
500
767

Using Precision Drawing Tools

Whether you're drafting with pencil and paper or using CAD, your drawing must have the greatest possible degree of accuracy and precision.

On paper, when you use scales and parallel rules, you need to carefully double-check dimensions to ensure accuracy.

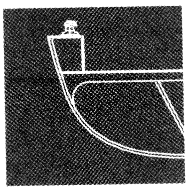


In AutoCAD LT, you can ensure exact dimensions using several methods. One method is to specify coordinates. All drawings are superimposed on an orthogonal grid (Cartesian coordinate system) with a horizontal *X* axis and a vertical *Y* axis. You can turn on or turn off the grid to display it or conceal it.

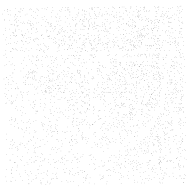
A single unit in the coordinate system represents the basic unit that you choose to use for your drawing (an inch, a millimeter, a kilometer, and so on).

As you draw, you can specify the exact (absolute) coordinates if you know them. You can also specify a *relative coordinate* as the *X* and *Y* distance from the last point you located, and a *polar coordinate* as a distance and an angle from the last point.

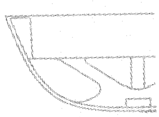
To gain accuracy without specifying coordinates, you can set a *snap* interval that will force the crosshairs to jump, or "snap," to intervals you set. *Ortho* helps you draw lines at right angles. When you need to constrain lines along the vertical and horizontal axes, use Ortho as your CAD T-square. You can also use *PolarSnap* to specify precise angles and distances. For example, you can specify an angle of 45 degrees and a distance of 2 units to draw lines to these constraints.



Snap limits the movement of the crosshairs to precise intervals.



Ortho helps you draw lines at right angles.

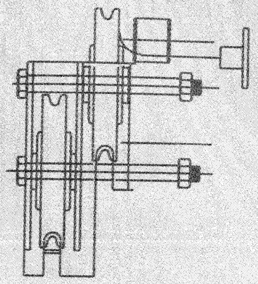


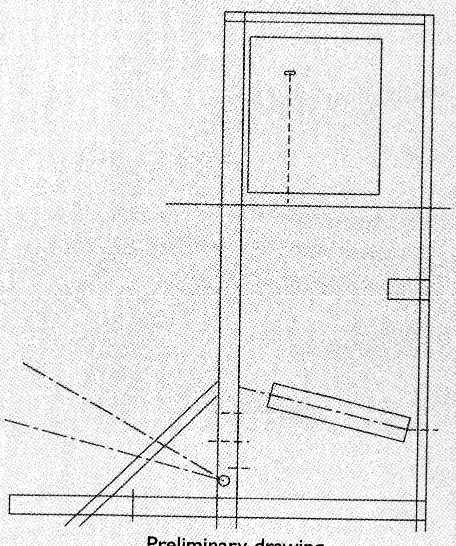
PolarSnap helps you draw lines at precise angles and distances.

Modifying Your Drawing

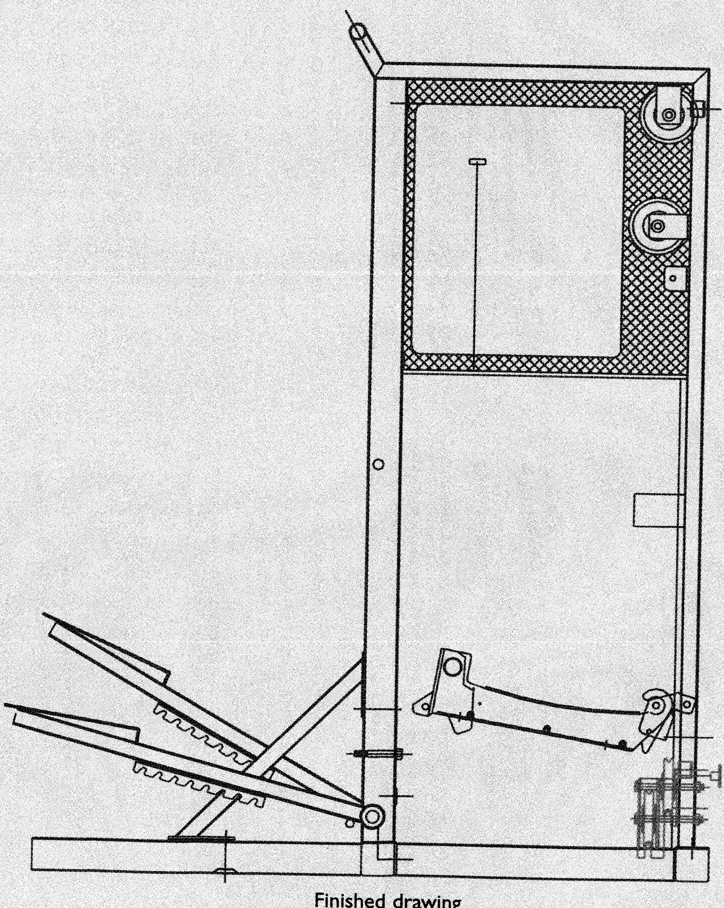
Revisions are a part of the development of any drawing project. Whether you work on paper or with CAD, you usually need to modify your drawings in some way.

In paper drawing, you have to manually erase and redraw.





Preliminary drawing

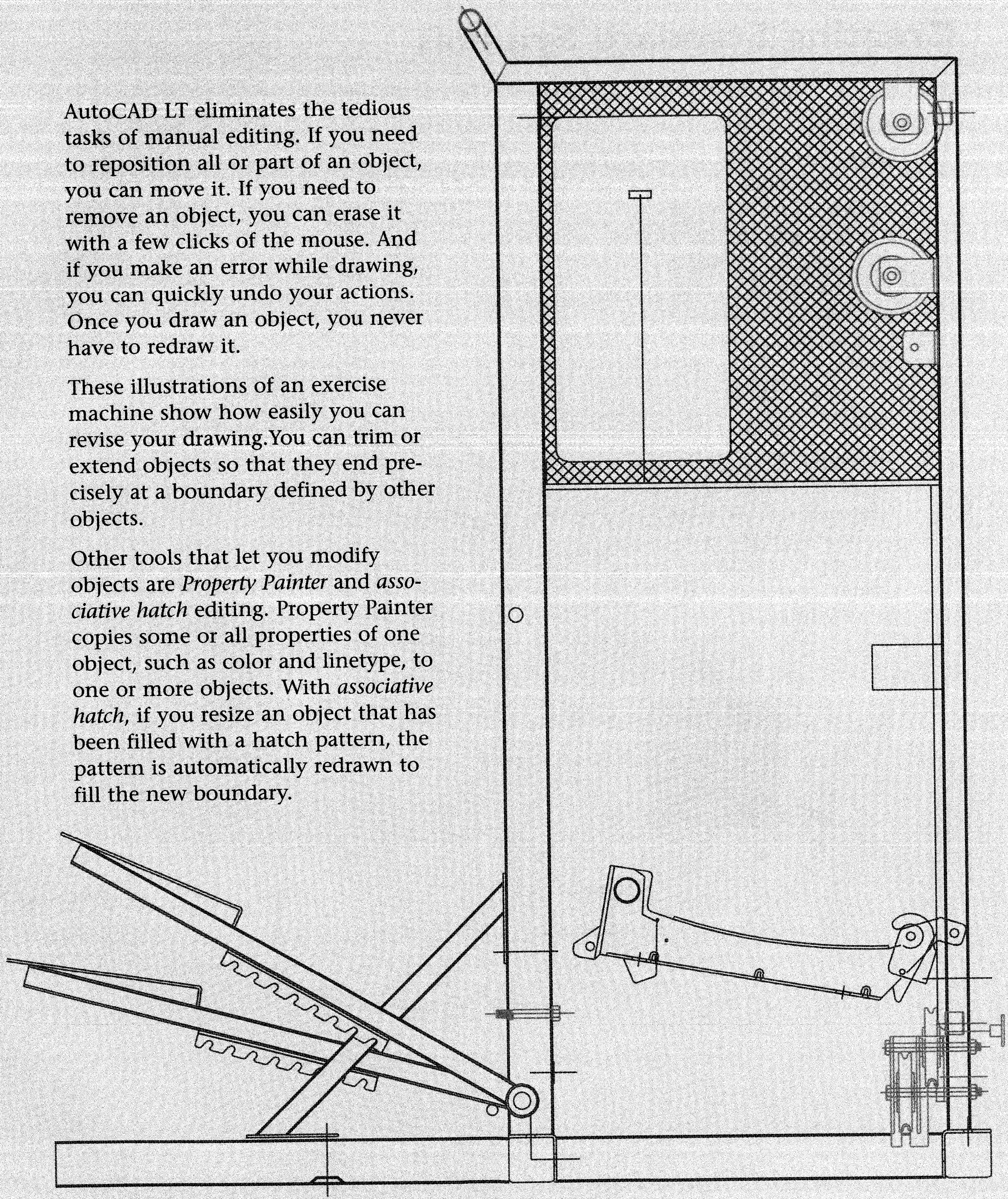


Finished drawing

AutoCAD LT eliminates the tedious tasks of manual editing. If you need to reposition all or part of an object, you can move it. If you need to remove an object, you can erase it with a few clicks of the mouse. And if you make an error while drawing, you can quickly undo your actions. Once you draw an object, you never have to redraw it.

These illustrations of an exercise machine show how easily you can revise your drawing. You can trim or extend objects so that they end precisely at a boundary defined by other objects.

Other tools that let you modify objects are *Property Painter* and *associative hatch* editing. Property Painter copies some or all properties of one object, such as color and linetype, to one or more objects. With *associative hatch*, if you resize an object that has been filled with a hatch pattern, the pattern is automatically redrawn to fill the new boundary.

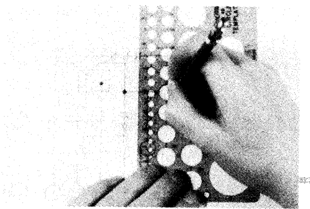


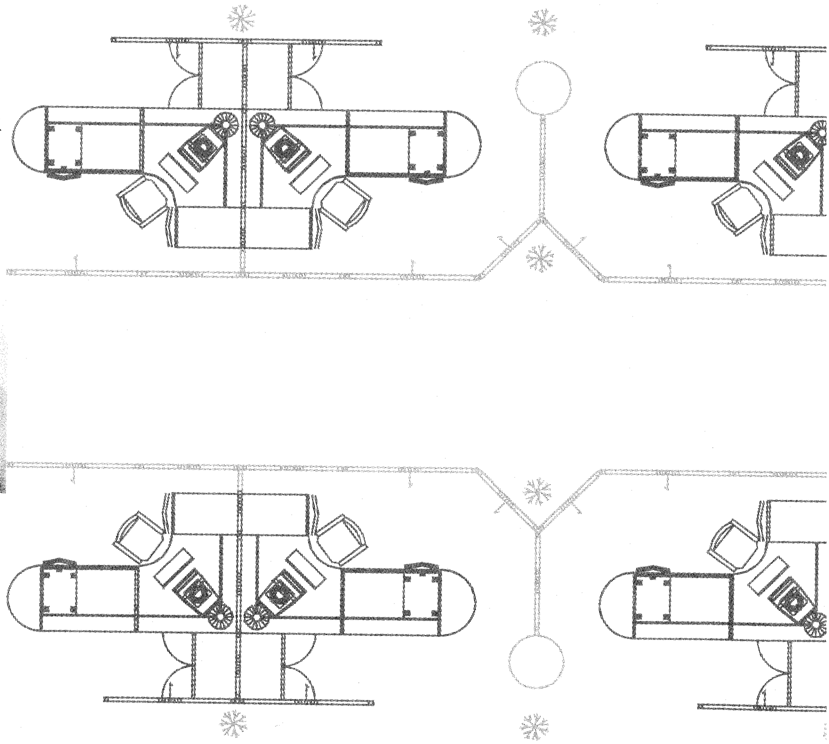
Modified drawing

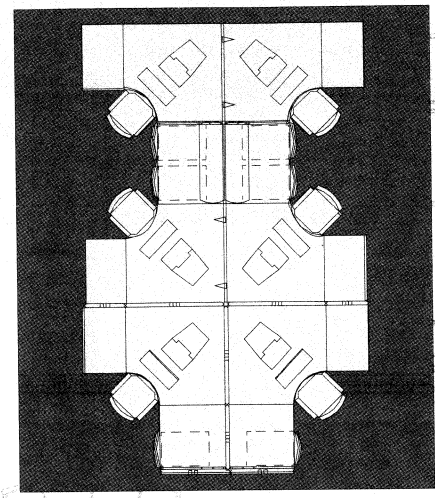
Creating Standard Symbols

Standard *symbols* are created by combining geometric objects to represent physical objects. Symbols are an efficient way to reuse the common elements in your drawings; once you have established a symbol library, you minimize the need to re-create drawing content.

When you draft on paper, you might use a template or a sticky-back to draw standard architectural, mechanical, design, electrical, and other symbols.

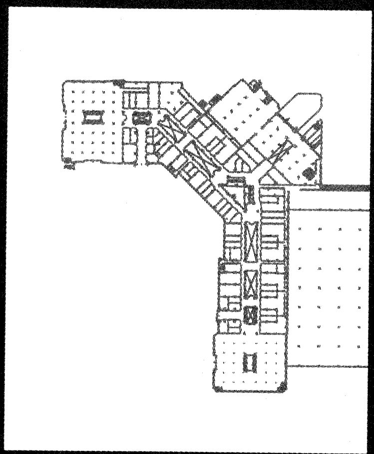




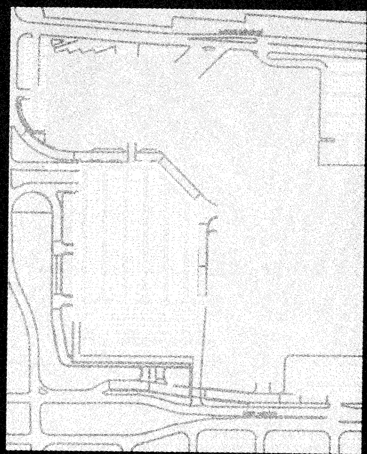


In AutoCAD LT, you can save drawing time by inserting copies of existing symbols anywhere in your drawing. When you need to create a custom symbol, you can group several objects into a single object called a *block*. You can then insert it as many times as you like into any drawing. In this example, office furniture was grouped into a block and inserted wherever this office grouping was required.

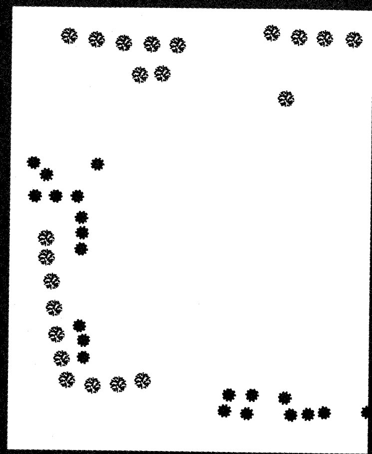
By creating blocks, you can develop a library of custom symbols that you can use repeatedly and update easily. You can also browse and preview symbols from your own drawings, your company's network, and even manufacturers' catalogs on the Autodesk Data Publishing Web site. You can then insert them directly into your current drawing.



Building plan: externally referenced in the site plan.



Site plan: contains external references to the building plan and landscaping.



Landscaping: externally referenced in the site plan.

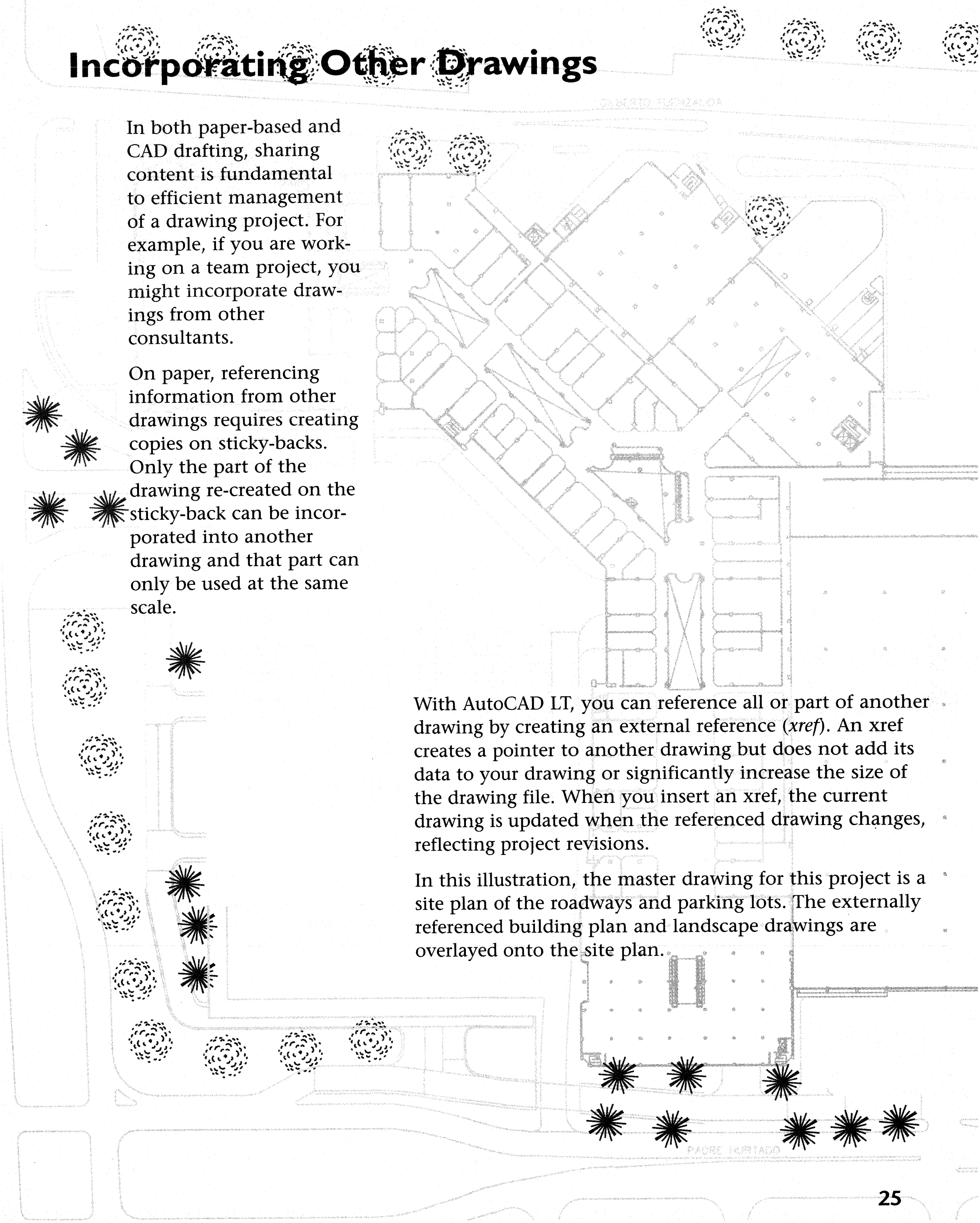
Incorporating Other Drawings

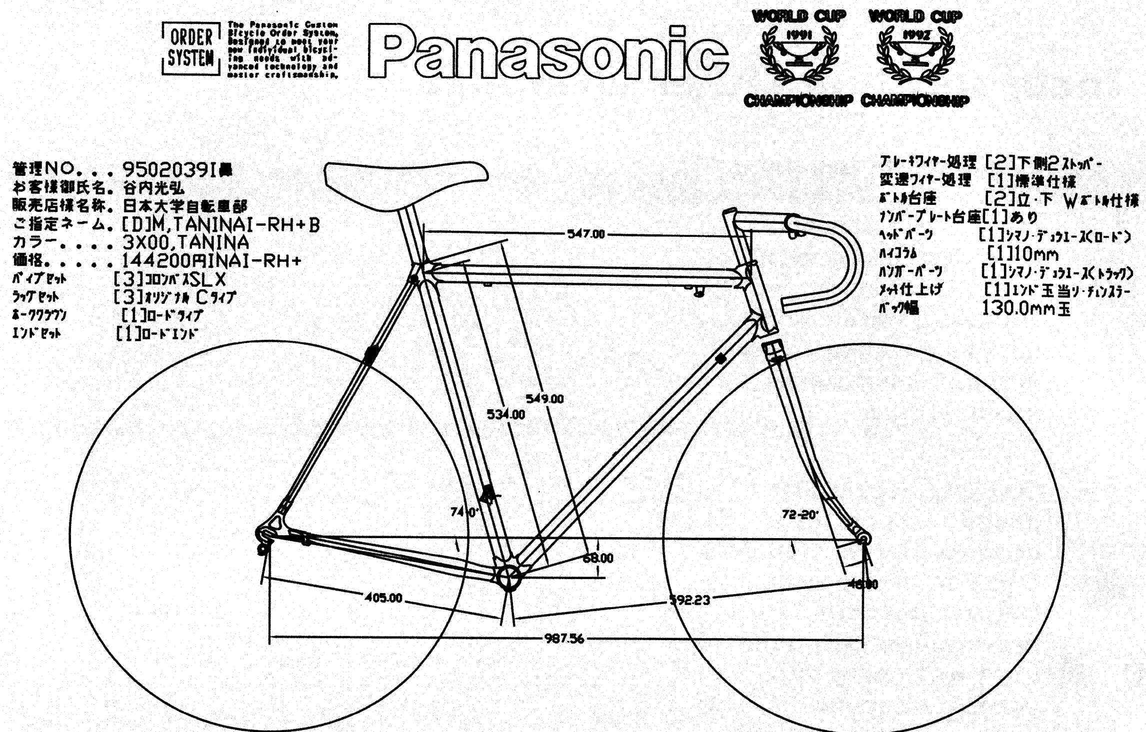
In both paper-based and CAD drafting, sharing content is fundamental to efficient management of a drawing project. For example, if you are working on a team project, you might incorporate drawings from other consultants.

On paper, referencing information from other drawings requires creating copies on sticky-backs. Only the part of the drawing re-created on the sticky-back can be incorporated into another drawing and that part can only be used at the same scale.

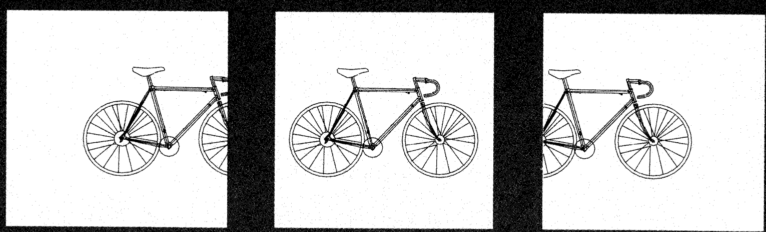
With AutoCAD LT, you can reference all or part of another drawing by creating an external reference (*xref*). An xref creates a pointer to another drawing but does not add its data to your drawing or significantly increase the size of the drawing file. When you insert an xref, the current drawing is updated when the referenced drawing changes, reflecting project revisions.

In this illustration, the master drawing for this project is a site plan of the roadways and parking lots. The externally referenced building plan and landscape drawings are overlayed onto the site plan.

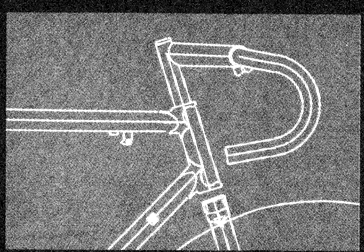




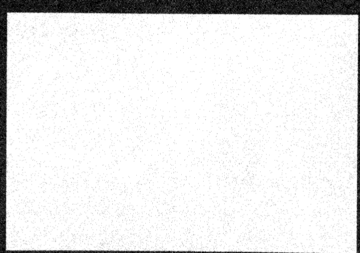
View the whole drawing.



Pan the drawing to view different sections.



Zoom in to view details.

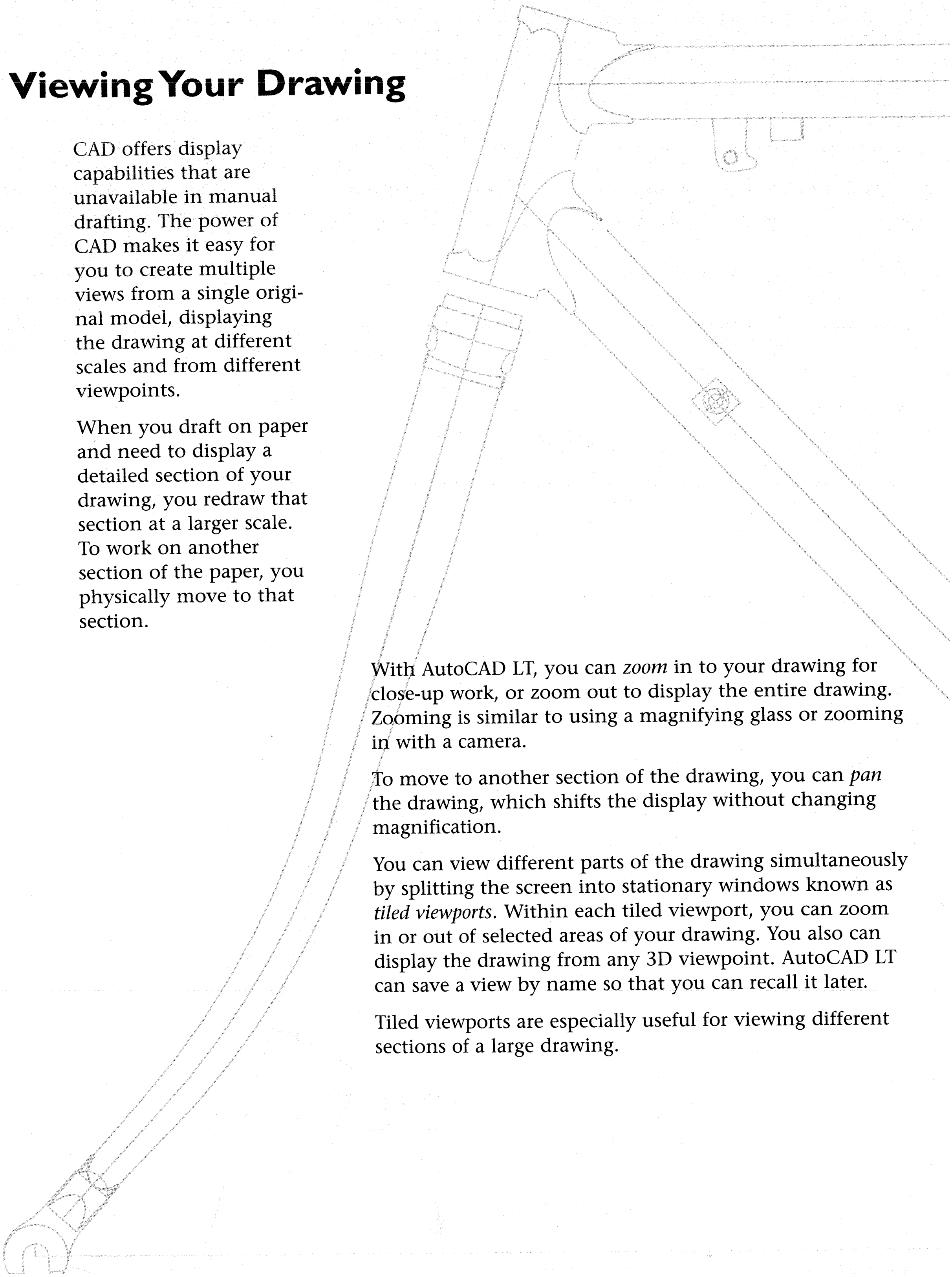


View different sections simultaneously using tiled viewports.

Viewing Your Drawing

CAD offers display capabilities that are unavailable in manual drafting. The power of CAD makes it easy for you to create multiple views from a single original model, displaying the drawing at different scales and from different viewpoints.

When you draft on paper and need to display a detailed section of your drawing, you redraw that section at a larger scale. To work on another section of the paper, you physically move to that section.



With AutoCAD LT, you can *zoom* in to your drawing for close-up work, or zoom out to display the entire drawing. Zooming is similar to using a magnifying glass or zooming in with a camera.

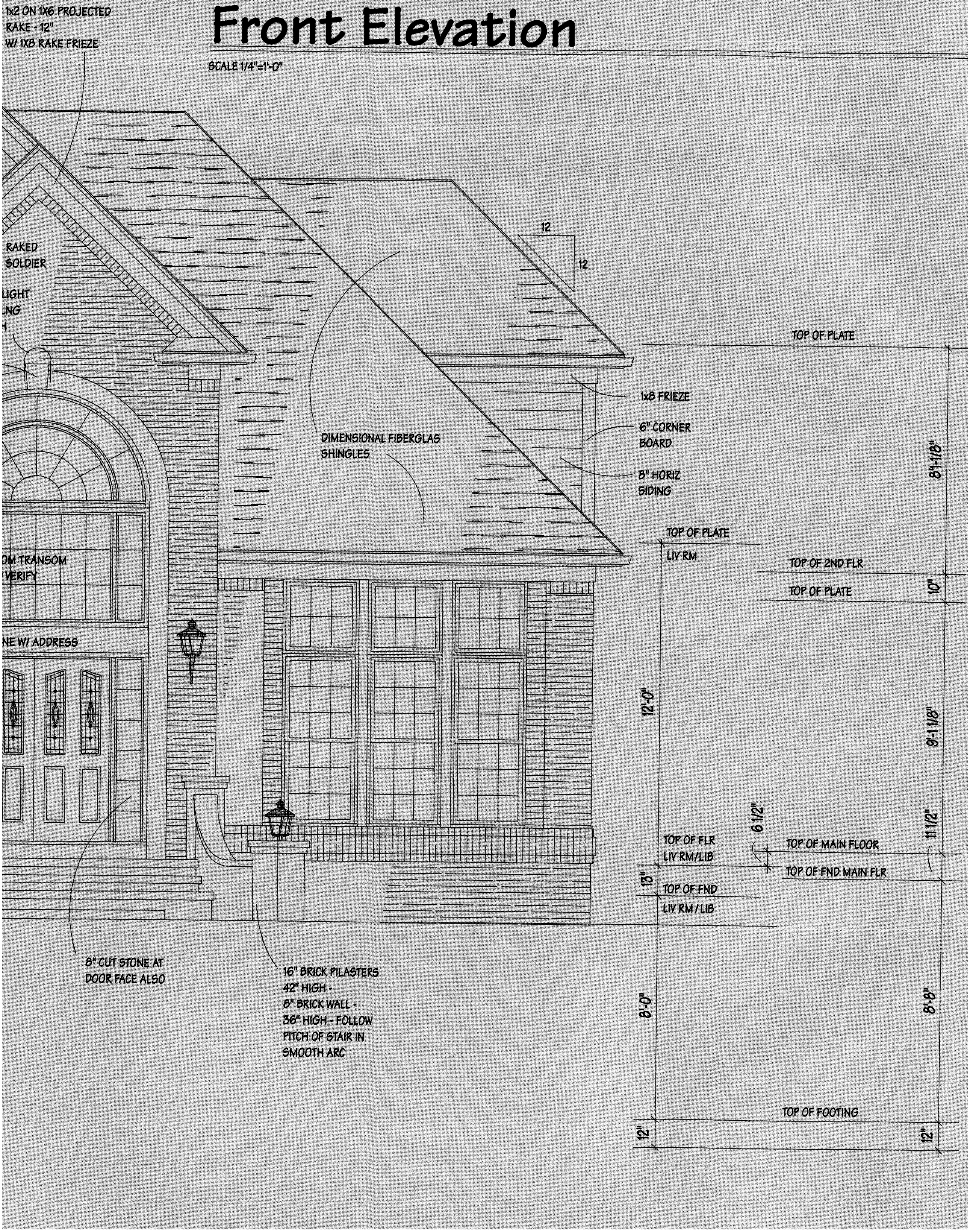
To move to another section of the drawing, you can *pan* the drawing, which shifts the display without changing magnification.

You can view different parts of the drawing simultaneously by splitting the screen into stationary windows known as *tiled viewports*. Within each tiled viewport, you can zoom in or out of selected areas of your drawing. You also can display the drawing from any 3D viewpoint. AutoCAD LT can save a view by name so that you can recall it later.

Tiled viewports are especially useful for viewing different sections of a large drawing.

Front Elevation

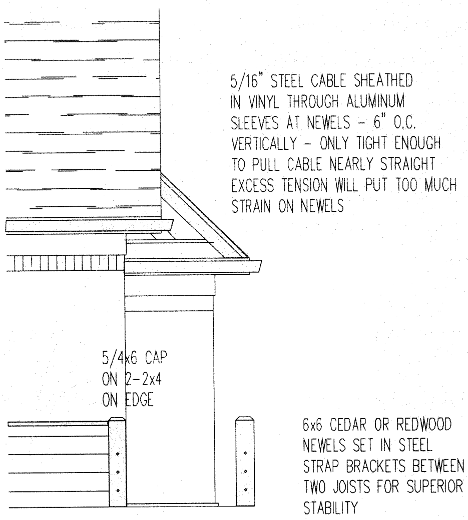
SCALE 1/4"=1'-0"



Updating Dimensions and Text

Dimensions and text convey important information in your drawing. Styles for your annotations help establish clear and professional-looking drawings.

When you work on paper, you typically draw to a specific scale and then add dimensions and annotation. If you resize any part of the drawing, you must erase and then redraw the dimensions.

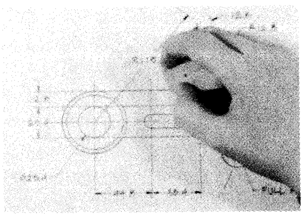


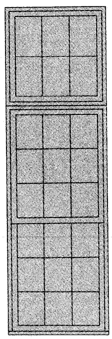
In AutoCAD LT, you can make dimensions associative: that is, if you stretch or scale a dimensioned object, you can simultaneously update the dimension value to reflect the new size.

Text can be created as a single line or as a paragraph. When information changes, you can revise the content, format, or size of the text quickly and easily.

You can attach invisible (or visible) data that describes an object. Such text is known as an *attribute* and can be extracted into a list or report.

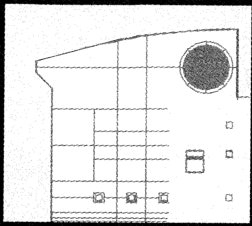
For example, you can attach attributes to a block, such as WINDOW, and enter different values for the attributes each time you insert WINDOW. Attribute information such as manufacturer, model, and cost, can be extracted from the drawing and used in a spreadsheet or database program.



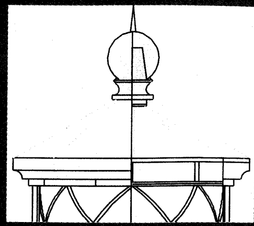


Window
Window MA 205
299.95
Marin Glass Co.
Wood Frame
3'w x 9'h

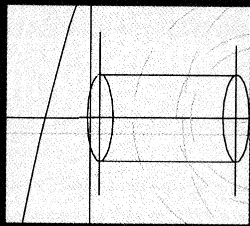
CREDITS



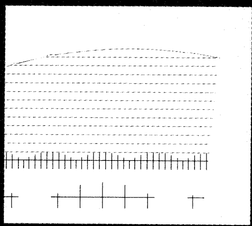
Anderson Debartolo Pan,
USA



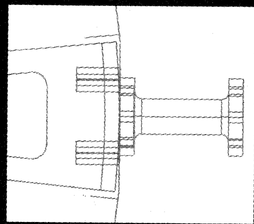
US Coast Guard,
Paul D. Herold, USA



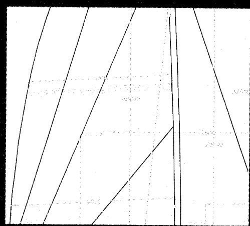
JPK Mould, P. Kusperth,
Germany



Marin Civic Center,
Artist Depiction,
Graziella Whipple,
USA

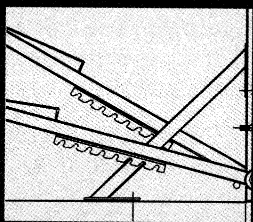


National Optical Astronomy
Observatories,
Gary P. Muller, USA



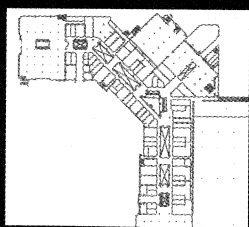
America3 Foundation,
Peter Wilson, USA

Nelson, Marek Yacht
Design, USA

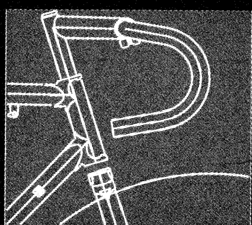


Proform Fitness,
USA

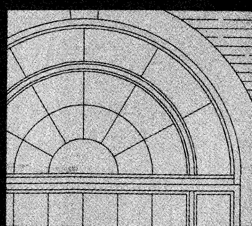
President Office Furniture,
Ralph Doran, USA



Sabbagh Architects,
Chile



Nippon Telephone and
Telegraph (NTT), Japan



Harkins and Associates,
W. Michael Harkins, USA

1

Installing AutoCAD LT 97

In this chapter

- System Requirements
- Installing AutoCAD LT for a single user
- Uninstalling AutoCAD LT

This chapter lists the hardware and software required to run AutoCAD LT and describes how to install AutoCAD LT for a single user. The program prompts you for specific information prior to transferring files from the release CD to subdirectories on your hard disk.

System Requirements

To run AutoCAD LT on your computer, the following hardware and software are required or supported as optional.

Hardware (Required)

- Intel 486 with math coprocessor or compatible processor (Pentium recommended)
- VGA video display (800 by 600 or higher resolution required)
- CD-ROM drive for initial installation and running *Learning Assistance*
- Windows-supported display adapter
- Mouse or other pointing device (IntelliMouse recommended)

Hardware (Optional)

- Printer or plotter
- Digitizing tablet

Software

- Windows NT 4.0 or Windows 95. It is recommended that you install and run your copy of AutoCAD LT on an operating system in the same language as your AutoCAD LT software or on an English version of one of the supported operating systems.
- DWF files are designed to be viewed on the Internet or company intranets using a Web browser. To view DWF files created with this version of AutoCAD LT or AutoCAD Release 14, you must download and install Release 2 of the *WHIP!* Browser Accessory. *WHIP!* is available as a Netscape Navigator plug-in and as a Microsoft Internet Explorer ActiveX control. For information on downloading and using *WHIP!*, please navigate to *http://www.autodesk.com.*

RAM and Hard Disk Space

- 16 MB of RAM (16 MB minimum, 32 MB recommended)
- 34–56 MB minimum of hard disk space (depends on configuration)
- 64 MB of disk swap space (64 MB minimum, 100 MB recommended)
- 8 MB of additional RAM for each concurrent AutoCAD LT session
- 5 MB of free disk space during installation only (this space is used for temporary files that are removed when installation is complete)

NOTE An additional 10 MB to 14 MB of space may be required for files installed in your Windows system folder (for example, *c:\windows\system*), which can be on a different drive than the folder you specify as the Destination Location (for example, *d:\acadlt97*).

Before Installing AutoCAD LT

To prepare for installing AutoCAD LT

1 Make sure you have write permission in the following locations:

- Folder where you are installing AutoCAD LT
- Windows system folder
- System registry
  If you are installing on Windows NT, make sure you have Administrator privileges. You must have permission to write to the necessary system registry sections. See "Using User Manager to Manage Your Computer's Security" in the Windows NT online Help, which you can access by clicking Start on the taskbar.

2 Ensure that you meet the minimum system requirements as specified in the previous section.

3 Start Windows and close any open applications.

Installing AutoCAD LT 97 for a Single User

The following procedure installs AutoCAD LT for a single user. You can cancel the setup process at any time by pressing F3 or ESC.

To install AutoCAD LT for a single user

1 Insert the CD into the CD-ROM drive.

- Autorun begins the installation process as soon as you insert the CD unless you hold down the SHIFT key while you insert the CD.
- If you have turned off Autorun, or if you have installed AutoCAD LT from this CD before, from the Start menu, choose Run, designate the CD-ROM drive and enter **setup**. For example, enter **d:\setup.**

2 When the Welcome screen is displayed, choose Next. Follow the prompts.

If the installation program finds a version of AutoCAD LT 97 registered on your system, it prompts you to specify whether you want to add or reinstall AutoCAD LT. For more information, see "Adding Components" on page 37 and "Reinstalling AutoCAD LT 97" on page 37.

You are prompted to specify the type of installation you want:

- *Typical* installs the following files:

  Program files (executables, menus, toolbars, help templates, TrueType fonts, and additional support files)
  Learning tools (Quick Tour and What's New executables, drawings and support files)
  Samples (sample drawings and images)
  English dictionary (American and British versions)
  Help (Help and online help files)

- ***Compact*** installs the executables, support files, and online Help only.
- ***Custom*** installs the files you specify. By default, the Custom installation option installs all AutoCAD LT components. Clear the check boxes beside the AutoCAD LT components you do *not* want to install. For example, you can choose not to install the folder containing sample drawings.

  Custom includes the following additional options:

  OLE/ADI Plot Use this printing feature if you want to print embedded OLE objects with ADI printer or plotter drivers. For details, see "Linking and Embedding Data (OLE)" in online Help.

  French-Canadian dictionary Use if you want to include this dictionary for checking spelling in your drawings.

3 Online Help requires version 3.02 of Internet Explorer or higher to run. The install program detects this Web browser on your computer. If the appropriate software is not detected, the install program installs Internet Explorer version 3.02. Choose Cancel to exit the install program.

NOTE If you attempt to access online Help from the Help menu and the appropriate files or version of Internet Explorer are not properly installed on your computer, a warning message is displayed, prompting you to run the install program again.

4 Content Explorer is a tool that lets you search, view, and organize drawing content on a project-by-project basis. You are prompted to specify the default tab set of symbols displayed in Content Explorer:

- ***Architectural*** Includes landscaping, house design, and kitchen design symbols
- ***Mechanical*** Includes welding, pipe fitting, and plant process symbols
- ***Electrical*** Includes CMOS-IC and electrical power symbols

When the installation is complete, you are prompted to view the *readme.hlp* file. This file provides information that was not available at the time AutoCAD LT documentation was printed. You can access the *readme.hlp* file at any time from the Help folder of the program folder you specified during installation.

You may be prompted to restart your computer to complete the installation. Restarting your computer is highly recommended to ensure a successful AutoCAD LT installation.

Upgrading from a Previous Release

Installing the upgrade version of AutoCAD LT 97 requires that you previously installed a full commercial release of AutoCAD LT (release 1, release 2, or AutoCAD LT 95) on a local or network drive. The installation program searches for the AutoCAD LT executable file (*aclt.exe*). If a dialog box warns you that a valid executable file could not be found, click Browse to search for the file. If the search is successful, the installation program continues as outlined in the procedure, "To install AutoCAD LT for a single user."

If you have not previously installed a full commercial release of AutoCAD LT (release 1, release 2, or AutoCAD LT 95) on a local or network drive, you cannot install the upgrade version of AutoCAD LT 97. Contact your vendor for the full version.

Overwriting a Previous Version of AutoCAD LT

Installing AutoCAD LT 97 to a *different* folder from your previous version of AutoCAD LT is highly recommended. If you install the upgrade version of AutoCAD LT 97 in the *same* program folder as your previous version of AutoCAD LT, the install program prompts you to uninstall AutoCAD LT 95. To ensure that you do not lose files that you have customized, such as menu files, copy them to a temporary folder before installing AutoCAD LT 97.

NOTE If you install the upgrade version of AutoCAD LT 97 in the same program folder as AutoCAD LT release 1 or release 2, no attempt is made to uninstall these previous releases.

Reusing Customizable Support Files

If you have customized support files from previous releases of AutoCAD LT, you can reuse the information in AutoCAD LT 97. For example, you can reuse customized hatch patterns previously saved in the hatch definition file (*aclt.pat*) by copying and pasting the customized definitions into the newly installed file using a text editor.

NOTE You should *not* replace the newly installed support files with versions from a previous release of AutoCAD LT.

The following table lists the customizable support files you can reuse.

Customizable support files you can reuse with AutoCAD LT 97

File name	Description
aclt.pat	Library file that contains standard hatch pattern definitions
acltiso.pat	Library file that contains metric hatch pattern definitions
aclt.lin	Library file that contains standard linetype definitions
acltiso.lin	Library file that contains metric linetype definitions
aclt.mnu	Menu template file that contains menu definitions and comments; generally used for reference
aclt.mnc	Compiled binary menu file that contains the command strings and menu syntax that define the functionality and appearance of the menu
aclt.mnr	Menu binary resource file that contains the bitmaps used by the menu
aclt.mns	Source menu file that can be customized by TBCONFIG or by using a text editor
aclt.pgp	The program parameters file is an ASCII text file that stores command alias definitions
aclt.dwt	Template drawing file based on feet and inches measurements
acltiso.dwt	Template drawing file based on metric measurements
aclt.unt	Text file that contains English and metric unit definitions and equivalents

ONLINE HELP See "Controlling the User Interface," "Customizing Menus, Toolbars, and Buttons," "Customizing Linetypes," and "Customizing Hatch Patterns."

Adding Components

You can add custom components at any time by running the AutoCAD LT 97 installation program again and choosing Add in the Setup Choices dialog box. This option is available only if you've already installed AutoCAD LT from this CD. Components that have not been previously installed are checked; uncheck the components that you do *not* want to add, and then proceed with the installation by following the prompts.

The install program prompts you to specify the customizable support files you want to save in a backup folder before proceeding with the installation of additional components.

Reinstalling AutoCAD LT 97

If you accidentally delete or alter files required by AutoCAD LT 97, you can reinstall files by choosing Reinstall in the Setup Choices dialog box. This option (available only if you've already installed AutoCAD LT from this CD) reinstalls the components you previously installed in the program folder you originally specified.

The install program prompts you to specify the customizable support files you want to save in a backup folder before proceeding with the reinstallation.

Uninstalling AutoCAD LT 97

Uninstalling removes all program files that were part of the installation from this CD, including any program components that were added later. Files that were created after installation of AutoCAD LT 97 are *not* removed. This includes *aclt4.cfg* and any drawing files that you have created in the AutoCAD LT 97 folder tree. You can manually delete these files after uninstalling AutoCAD LT 97 if you have no further use for them.

To uninstall AutoCAD LT 97, use one of the following procedures:

To uninstall AutoCAD LT 97 from the Programs menu in Windows 95 or NT

1 From the Windows Start menu, choose Programs. Then choose AutoCAD LT 97 (or the folder you used to install AutoCAD LT).
2 Choose Uninstall AutoCAD LT 97.

To uninstall AutoCAD LT 97 from the Control Panel in Windows NT

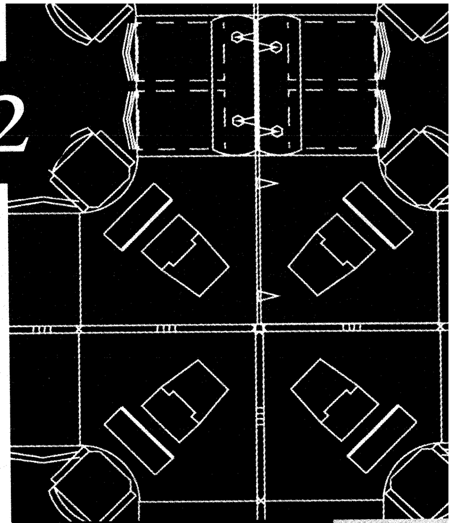
1 From the Windows Start menu, choose Settings ➤ Control Panel.
2 In the Control Panel, choose Add/Remove Programs.

On the Install/Uninstall tab in the Add/Remove Programs Properties dialog box, choose AutoCAD LT 97.

NOTE You cannot run AutoCAD LT when attempting to uninstall. The uninstallation program checks that AutoCAD LT is not running before proceeding to uninstall it.

When AutoCAD LT has been successfully uninstalled, you are strongly recommended to restart your computer to ensure that any files that may have been in use by your operating system are properly uninstalled. Choose Cancel if you don't want to restart your computer immediately.

2



Getting Started

This chapter describes the fundamentals that you need to know to use AutoCAD LT. It explains how to set up the drawing environment and use the AutoCAD LT interface.

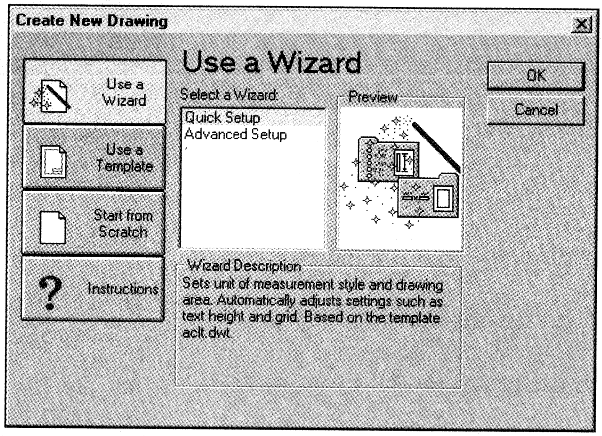
In this chapter

- Starting AutoCAD LT
- Understanding the AutoCAD LT interface
- Getting assistance while you work
- Using pointing devices
- Choosing a printer or plotter
- Using commands and system variables
- Setting AutoCAD LT preferences

Starting AutoCAD LT

To start AutoCAD LT, click Start on the Windows taskbar, then choose Programs. Choose the AutoCAD LT 97 folder, then choose AutoCAD LT 97 from the menu.

In the Create New Drawing dialog box, make one of the following choices to set up a new drawing:

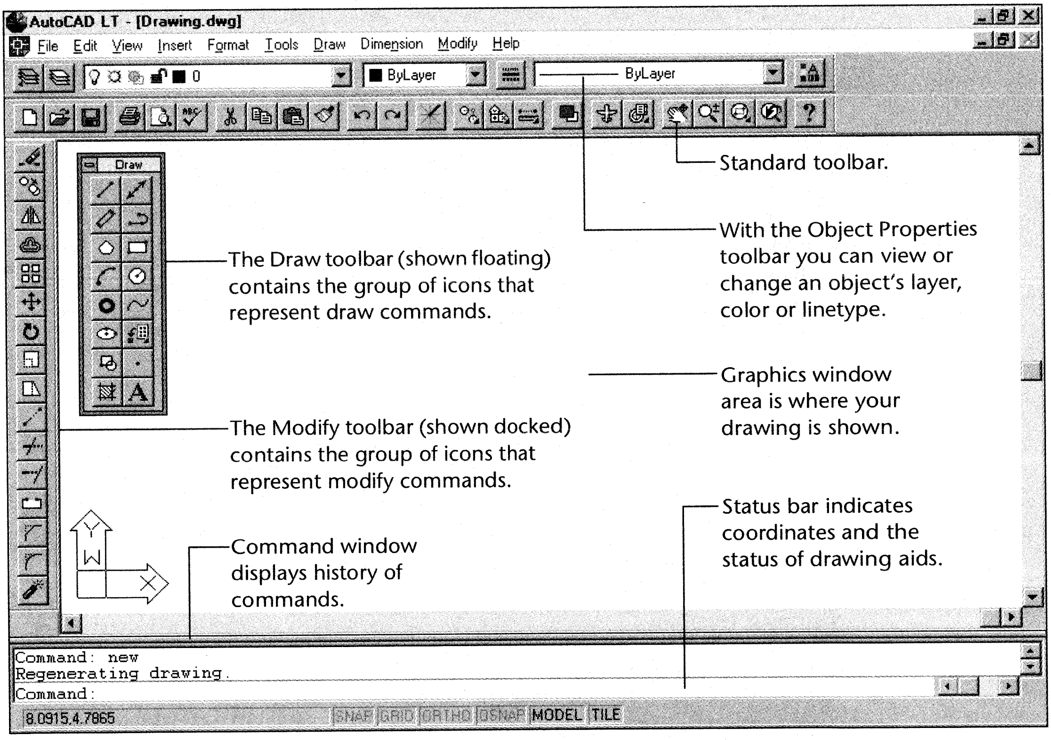


- ***Use a Wizard*** Under Select a Wizard, choose Quick Setup or Advanced Setup to use AutoCAD LT automatic setup features.
- ***Use a Template*** Choose a template to establish your drawing settings.
- ***Start from Scratch*** Select one of the two measurement systems.

For more information on using start-up wizards, or templates or starting from scratch, see chapter 3, "Organizing Your Project."

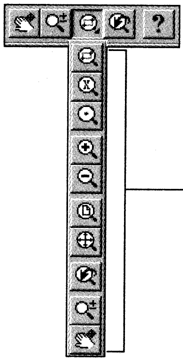
Understanding the AutoCAD LT Interface

When you first start AutoCAD LT, the initial screen contains the menu bar at the top, the status bar at the bottom, the drawing window, the command window, and several toolbars.



Toolbars

Toolbars contain tools, represented by icons, that start commands. When you move the pointing device over a tool, a tooltip displays the name of the tool. Tool icons with a small black triangle in the lower-right corner have flyouts that contain related commands or command options. With the cursor over the tool icon, hold down the pick button to display the flyout. If you choose an icon from the flyout, it replaces the original icon on the toolbar.



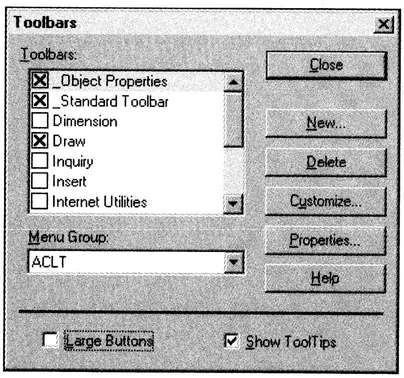
the Zoom flyout

The Standard toolbar at the top of the graphics area is visible by default. It contains frequently used AutoCAD LT tools such as Aerial View, Pan, and Zoom, as well as Microsoft Office standard tools such as Open, Save, Print, and Spell Check. You can display multiple toolbars on screen at once, change their contents, and resize them.

To display a toolbar

■ Choose View ➤ Toolbars

Click the box beside the toolbar you want to display.



Control the size of toolbar buttons and whether tooltips are displayed.

You can "dock" or "float" a toolbar. A docked toolbar is attached to any edge of the graphics area. A docked toolbar cannot be resized and does not overlap the graphics area. You can float a toolbar by dragging it anywhere in the graphics area of the AutoCAD LT window. Floating toolbars can be resized by grabbing any edge with the cursor.

To use a tool from a toolbar

- Select a tool from a toolbar, and then select options in a dialog box or respond to prompts displayed on the command line.

ONLINE HELP See "Customizing Menus, Toolbars, and Buttons."

Pull-Down Menus

The menus are available from the menu bar at the top of the AutoCAD LT window. You can choose menu commands in *any* of the following ways:

- After you click the menu name to display a list of commands, click the command to choose it.
- Hold down ALT and then enter the underlined letter in the menu name. For example, hold down ALT while pressing F (ALT+F) to open the File menu. Then enter the underlined letter of the menu command. For example, press A to choose Save As.
- Use shortcut keys to choose menu commands directly. For example, CTRL+P chooses Print.

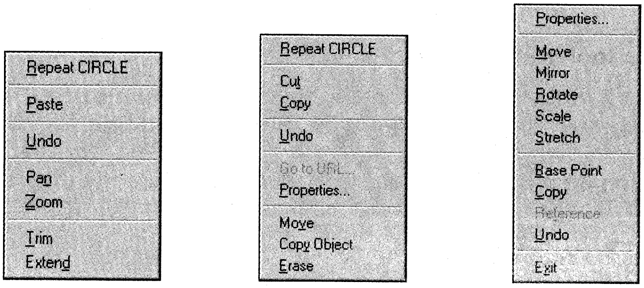
The default menu file is *aclt.mnu*. You can specify a different menu (for example, a menu you have customized) on the Files tab in the Preferences dialog box.

Shortcut Menu

You can set AutoCAD LT to display a shortcut menu when you right-click your pointing device (see "Setting General Operating Preferences" on page 58). The commands in the shortcut menu change, depending on whether you right-click in the graphics area, text window, or command window.

The shortcut menu displayed in the graphics area is context sensitive; the list of commands changes if you have objects selected or are using grip commands:

■ ***Objects selected*** The shortcut menu displays the standard editing commands, and you can view the properties of the selected objects.
■ ***Grips selected*** The shortcut menu lists the grip editing commands, and you can view the properties of the selected objects.



no object selected | object selected | grips selected

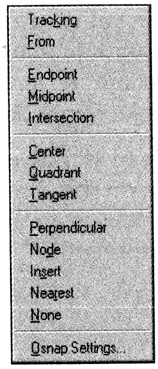
Examples of shortcut menus in the graphics area

If you have entered text on the command line, the right-click button is equivalent to pressing ENTER on your keyboard. For example, to draw a circle you can enter **c** on the command line, and then press ENTER to start the command.

NOTE If you change the PICKFIRST system variable setting to 0, the option shortcut menu is unavailable; the right-click button is equivalent to pressing ENTER on your keyboard.

Cursor Menu

The cursor menu is displayed at your cursor location when you hold down SHIFT and press the return button on the pointing device. On a two-button mouse, the return button is usually the right button. On a three-button mouse or an IntelliMouse, you can use the middle button or wheel button to display the cursor menu. See "Using the IntelliMouse" on page 53.



By default, the cursor menu lists object snaps and tracking (see “Snapping to Geometric Points on Objects” on page 124). If you want to change the options, you can customize the cursor menu.

ONLINE HELP To customize menus and toolbar, see “Customizing Menus, Toolbars, and Buttons.”

Command Line

The command line provides a method of command entry and also displays additional command options, feedback, and prompts during the command process. For example, when you enter **circle** at the Command prompt, the following prompt is displayed.

3P/TTR/<Center point>:

The brackets around Center Point indicate that it is the current option. You can choose the Center Point option either by entering an *X,Y* coordinate or by selecting (clicking) a point on the screen.

To choose a different option, enter the capitalized letters of the option name. You can enter uppercase or lowercase letters. For example, to choose the three-point option (3P), enter **3p**.

NOTE To execute commands, press SPACEBAR or ENTER after entering command names or responses to prompts. This guide does not specifically instruct you to press ENTER after each entry you make on the command line.

You can run any command by entering its full name at the Command prompt. Some commands also have abbreviated names or *aliases*. Instead of entering **circle** to start the CIRCLE command, for example, you can enter **c**. For a partial list of command aliases, see the *Quick Reference Card*.

Use the standard text-editing keyboard commands in Windows to navigate and modify command line text. You can delete text on the command line by pressing either BACKSPACE or DEL. You can paste text from the Clipboard, but you cannot select text to cut or copy to the Clipboard.

Repeating and Canceling Commands

If you want to repeat a command that you have just used, press ENTER OR SPACEBAR at the Command prompt. You can set the right button on your pointing device to be equivalent to pressing ENTER (see "Setting General Operating Preferences" on page 58). You also can repeat a command by entering **multiple** on the command line and then the command name, as shown in the following example.

Command: **multiple**
Multiple command: **circle**

The default method for canceling any command is to press ESC. You can also use CTRL+C by changing settings in the Preferences dialog box.

If you want to reissue a command you used during the current AutoCAD LT session, you can access it in the command history above the command line.

To repeat any command used in the current session

1 Make sure your cursor is on the command line, then locate the line containing the command you want to issue by pressing UP ARROW or DOWN ARROW. The previously entered commands appear in succession on the command line.

   If you want to view the command history as you search, activate the text window by pressing F2.

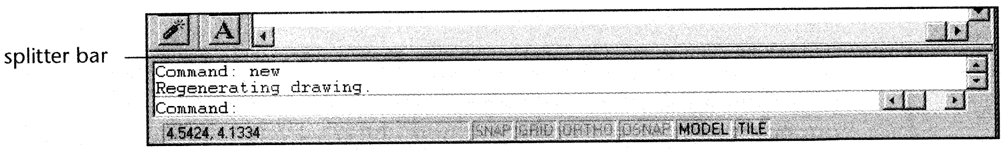
2 After locating the command, run the command by pressing ENTER.

Command Window

The command window is a dockable window in which you enter commands or review prompts and messages. For most commands, a command line with two or three lines of previous prompts, called the command history, is sufficient. For commands with text output, such as LIST, you might need to make the command window larger. When the command history has more than one line, you can scroll through it with scroll bars.

By default, the command window is docked below the graphics area and is the width of the AutoCAD LT window. If text that is entered becomes longer than the width of the command line, the window pops up in front of the command line to show the full text of the line.

You can resize the window vertically using the splitter bar, which is located on the top edge of the window when the window is docked at the bottom and on the bottom edge when it's docked at the top. Resize by grabbing the splitter bar with your pointing device and dragging the window to the required height.



Docked command window

Undock the command window by selecting any part of its border and dragging it away from the docking region. Drop the window to make it a floating window. When you undock the window, the floating window is the size it was the last time you undocked it. You can move the floating command window anywhere on the screen and change its width and height with the pointing device.

Dock the command window by dragging it until it is over the top or bottom docking regions of the AutoCAD LT window.

Text Window

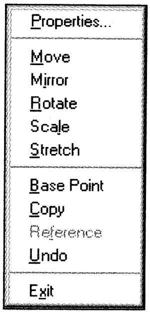
The text window is a larger version of the command window in which you enter commands and view prompts and messages. The text window contains up to 200 lines of command history and is useful when viewing lengthy output of commands such as LIST, which displays detailed information about objects you select.

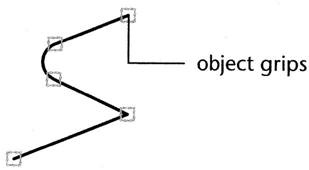
To display the text window while you are in the graphics area, press F2. The text window is displayed in front of the graphics area. If you press F2 while in the text window, the graphics area is redisplayed. If either the graphics area or the text window has been minimized, press F2 to display it at its last configured size.

NOTE The contents of the text window are read-only and cannot be modified.

Using Grip Commands

One of the easiest ways to manipulate objects is to use *grips* and grip commands. Grips mark control locations on a selected object; these points provide a powerful editing tool. *Grip commands* are accessible from a shortcut menu and include the most frequently used commands for modifying objects.





The shortcut menu is displayed if you set AutoCAD LT to display a menu when you right-click your pointing device. By accessing grip commands with your pointing device, you can combine object selection and editing, without having to use menus or the command line. For more information, see "Editing with Grips" on page 151.

Using Two Commands at the Same Time

Many commands can be used *transparently*, which means they can be entered on the command line while you are using another command. Transparent commands frequently are commands that change drawing settings such as ZOOM or that turn on drawing aids such as SNAP or GRID.

To use a command transparently

- Enter an apostrophe (') before entering a command at any prompt
- Choose a tool or menu

On the command line, double angle brackets (>>) precede prompts for transparent commands. After you complete the transparent command, the original command resumes.

The following example illustrates turning on the grid and setting it to one-unit intervals while you are drawing a line.

Command: **line**
From point: **'grid**
>>Grid spacing (X) or ON/OFF/Snap/Aspect <0.000>: **1**
Resuming LINE command
From point: *Continue drawing the line*

Switching from Dialog Box to Command Line

Some functions are available both on the command line and in a dialog box. You can force AutoCAD LT to display prompts on the command line instead.

To switch command functions to the command line

- Enter a hyphen before the command to suppress the dialog box

For example, entering **layer** at the command line displays the Layer & Linetype Properties dialog box, in which you can define layer and linetype properties. Entering **-layer** at the command line displays the equivalent command line options for defining layer and linetype properties. This may be useful for creating automatic scripts, text files that can be used for executing a sequence of commands

NOTE There may be slight differences between the options in the dialog box and those available on the command line.

Using Scripts to Run Commands

A script is a series of AutoCAD LT commands in a text file that can be used to carry out a task. With scripts, you can run several commands in succession. For example, you can script a series of AutoCAD LT commands for overnight batch processing. You can also create a group of drawing display files known as *slides* and then write a script that displays them in a slide show. For slide shows, you can specify a delay in the running of the script, resume scripts that you have canceled, or stop a script and restart it at the beginning.

ONLINE HELP To find specific command information, see "Command Reference." To use scripts, see "Using Slides and Scripts."

Getting Assistance While You Work

At any time during an AutoCAD LT session, you can access online information from the Help menu. Use Help Topics for assistance as you work. Browse the Table of Contents of the Help Topics to locate the information you need, or look up index entries to pinpoint specific topics.

To access Help topics

Standard Toolbar	Menu	Command
	Help ➤ AutoCAD LT Help Topics	HELP

Related Press F1 or enter **?** on the command line.

If you are unable to access the online information, run the AutoCAD LT install program to install it. Online Help requires version 3.02 of Internet Explorer or higher to run. The AutoCAD LT install program installs it if the appropriate software is not detected on your computer. For details, see "Adding Components" on page 37.

ONLINE HELP For more detailed information about using the online documentation, see "Welcome to the AutoCAD LT Help System."

If you are not sure what a toolbar button does, display a *tooltip* to identify it. Move your cursor over a toolbar button and a tooltip is displayed. A brief explanation is also displayed on the status bar. Tooltips are on by default.

To turn on tooltips

1 Choose Toolbars from the View menu.
2 Select the Show ToolTips checkbox.

You can also get explanations about parts of dialog boxes by using either of the following methods:

- Click the question mark in the upper-right corner of the dialog box. When the question mark appears on the cursor, click the item you want information on. Click again to dismiss the information.
- Move your cursor over the dialog box feature, such as a list, and right-click. Click again to dismiss the information.

Using Pointing Devices

You can control AutoCAD LT with a wide range of pointing devices, such as a standard mouse, IntelliMouse, or digitizing tablet. You can specify points either by clicking the pointing device or by entering coordinates from a keyboard. A pointing device, such as a mouse or a digitizing puck, may have a number of buttons. The first 10 buttons are automatically assigned by AutoCAD LT, but you can reassign all but the pick button by modifying the menu file (*aclt.mnu*).

ONLINE HELP See "Customizing Menus, Toolbars, and Buttons."

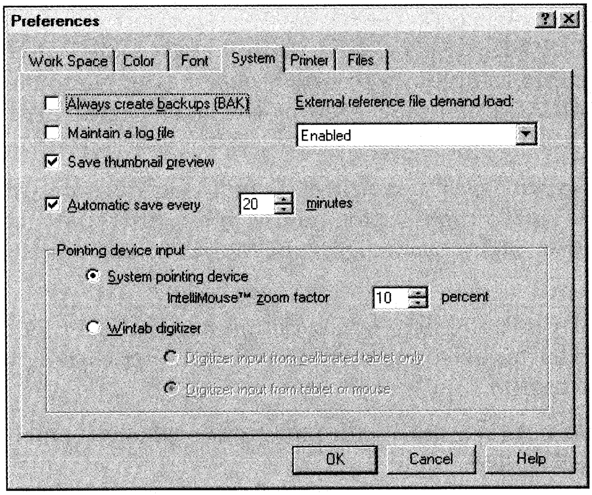
Configuring the Pointing Device

After you install AutoCAD LT you don't need to perform additional configuration tasks on the mouse or digitizer; AutoCAD LT uses the current Windows system pointing device. To change the current pointing device, use the options on the System tab in the Preferences dialog box. You can also control whether AutoCAD LT accepts input from a digitizer only or from both a digitizer and a mouse. For more information on calibrating a digitizer, see "Calibrating the Tablet" on page 424.

To adjust pointing device settings

1 From the Tools menu, choose Preferences.

2 In the Preferences dialog box, choose the System tab.



3 Under Pointing Device Input, select System Pointing Device or Wintab Digitizer.

4 If you selected System Pointing Device, and the IntelliMouse is your current pointing device, specify the *zoom factor* when using the IntelliMouse wheel for zooming. The zoom factor is expressed as a percentage zoom change when the IntelliMouse wheel is rotated one discrete increment. See "Using the IntelliMouse" on page 53.

5 If you selected Wintab Digitizer as your pointing device, select *one* of the following:

- ***Digitizer input from calibrated tablet only*** Specifies that AutoCAD LT accepts input from a digitizer only and ignores the mouse.
- ***Digitizer input from tablet or mouse*** Specifies that AutoCAD LT accepts input from either the digitizer or the mouse, whichever moved last or sent the latest coordinate setting.

6 Choose OK to record the current pointing device settings in the system registry and close the dialog box.

Using the Mouse

You can choose menu options and tools by clicking them with a mouse. You also can use the mouse to draw, select, or modify objects on the screen.

On a two-button mouse, the left button is the *pick* button, used to specify points or select objects on the screen. The right button is the *return* button. With AutoCAD LT, you can set the return button to perform two different functions: either to display a *shortcut menu*, or as an alternate way to press ENTER (AutoCAD classic). See "Setting General Operating Preferences" on page 58.

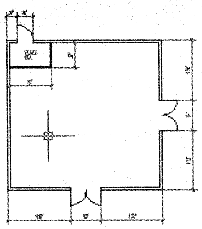
If you hold down SHIFT and right-click in the drawing area, the *cursor menu* is displayed. With a three-button mouse, you can set the middle button to open the cursor menu by changing the MBUTTONPAN system variable setting to 0.

In some situations, the right mouse button has a special function. For example, you can customize the tools in the toolbar after clicking them with the right mouse button.

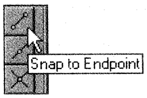
ONLINE HELP To reassign mouse buttons, see "Reset Your Primary Mouse Button" in Windows Help.

To practice using the mouse

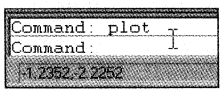
1 Move your mouse and notice how the pointer on the screen changes from crosshairs while the pointer is in the graphics area, to an arrow when it's *not* over the graphics area, and to an I-beam when it's in the text window or on the command line.



crosshairs pointer in the graphics area

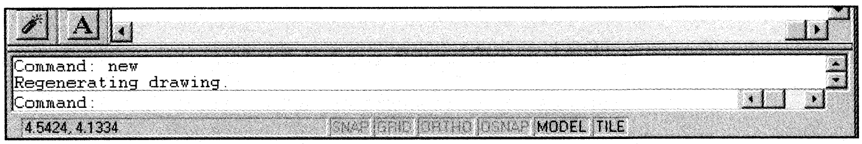


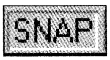
arrow pointer over a toolbar



I-beam cursor in the command window

As you continue to move the mouse, notice how the numbers change in the coordinate display on the status bar. These numbers indicate the exact location, or coordinate, of the crosshairs on the screen.





2 Find Snap on the status bar and double-click it with the pick button on your mouse (usually the left button). Notice that Snap darkens to indicate that snap has been turned on.

   Move the pointer around the screen and observe how it seems to adhere, or "snap," to points at predetermined and equivalent intervals on the screen. You can change these intervals.

3 Double-click Snap on the status bar again to turn snap off.

4 Move the pointer over the Standard toolbar at the top of the graphics area. As you leave the cursor over a tool icon for a few moments, notice a pop-up label, called a tooltip, that identifies the tool.

Using the IntelliMouse

The IntelliMouse is a two-button mouse with a small wheel between the buttons. This wheel can be rotated by discrete values, which are controlled by a series of *detents*. You can use the wheel to zoom and pan your drawing, without using any AutoCAD LT commands. The following table describes the IntelliMouse mouse actions supported in AutoCAD LT.

Using the IntelliMouse with AutoCAD LT	
To...	**Do this...**
Zoom in or out	Rotate the mouse wheel forward to zoom in, backward to zoom out
Zoom in or out (optional method)	Hold down CTRL, then rotate the wheel forward to zoom in, backward to zoom out (optional method)
Pan	Hold down the wheel button and drag the mouse
Open the cursor menu	Change the MBUTTONPAN system variable setting to 0. Click the wheel button.

Setting the IntelliMouse Zoom Factor

When you rotate the IntelliMouse wheel, the amount by which the zoom increases or decreases is controlled by the *zoom factor.* You can set the zoom factor on the System tab of the Preferences dialog box. By default the zoom factor is set to 10%; each increment in the wheel rotation changes the zoom level by 10%.

For more information about zooming and panning, see "Using Zoom and Pan" on page 196.

ONLINE HELP To use the Microsoft IntelliMouse, see "The IntelliMouse Wheel and Wheel Button" in Windows Help.

Using the Tablet

You can use a digitizing tablet to select frequently used commands, to select objects on the screen, or to draw. The tablet's pointing device, which you use for selection, can be a *puck* or a *stylus*. The crosshairs on the screen follow the movement of the pointing device in the screen pointing area of the tablet.

You can also use a tablet to digitize drawings by tracing objects into the AutoCAD LT drawing database using coordinates that relate to the original drawing. For example, if you are working with a printed circuit design that was originally prepared by hand, you can now store and edit that drawing in AutoCAD LT.

To digitize a drawing, you *calibrate*, or align, the tablet with the paper drawing's coordinate system. If the paper drawing is too big to fit on the tablet, you can enter the drawing in pieces, making sure each piece is correctly aligned.

AutoCAD LT has a special input mode called Tablet mode. In Tablet mode the active area of your tablet coincides with the current display window. When Tablet mode is off, the screen-pointing area of your tablet coincides with specific coordinates in your drawing; the portion of your drawing that you are currently viewing is irrelevant. Any command that requires you to select objects with the pointing device works in Tablet mode. The TABMODE system variable turns Tablet mode on and off. For more information about configuring and calibrating tablets, see appendix C, "Configuring a Digitizing Tablet."

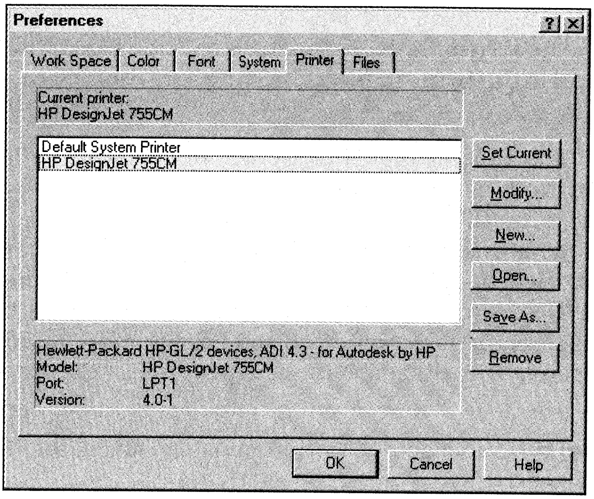
ONLINE HELP To configure a Wintab-compatible tablet, see "Setting Up a Digitizing Tablet."

Choosing a Printer or Plotter

After you install AutoCAD LT you don't need to perform additional configuration tasks on the printer or plotter; AutoCAD LT uses the current Windows system printer. However, AutoCAD LT uses ADI (Autodesk Device Interface) plotter and printer drivers to interface with supported hardcopy output devices. You can change the current printer or plotter using the options on the Printer tab in the Preferences dialog box. You can also load printers and plotters and change the settings of existing printers and plotters. For information on loading printers and plotters, see chapter 13, "Plotting Your Drawing." For more information on configuring plotter and printer drivers, and a list of supported output devices, see appendix B, "Configuring Plotters and Printers."

To set printer preferences

1 From the Tools menu, choose Preferences.
2 In the Preferences dialog box, choose the Printer tab.



3 On the Printer tab, select the printer or plotter you want to make current from the list of configured printers and plotters.

4 Choose Set Current.

5 Choose OK to update the current printer.

Using System Variables

System variables are settings that control how certain commands work. They can turn on or off drawing aids such as Snap, Grid, or Ortho. They can set default scales for hatch patterns. They can store information about the current drawing and the AutoCAD LT configuration. Sometimes you use a system variable to change a setting; sometimes you use a system variable to display the current status.

For example, the GRIDMODE system variable turns the dot grid display on and off. In this case, the GRIDMODE system variable is functionally equivalent to the GRID command. DATE is a read-only system variable that stores the current date. You can display this value, but you cannot change it.

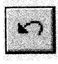
You can examine or change a variable's setting transparently, that is, while using another command; however, new values may not take effect until the interrupted command ends.

ONLINE HELP See "Entering System Variables."

Correcting Mistakes

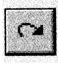
With AutoCAD LT you can undo your most recent action or actions using any of several methods. The simplest is to use the UNDO command to undo a single action. Or you can use the UNDO command to undo a specific number of actions.

To undo the most recent action

Standard Toolbar	Menu	Command
	Edit ➤ Undo	UNDO

To redo U or UNDO, you must use REDO immediately after using U or UNDO.

To redo an action

Standard Toolbar	Menu	Command
	Edit ➤ Redo	REDO

Refreshing the Screen Display

AutoCAD LT provides the option to display *blips*, or temporary markers that indicate points you specify as you draw. To remove blips, you refresh the display by choosing to either *redraw* or *regenerate*. Redrawing cleans up only the display. Regenerating, however, not only cleans up the display, but also updates the drawing database with the screen coordinates for all objects in the drawing. Because regeneration can take a long time in complex drawings, you will usually redraw.

To redraw the screen

Menu	Command
View ➤ Redraw	REDRAW

Related To force AutoCAD LT to redraw, toggle the grid on and off by pressing F7 twice.

To regenerate a drawing

Menu	Command
View ➤ Regen	REGEN

Setting General Operating Preferences

Many of the settings that affect the AutoCAD LT interface and drawing environment are located in the Preferences dialog box.

To set preferences

Menu	Command
Tools ➤ Preferences	PREFERENCES

The various options are grouped according to the following tabs:

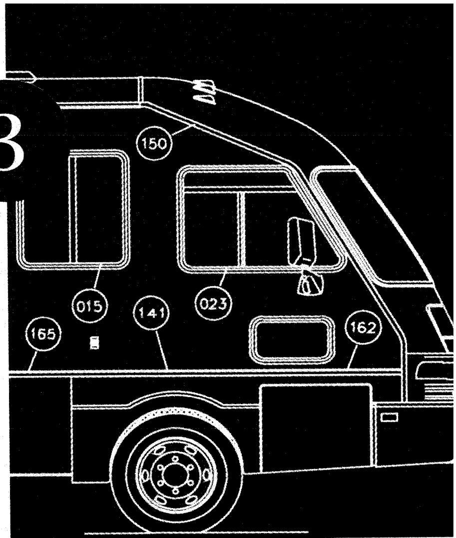
- ***Work Space*** Specifies different aspects of the drawing environment so you can tailor it to your working style. Experiment with the settings in the Preferences dialog box until you find the most suitable environment.
- ***Color*** Specifies the colors used by the AutoCAD LT Window to suit your personal taste. You can change the color of the graphics area, the text window and command window backgrounds, the text in the command window and text window, and the crosshairs.
- ***Font*** Specifies the fonts used in the AutoCAD LT command window and in the text window. This setting does not affect the text in your drawings.
- ***System*** Specifies how your system works, including whether AutoCAD LT creates backup files of your work, how frequently your work is saved, which pointing device is used, and how you want AutoCAD LT to load externally referenced drawings.
- ***Printer*** Specifies the current printer or plotter configuration. You can also load, modify, and remove configured printers and plotters.
- ***Files*** Specifies the search path AutoCAD LT uses to find support files such as text fonts, drawings, linetypes, and hatch patterns. This option helps improve performance when AutoCAD LT is loading these files. You can also specify other settings, such as a log file and custom dictionary.

Specific details about preference settings are described in the relevant sections throughout the AutoCAD LT documentation.

ONLINE HELP To control general preferences, see "Setting Interface Preferences." To control AutoCAD LT using switches, see "Starting AutoCAD LT Using Command Line Switches."

3

Organizing Your Project



Many of the fundamental differences between paper and CAD become apparent in the planning stages. Tasks that you must do first on paper, such as laying out your drawing and deciding on a scale, can be saved until you are ready to plot (print) with AutoCAD LT. Organizing your drawing project begins by establishing layer, linetype, and scale factor settings. After the drawing is complete, you can refine the layout and then plot to the scale you require.

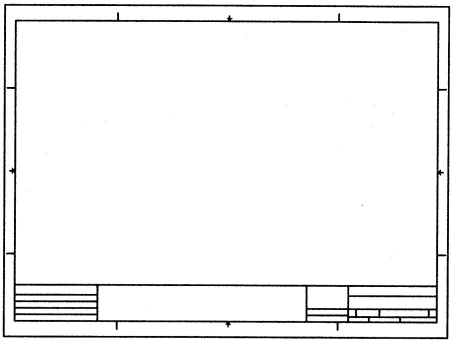
In this chapter

- Opening an existing drawing
- Beginning a new drawing
- Units style settings
- Scale factor settings
- Drawing boundary settings
- Grid and snap settings
- Creating layers to organize your drawing
- Saving a drawing or template drawing

Conforming to Standards

Drawing standards come from many sources. Perhaps you are following standards dictated by a client or by industry requirements, or maybe you are establishing your own standards. Whether you are handling all aspects of the project yourself or supervising it, the initial drawing setup is a key factor in producing a professional drawing.

For example, an architectural drawing might consist of several elements: a floor plan, piping, and heating and air conditioning. If you use different contractors to draft each of these elements, you want them to adhere to the same standards as they produce their parts of the drawing. You can do this by providing a template file containing drawing setup parameters. An initial drawing setup includes border, title block, units of measurement, layering, and linetypes.



typical title block and border

Equally important are the styles used for text, hatching, and dimensions. By setting up styles for these in advance, you can ensure that everyone working on the drawing is conforming to your established standards.

You should also consider the scale. By choosing a scale factor—the size of what's being drawn versus the size of the plotted drawing—you can help ensure that the lettering for annotations and dimensions is appropriate for the final scale of your project.

Although you can delay laying out the final drawing until later in the project, you should think about it ahead of time. This planning may involve using model space, where you create your drawing, and paper space, where you arrange the drawing layout for plotting. In paper space, you apply scaling factors to ensure that the final plotted product is at the correct scale with respect to text, linetypes, dimensions, and drawn entities.

The work you put into setting up the drawing can be saved as a template and reused for similar drawings or altered to accommodate slightly different needs.

Opening Existing Drawings

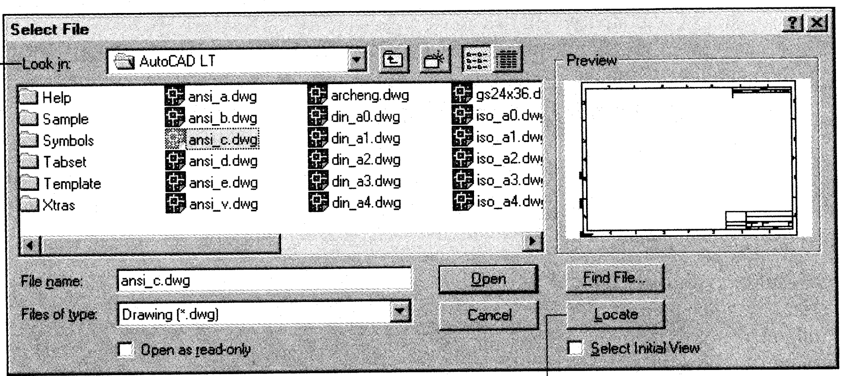
To open an existing AutoCAD LT drawing, you choose its name from a list. If you already have a drawing open, AutoCAD LT prompts you to save that drawing when you open another one because you can have only one drawing open at a time.

To open a drawing

Standard Toolbar	Menu	Command
	File ➤ Open	OPEN

Specify the file location.

Double-click a file name to open a drawing or enter a file name.



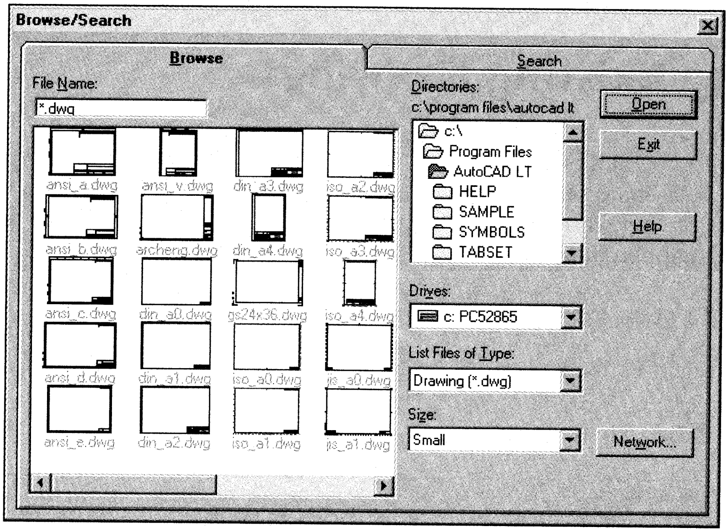
Enter a file name, then click Locate; AutoCAD LT searches for the specified file's path and folder.

Browsing Through Files

Use the drawing browser to view small images of drawings, to open drawings, and to search for files. You can use the browser to search for files across multiple directories on a single drive or on multiple drives.

The Browse/Search dialog box, accessed by clicking Find File, displays small images of drawings in the directory you specify. You can sort the images by file type. Click an image to select it.

You can change the size of the images by choosing from the Size list on the Browse tab in the Browse/Search dialog box.

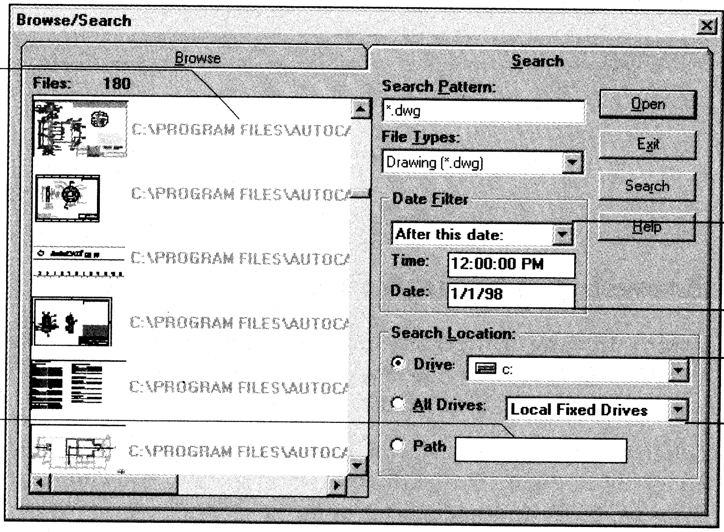


Specify the directory you want to browse.

Double-click a file image to open it.

Searching for Files

You can search for files by specifying a file type and referencing the date of creation. For example, you could search for *.dwg* files that were created after 3:00 P.M. on a certain day.



Search results are listed with full path and image of drawing.

Searches by date of file creation.

Lists local fixed drives and removable network drives.

Specifies a path.

Setting Up New Drawings

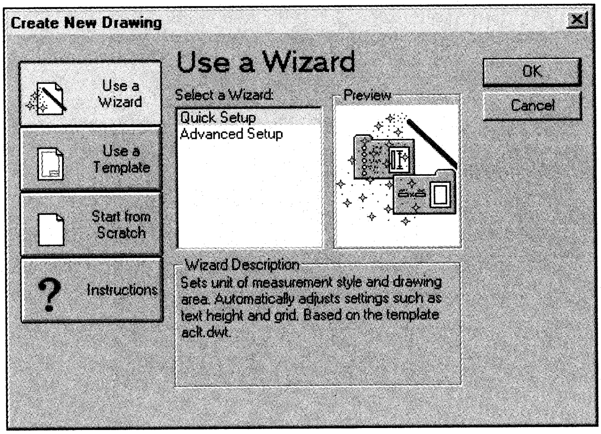
When you create a new drawing, you can choose one of four options in the Create New dialog box:

- ***Use a Wizard: Quick Setup*** Establishes basic settings of measurement units and area; automatically adjusts scale factors for dimension settings and text height; based on either the English or metric default template.
- ***Use a Wizard: Advanced Setup*** Establishes units and angles of measurement, direction; automatically adjusts scale factors for dimension settings and text height; based on either the English or metric default template.
- ***Use a Template*** Specifies a template drawing file that contains settings such as measurement units, scale factors for text and dimensions, and drawing elements such as a title block and border.
- ***Start from Scratch*** Specifies the measurement system used (English or metric units).

To start a new drawing

Standard Toolbar	Menu	Command
	File ➤ New	NEW

Choose an option to start a new drawing using a wizard, or a template or starting from scratch.



Notice how the information displayed in the dialog box changes when you choose a new option.

The drawing file extension (*.dwg*) is automatically appended to the file name.

Using the Quick Setup Wizard

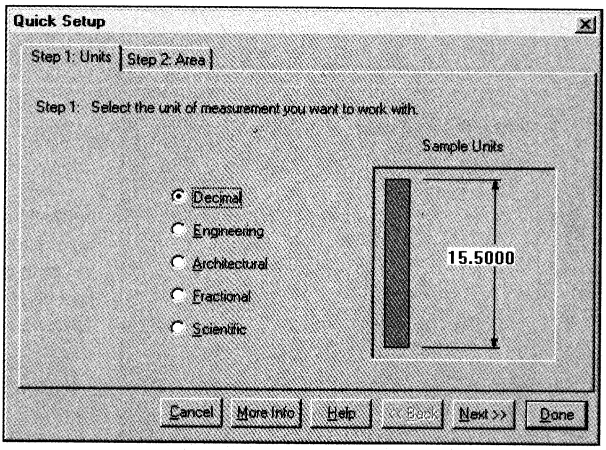
With the Quick Setup wizard, you establish basic settings that help define the units of measurement and drawing area. These settings include the basic unit type to be used for display and plotting:

- Architectural
- Decimal
- Engineering
- Fractional
- Scientific

NOTE The Quick Setup wizard uses either the *aclt.dwt* (English) or the *acltiso.dwt* (metric) template drawing file depending upon the MEASUREINIT setting in the system registry. For more information, see "Setting the System of Measurement" on page 68.

You also specify the width and length of the drawing area and thereby establish the drawing's imaginary boundaries, called limits. The area within these limits defines the final plotted sheet size. After accepting the settings, the Quick Setup wizard starts your drawing session in the model space environment.

Use a wizard to guide you through steps to set up your drawing.



The dialog box describes each step you take to set up your drawing.

Using the Advanced Setup Wizard

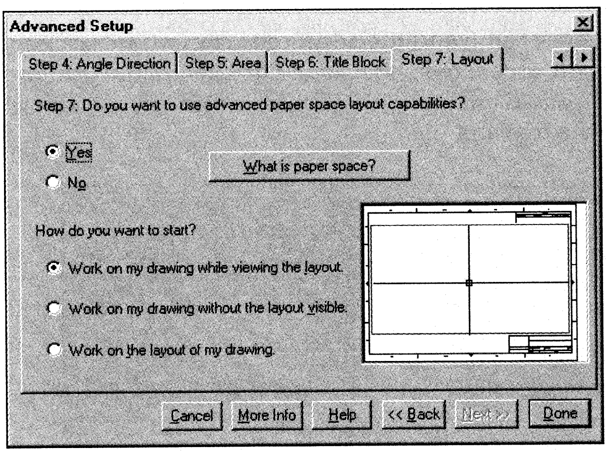
With the Advanced Setup wizard, you can define unit of measurement, angle of measurement, and direction. In addition to these setup controls in model space, you can also designate angles and angle direction and insert an existing title block and border in paper space. The Quick Setup wizard affects only model space; the Advanced Setup wizard includes options to work in both model and paper space. For more information on working in model and paper space, see "Using Paper Space and Model Space" on page 330.

NOTE The Advanced Setup wizard uses either the *aclt.dwt* (English) or the *acltiso.dwt* (metric) template drawing file depending upon the MEASUREINIT setting in the system registry. For more information, see "Setting the System of Measurement" on page 68.

After you have used the Advanced Setup Wizard to specify the drawing settings, a new drawing is opened with the appropriate settings. For example, you might specify a drawing with a title block and border displayed in paper space. A single model-space floating viewport is available for you to begin drawing.

Choose Yes to begin your drawing in paper space.

Specify the layout option you want.



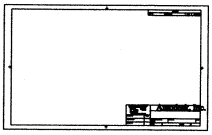
The preview changes to reflect the option you choose. The red rectangle represents the editable area of the layout you specify.

Whenever you change the default settings in the Quick Setup or the Advanced Setup wizards, AutoCAD LT automatically adjusts the system variables that control scalable elements such as linetypes, dimensions, and text.

ONLINE HELP To control setup wizard system variables, see "Using the Advanced Setup Wizard."

Using Templates

A *template drawing* file contains standard settings; AutoCAD LT supplies two default template drawing files: for English (feet and inches) and for metric measurement settings. Although you can save any drawing as a template, you might want to prepare some standard templates that include settings and basic drawing elements consistent with your office standards:



template with border and title block

- Unit type and precision
- Drawing limits
- Snap, Grid, and Ortho settings
- Layer organization
- Title blocks, borders, and logos
- Dimension and text styles
- Linetypes

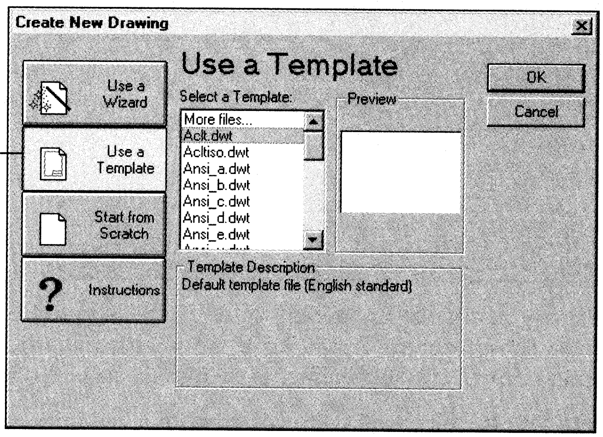
Using an Existing Template

All new drawings are based on a template. You can use any drawing as a template drawing. When you use an existing drawing as a template, all information is passed on to the new drawing.

Both the Quick Setup and the Advanced Setup wizards cause AutoCAD LT to adjust certain scale factors that are established in the default template drawing files. To avoid changing those custom settings, use an existing drawing as a template drawing file.

Choose Use a Template.

Select a template from the list or choose More files.



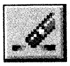
AutoCAD LT opens the drawing as *drawing.dwg*.

Creating a Template

When you need to create several drawings with similar requirements, you can save time by saving one of the drawings as a template.

To create a template

1 Open a drawing and change any drawing settings to match what you plan to use as defaults in your template. If needed, add a border and title block.



2 Erase all the objects except the border and title block by starting ERASE and entering **all** at the Select Objects prompt.

3 Enter **r** (Remove) and select the border and title block to remove them from the selection set to be erased.

4 To save your newly created template, in the Save As dialog box, choose Drawing Template file type under Save as Type.

   You can enter a description of the template and specify the measurement system in the displayed dialog box.

   The new template is saved in the *template* folder.

ONLINE HELP To work with templates, see "Starting with a Template File."

Starting from Scratch

By choosing Start from Scratch, you create a new drawing without a template. You specify the measurement system (English or metric units), then AutoCAD LT opens a new drawing named *drawing.dwg*.

Changing Setup Options

Once a drawing is open, you can change the values that you initially specified with the setup options. The following sections describe additional procedures for setting up your drawing and provide more information about some of the Quick Setup and Advanced Setup wizard features.

Setting the System of Measurement

Whether you work with an English (feet and inches) or a metric system of measurement affects the files AutoCAD LT uses for the template drawing, linetype, and hatch pattern.

File names for English and metric units		
File type	**English**	**Metric**
Template drawing	*aclt.dwt*	*acltiso.dwt*
Hatch pattern	*aclt.pat*	*acltiso.pat*
Linetype	*aclt.lin*	*acltiso.lin*

If you start a drawing using one system of measurement and then want to switch to the other, you need to scale the drawing by a conversion factor in order to obtain accurate dimensions. For example, to convert inches to millimeters, you scale the drawing by a factor of 25.4. To convert from millimeters to inches, the scale factor is 1/25.4.

When you open a new drawing, AutoCAD LT uses the MEASUREINIT system variable setting saved in the system registry to determine whether the new drawing will use English or metric units (English = 0, metric = 1). By default, MEASUREINIT uses the current setting in Regional Settings in the Windows Control Panel, unless you specify the measurement setting using the Start from Scratch option in the Create New Drawing dialog box or the Start Up dialog box.

When you create a new drawing using a setup method chosen from the Create New Drawing dialog box, AutoCAD LT determines the measurement settings:

- ***Use a Wizard*** Uses either the *aclt.dwt* (English) or the *acltiso.dwt* (metric) template drawing file depending upon the MEASUREINIT setting in the system registry. The current template setting is displayed in the Wizard Description panel of the Create New Drawing dialog box.
- ***Use a Template*** Uses the measurement setting (MEASUREMENT) saved in the specified template file to determine hatch and linetype files.
- ***Start from Scratch*** Specifies the measurement setting (MEASUREINIT) in the registry when the new drawing is created.

NOTE Using a template file or opening an existing drawing overrides the MEASUREINIT setting in the system registry and uses the measurement settings saved in the template or drawing file (MEASUREMENT system variable). Both MEASUREINIT and MEASUREMENT system variables can be set at the Command prompt using SETVAR.

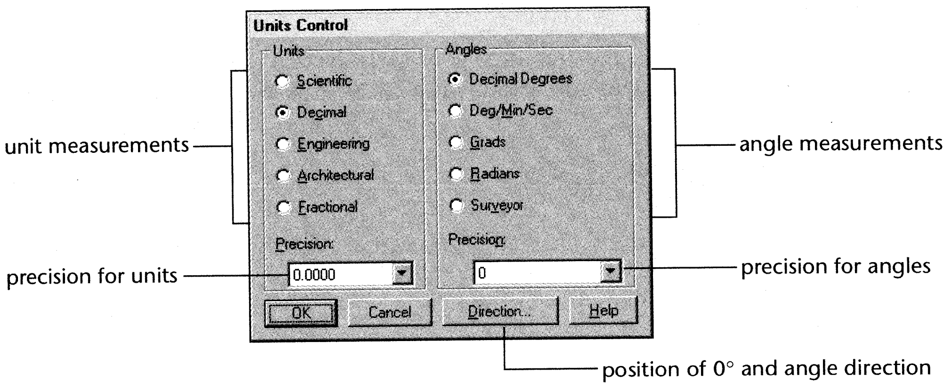
Setting Units Style

Every object you draw is measured in units. You specify the units style within AutoCAD LT before you draw. The *unit type* specifies the notation used to display coordinates and dimensioning measurements. You also set the level of *precision*, the number of decimal places or fractional settings you need. The unit type and precision settings control how AutoCAD LT interprets your coordinate and angle entries and how it displays coordinates and dimensions.

You can set the unit type in the Quick Setup or Advanced Setup wizard when you create a new drawing. You also can change the settings once the drawing is open. Use the Units Control dialog box to assign and control linear and angular measurement units.

To set the unit type

Menu	**Command**
Format ➤ Units	DDUNITS



The angle direction controls the point from which AutoCAD LT measures angles and the direction in which they're measured.

ONLINE HELP To specify angle conventions, see "Specifying Angle Conventions."

Determining the Scale Factor

Although you do not specify the scale of your drawing until you plot, you can specify the scale factor for elements such as dimensions, text height, and linetype scale in advance. Establishing a scale factor for text height, dimensions, and linetypes ensures that these elements are the appropriate size in the final drawing.

Within model space, if you use the Quick Setup or Advanced Setup wizard when you open a new drawing, AutoCAD LT automatically adjusts many of the scale factors to be appropriate for the world size, or limits, you specify.

Calculating the Scale Ratio

You can also manually calculate the scale factor by converting the drawing scale to a ratio of 1:*n*. This ratio compares plotted units to drawing units. You can then multiply your sheet size by the scale factor to calculate the limits of your drawing.

For example, if you plot at a ratio of 1/4 inch = 1 foot, you would calculate the scale ratio as follows:

1/4" = 12"
$1 = 12 \times 4$
1 = 48

Using the same calculation, the scale ratio for 1/8 inch = 1 foot is 96, and the scale ratio for 1 inch = 20 feet is 240.

If you are working in metric units, you might have a sheet size of 210×297 mm (A4 size) and a scale ratio of 20. You calculate drawing limits as follows:

210×20 = 4,200 mm = 4.2 m
297×20 = 5,900 mm = 5.9 m

Once you establish the scale factor, you can use it to set linetype scale, text and dimension sizes. The following table shows some standard architectural scale ratios and equivalent text sizes.

Using scale factors to derive text size				
Scale		**Scale factor**	**To plot text size at...**	**Set drawing text size to...**
1/8"	= 1'-0"	96	1/8"	12"
1/4"	= 1'-0"	48	1/8"	6"
1/2"	= 1'-0"	24	1/8"	3"
1"	= 1'-0"	12	1/8"	1.5"

When the drawing is complete, you can plot it at any scale, or you can plot different views of your drawing at different scales. You can position and scale paper space views without affecting the scale of the objects in the drawing. For more information, see "Scaling Views Relative to Paper Space" on page 340.

Setting the Grid

The *grid* is a pattern of dots that extends over the area specified by the limits. Using the grid is similar to placing a sheet of grid paper under a drawing. The grid helps you align objects and visualize the distances between them. You can turn the grid on and off at any time, even while in another command. The grid does not appear in the plotted drawing.

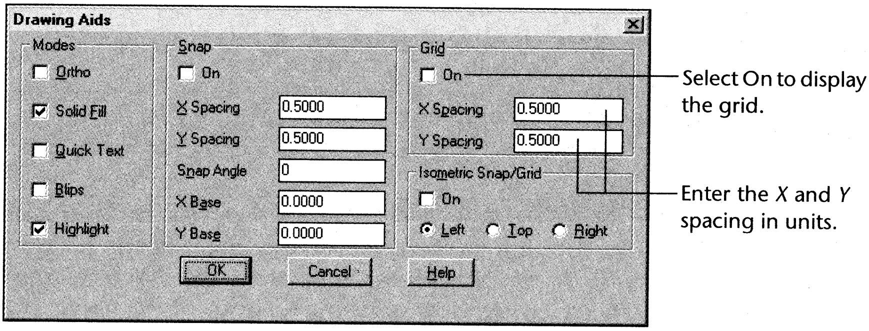
To turn the grid on or off

Status Bar	Command
GRID	GRID

Related Turn the grid on or off by pressing F7 or CTRL+G.

To set grid spacing

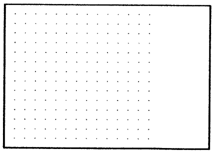
Menu	Command
Tools ➤ Drawing Aids	DDRMODES



If you zoom in or out of your drawing, you might need to adjust grid spacing to be more appropriate for the new magnification.

Although the grid does not necessarily correspond to the current snap interval, it does reflect the current snap angle and Isometric Snap/Grid settings.

Setting Grid Limits



grid limits shown by range of grid

You can set the rectangular boundary, or limits, of the drawing area covered by grid dots when the grid is turned on. The grid provides a visual representation of the limits. Setting the limits controls the extent of the display of the grid and serves as

- A reference tool that marks the area in your drawing on which you're currently working
- A drawing tool that you can use to prevent drawing outside the limits
- A plot option that defines an area to be printed

In model space, the limits should encompass the full-scale size of the model. For example, if the object is 100 × 200 mm, you should set your limits to represent a slightly larger area.

To set grid limits

Menu	Command
Format ➤ Drawing Limits	LIMITS

If you define limits in paper space, the limits boundary usually represents the final sheet size of the paper. Therefore, in paper space, the grid limits should be set to the proper size to contain the entire sheet of paper, including the drawing, dimensions, title blocks, and other information. For example, if you have a sheet of paper that is 210 × 297 mm, you could set decimal units and then specify 0,0 as the lower-left corner and 210,297 as the upper-right corner of the limits.

When you open a new drawing and use the Quick Setup or Advanced Setup wizard, you set grid limits in model space.

To set grid limits once a drawing is open

1 From the Format menu, choose Drawing Limits.
2 Specify the lower-left corner of the limits, which corresponds to the lower-left corner of your drawing area. Press ENTER to use the default (0,0).
3 Specify the upper-right corner of the limits, which corresponds to the upper-right corner of your drawing area.

For example, if the lower-left corner is at 0,0, you can specify a limits area that is 30 units wide and 10 units high by entering **30,10**.

To view the graphics area defined by the grid limits you specified, turn on GRID, then from the View menu, choose Zoom ➤ All.

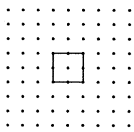
Setting Grid and Snap Spacing

Snap restricts the movement of the crosshairs to intervals that you have defined. When Snap is on, the cursor seems to adhere, or "snap," to an invisible grid. Snap is useful for specifying precise points with the pointing device. You control snap precision by setting the *X* and *Y* spacing. Snap has a toggle control and can be turned on or off during another command.

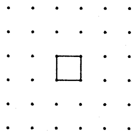
Snap spacing does not have to match grid spacing. For example, you might set a wide grid spacing to be used as a reference but maintain a closer snap spacing for accuracy in specifying points. You can also set the grid spacing to be smaller than the snap spacing.

To set snap spacing

Menu	Command
Tools ➤ Drawing Aids	DDRMODES



snap is twice the grid



snap matches the grid

Related To turn Snap on and off, double-click the Snap button on the status bar, enter the SNAP command, press CTRL+B, or press F9.

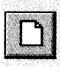
ONLINE HELP To set the grid, grid limits, and snap spacing, see "Adjusting Snap and Grid."

Adding Borders and Title Blocks

The border and title block can be customized to meet your company requirements and then used in several ways. As you open a new drawing, you can use the Advanced Setup wizard to insert a title block and border on their own layer in paper space or model space.

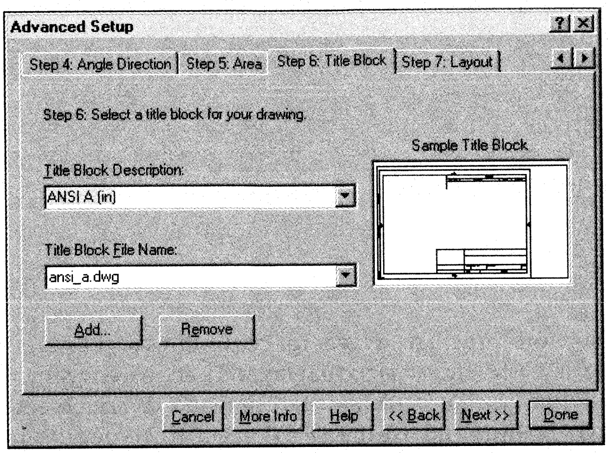
The title block and border reside in a separate drawing that is inserted as a block—a single object created by grouping one or more objects. AutoCAD LT provides several standard title blocks to choose from. If you prefer, you can open a new drawing using a template that already contains a title block and border.

You can open and customize standard title blocks, or you can modify them after you insert them in the drawing. You can also create and use your own title blocks and add them to the list that is displayed when you use the Advanced Setup wizard.

Standard Toolbar	Menu	Command
	File ➤ New	NEW

Choose a title block from the description list.

Click Add to choose an unlisted title block.



Your specified title block is displayed in the preview box.

The file names generally correspond to paper sizes.

ONLINE HELP To add a title block to a drawing, see "Starting with a Wizard."

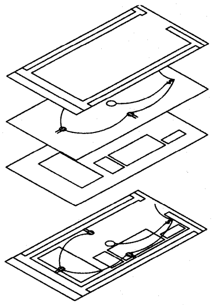
Organizing Information on Layers

Layers are the equivalent of the overlays used in paper-based drafting. They are the primary organizational tool in AutoCAD LT. Layers are used to group information by function and to enforce linetype, color, and other standards. The flexibility of the layers feature is a major advantage of creating drawings with AutoCAD LT instead of with paper and pencil.

AutoCAD LT also provides options to control the display of layers to reduce the visual complexity of a drawing. Use the option to lock a layer to prevent objects from being accidently selected and modified.

Planning Layers

You are always drawing on a layer. It may be the default layer (0) or a layer you create and name yourself. Each layer has an associated color and linetype. You can group drawing components by assigning similar objects to the same layer.



For example, you can create a layer for centerlines and assign the color blue and the CENTERLINE linetype to that layer. Then, whenever you draw centerlines, you switch to that layer and start drawing. Every object you draw on that layer will be blue and use the CENTERLINE linetype automatically. Later, if you don't want to display or plot centerlines, you can turn off that layer.

With layers, you have great flexibility in the amount of detail you can display. You can display any combination of layers. You can hide construction lines or details. In paper space, you can make some layers visible only in certain viewports.

If you use a black-and-white plotter, the colors you assigned to the layers in your drawing can be used to control line width. When you plot, you assign each color to a certain pen. Your plotter then uses the pen (or pen width) associated with the layer color.

Creating and Naming Layers

AutoCAD LT sorts layer names alphanumerically by default. If you're organizing a layer scheme, choose layer names carefully. By using common prefixes to name layers with related drawing components, you can use wild-card characters or filters when you need to find those layers quickly. For example, if you create a set of layers that start with PIPING, such as PIPING-FLOOR 1, PIPING-FLOOR 2, and so on, you can later sort for PIPING to display only those layers.

Controlling the Display of Layers

To make objects on a layer invisible, turn the layer off or freeze it. Turning off or freezing layers is useful if you need an unobstructed view when working in detail on a particular layer or set of layers or if you don't want to plot details such as reference lines. Whether you choose to freeze layers or turn them off depends on how you work and on the size of your drawing.

Turning Off Layers

Turn a layer off rather than freezing the layer if you frequently need to switch a layer's visibility. When you turn a layer back on, the objects on the layer will be redisplayed more quickly. Layers that are turned off are generated during a regeneration of the drawing but are not displayed.

Freezing Layers

Freeze a layer if you do not need that layer to be visible for a long period of time. This reduces the time required by subsequent drawing regeneration and object selection in a large drawing. You redisplay frozen layers by thawing them. Because thawing one or more layers causes an automatic regeneration of the drawing, thawing a layer is slower than turning a layer on. Layers that are frozen are not regenerated during a regeneration of the drawing.

Locking Layers

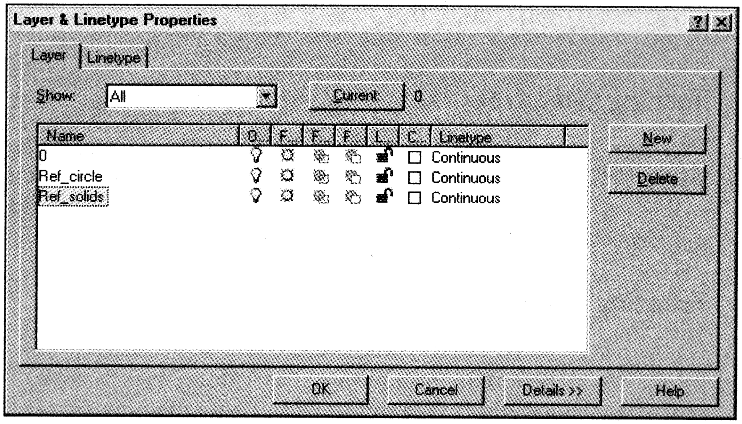
Prevent objects on a layer from being modified by locking it. When a layer is locked, none of the objects on that layer can be modified until you unlock the layer. This reduces the possibility of modifying objects accidentally. Objects on a locked layer are visible. You can still apply object snaps to objects on a locked layer and perform other operations that do not modify those objects.

To create and modify layers

Object Properties Toolbar	Menu	Command
	Format ➤ Layer	LAYER

To rename a layer, click the highlighted name and enter a new name.

To change a layer's color, click its Color icon.



Choose New to create a layer.

To change a layer's linetype, click its Linetype name.

To select more than one layer, hold down CTRL.

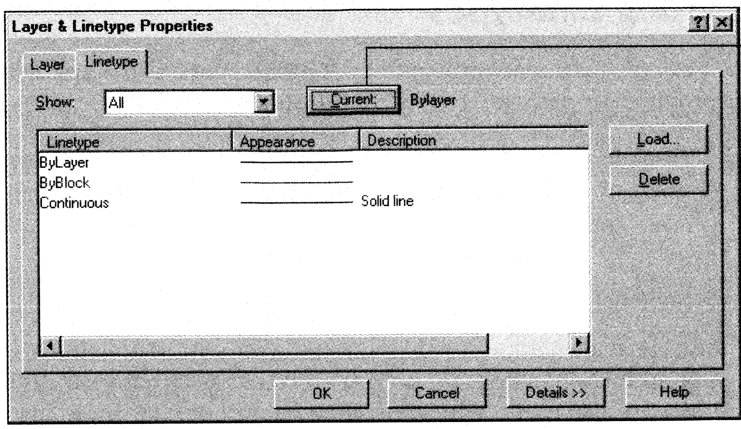
To change a layer's property, such as lock or freeze, click the appropriate icon.

ONLINE HELP To create and modify layers, see "Working with Layers."

Loading Linetypes

At the start of a project, you load additional linetypes that are required for the project so that they are available when you need them. The linetypes BYLAYER, BYBLOCK, and CONTINUOUS are always available and cannot be deleted. Each linetype has a name and a definition. The definition describes the sequence of dashes, dots, and spaces; the relative lengths of dashes and blank spaces; and the characteristics of any included text or shapes. You can associate linetypes with all AutoCAD LT objects except text, points, viewports, and blocks. If a line is too short to hold even one dash sequence, AutoCAD LT draws a continuous line between the endpoints.

Loaded line-types are listed.

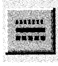


Click Current to set the current linetype.

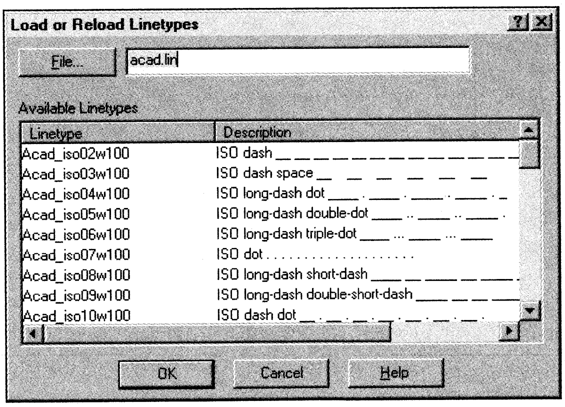
To use a linetype, you must first load it into your drawing. A linetype definition must exist in a *.lin* linetype library file before it can be loaded into a drawing. AutoCAD LT includes two linetype definition files:

- *aclt.lin* for drawings using English (feet and inches) units
- *acltiso.lin* for drawings using metric units

To load a linetype

Object Properties Toolbar	Menu	Command
	Format ➤ Linetype	LINETYPE

Use the File option to load an alternate LIN file for additional linetypes.



Select a linetype from the list of available linetypes.

NOTE The ISO pen width option is available only if you've selected and made current a linetype whose name begins with ACAD_ISO.

ONLINE HELP To load linetypes, see "Loading Linetypes."

Using Linetypes

A linetype is a repeating pattern of dashes, dots, and blank spaces. Linetypes provide another way to convey information visually.

CONTINUOUS

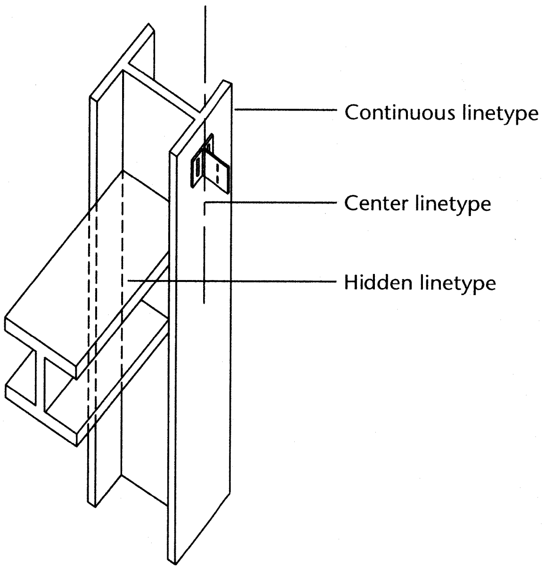
HIDDEN

CENTER

PHANTOM

Examples of linetypes

You can use linetypes to distinguish the purpose of one line from another, for example, dashed lines for hidden objects, dot-dash lines for centerlines. Linetypes are also useful when you don't have or don't want to use color. You can use any of the standard linetypes that AutoCAD LT provides, or you can create your own linetypes.



Like colors, linetypes can be assigned to individual objects as well as layers in a drawing. AutoCAD LT assigns the current linetype automatically to new objects you create. If the current linetype is set to BYLAYER (default), objects are created with the linetype assigned to the current layer.

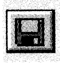
ONLINE HELP To create your own linetypes, see "Customizing Linetypes."

Saving Drawings

When you are working on a drawing, you should save it frequently. On the System tab in the Preferences dialog box, you can set the time interval at which AutoCAD LT automatically saves your work.

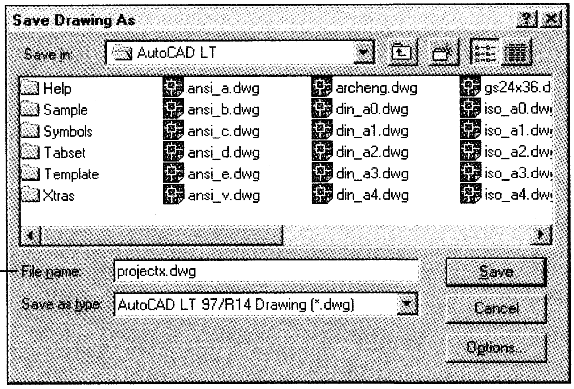
If you want to create a new version of a drawing without affecting the original drawing, you can save it under another name.

To save a drawing

Standard Toolbar	Menu	Command
	File ➤ Save	SAVE

Related SAVEAS saves a copy of the current drawing under the name you specify.

ONLINE HELP For more information about saving files to other formats, see "Managing Files and Formats."



Enter the desired drawing name; the file extension is not required.

Using Long File Names

With AutoCAD LT, you can use long file names in both Windows 95 and NT. These names can contain up to 256 characters and can contain embedded spaces and punctuation; for example, you can use file names such as *summer home.dwg* and *test file.dwg*.

Long file names for blocks and externally referenced drawings need special treatment. See "Inserting Blocks with Long File Names" on page 299.

Exiting AutoCAD LT

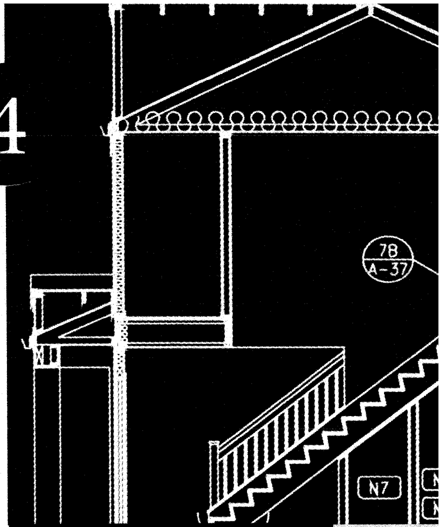
If you have saved your most recent changes, you can exit AutoCAD LT without saving the drawing again. If you have not saved your changes, AutoCAD LT prompts you to save or discard the changes or cancel the command.

To exit a drawing

Menu	Command
File ➤ Exit	EXIT or QUIT

4

Using Coordinate Systems



In this chapter

- Using World and user coordinate systems
- Working with Cartesian and polar coordinates
- Specifying coordinates
- Moving and rotating a user coordinate system (UCS)

In online Help

- See "Working in Three Dimensions" and "Creating Realistic Images."

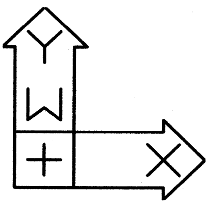
As you draw, you'll discover that you use some AutoCAD LT features repeatedly. One such feature is the coordinate system, which you use to specify points in the drawing. You can set up your own movable user coordinate system (UCS) for working on angled, isometric, or three-dimensional (3D) views.

Using World and User Coordinate Systems

AutoCAD LT, has two coordinate systems: a fixed system called the *World Coordinate System* (WCS), and a movable system called the *user coordinate system* (UCS). In the WCS, a global reference system used to define other user coordinate systems, the *X* axis is horizontal, the *Y* axis is vertical, and the *Z* axis is perpendicular to the *XY* plane. The origin is where the *X* and *Y* axes intersect (0,0) in the lower-left corner of the drawing. When you move the UCS, you define its new location in terms of the WCS. Virtually all coordinate entry is performed using the current UCS.

The user coordinate system provides an alternate movable coordinate system for coordinate entry, planes of operation, and viewing. Most AutoCAD LT editing commands are dependent on the location and orientation of the UCS.

The UCS command sets the orientation of the user coordinate system in three-dimensional space.



UCS icon

By default, an icon that indicates the current coordinate system is displayed in the lower-left corner of the graphics area. It represents the orientation of the UCS axes and the location of the current UCS origin. The letter "W" is displayed in the *Y* portion of the icon if the current UCS is the same as the World Coordinate System. The *X* and *Y* denote the positive directions of the *X* and *Y* axes.

To display or hide the UCS icon

Menu	**Command**
View ➤ Display ➤ UCS Icon ➤ On or Off	UCSICON

ONLINE HELP See "Controlling the User Coordinate System (UCS)."

Using a Coordinate System to Specify Points

When a command prompts you for a point, you can use the pointing device to specify a point in the graphics area or you can enter coordinate values on the command line. This section describes how to enter coordinate values.

Using Cartesian and Polar Coordinate Systems

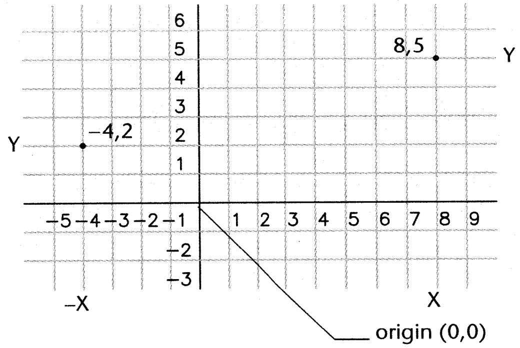
A Cartesian coordinate system has three axes, *X*, *Y*, and *Z*. When you enter coordinate values, you indicate a point's distance (in units) and its direction (+ or –) along the *X*, *Y*, and *Z* axes relative to the coordinate system origin (0,0,0). When you begin a new drawing in AutoCAD LT, you use the World Coordinate System (WCS) automatically: the *X* axis is horizontal, the *Y* axis is vertical, and the *Z* axis is perpendicular to the *XY* plane.

Polar coordinate systems use a distance and an angle to locate a point.

Locating Points

The following illustration demonstrates the location of points on the *XY* plane.

The –4,2 coordinate represents a point 4 units in the negative *X* direction and 2 units in the positive *Y* direction.



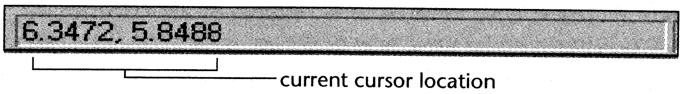
The 8,5 coordinate indicates a point 8 units in the positive *X* direction and 5 units in the positive *Y* direction.

Two-dimensional coordinate system

In AutoCAD LT, you can enter coordinates in scientific, decimal, engineering, architectural, or fractional notation. You can enter angles in decimal degrees, grads, radians, and surveyor's units, or in degrees, minutes, and seconds. This guide uses decimal units and degrees.

Displaying Coordinates

AutoCAD LT displays the current cursor location as a coordinate in the status bar at the bottom of the AutoCAD LT screen.



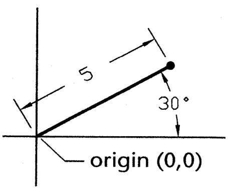
There are three types of coordinate display available:

- Dynamic display updates as you move the cursor.
- Static display updates only when you specify a point.
- Distance and angle *(distance<angle)* display updates as you move the cursor. This option is available only when you draw lines or other objects that prompt for more than one point.

You can cycle through the three types of coordinate display by pressing F6 or CTRL + D.

Specifying Cartesian and Polar Coordinates

In two-dimensional space, you specify points on the *XY* plane, also called the *construction plane*. The construction plane is similar to a flat sheet of grid paper. The *X* value of a Cartesian coordinate specifies horizontal distance, and the *Y* value specifies vertical distance. The origin point (0,0) indicates where the two axes intersect.



polar coordinates

You can enter 2D coordinates as either Cartesian (*X,Y*) or polar coordinates. Polar coordinates use a distance and an angle to locate a point. You can use absolute or relative values with each method. Absolute coordinate values are based on the origin (0,0). Relative coordinate values are based on the last point entered. They are useful for locating a series of points that are a known distance apart.

Specifying 3D coordinates is the same as specifying 2D coordinates with the addition of a third dimension, the *Z* axis. When drawing in 3D, you specify *X*, *Y*, and *Z* values of the coordinate in either the World Coordinate System (WCS) or the user coordinate system (UCS).

ONLINE HELP See "Entering 3D Coordinates."

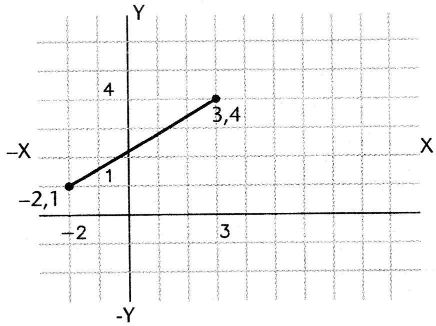
Entering Absolute *X,Y* Coordinates

To enter an absolute *X,Y* coordinate, specify a point by entering its *X* and *Y* values in the format *X,Y*. Use absolute *X,Y* coordinates when you know the precise *X* and *Y* values of the location of the point.

For example, to draw a line beginning at an *X* value of –2 and a *Y* value of 1 and ending at (3,4), make the following entries on the command line:

Command: *Enter* **line**
From point: *Enter* **–2,1**
To point: *Enter* **3,4**

AutoCAD LT locates the line as follows:



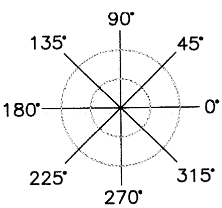
Entering Relative Coordinates

Use relative *X,Y* coordinates when you know the position of a point in relation to the previous point. For example, to locate a point relative to –2,1, precede the next coordinate with the @ symbol:

Command: *Enter* **line**
From point: *Enter* **–2,1**
To point: *Enter* **@5,3**

This draws the same line shown in the preceding illustration.

Entering Polar Coordinates



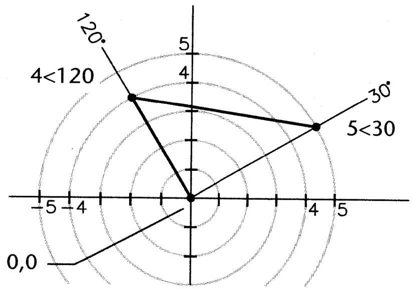
To enter a polar coordinate, enter a distance and an angle, separated by an angle bracket (<). For example, to specify a point that is a distance of 1 unit from the previous point and at an angle of 45 degrees, enter **@1<45**.

By default, angles increase in the counterclockwise and decrease in the clockwise direction. To move clockwise, enter a negative value for the angle. For example, entering **1<315** is the same as entering **1<- 45**. You can change the angle direction for the current drawing with the DDUNITS command or the ANGDIR system variable. Also, ANGBASE sets the direction of the Ø angle.

You also can use a feature called PolarSnap. With PolarSnap, you can specify a relative distance and angle. For more information, see "Using PolarSnap" on page 121.

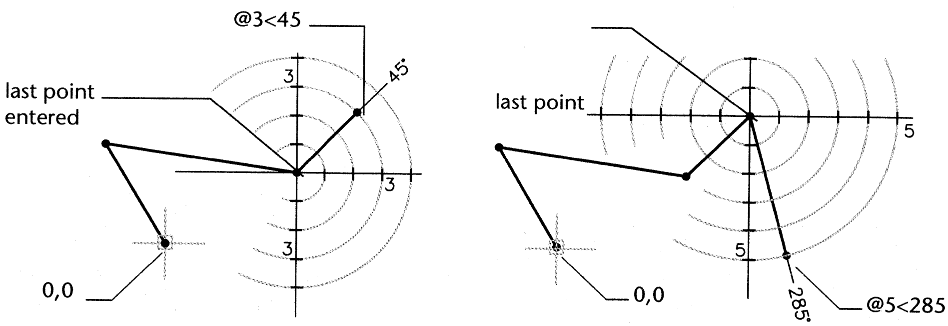
The following example shows line segments drawn with polar coordinates.

Command: *Enter* **line**
From point: *Enter* **0,0**
To point: *Enter* **4<120**
To point: *Enter* **5<30**



To point: *Enter* **@3<45**
To point: *Enter* **@5<285**
To point: *Press* ENTER

Notice how the last two points entered specify *relative* polar coordinates (indicated by the @ symbol), so that the angle and distance specified is measured from the last point entered, as shown in the following example.



You also can use a feature called *direct distance entry*. With direct distance entry, you can specify a relative coordinate by moving the cursor to specify a direction and then entering a distance. For more information, see "Using Direct Distance Entry" on page 123.

Specifying Units and Angles

You can specify the unit type according to your drawing's requirements: architectural, decimal, scientific, engineering, or fractional. Depending on what you specify, you can enter coordinates in decimal form or in feet, inches, and degrees or in other notation. To enter architectural feet and inches, indicate feet using the prime symbol ('): for example, **72'3,34'4**. You don't need to enter the double prime symbol or quotation marks (") to specify inches.

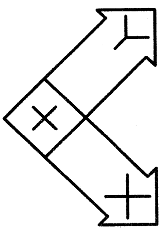
If you use surveyor's angles when specifying polar coordinates, indicate whether the angles are in the north, south, east, or west direction. For example, to enter a coordinate relative to the current coordinate for a property line that is 72 feet, 8 inches, long with a bearing of 45 degrees north, 20 minutes, 6 seconds, east, enter

@72'8"<n45d20'6"e

You can enter three-dimensional coordinates in the same input formats as two-dimensional coordinates: scientific, decimal, engineering, architectural, or fractional notation. Also, you can enter angles using decimal degrees, grads, radians, or surveyor's units or using degrees, minutes, and seconds.

ONLINE HELP To control the display of coordinates, see "Displaying Coordinates on the Status Bar." To specify the angle direction for polar coordinates, see "Entering Polar Coordinates."

Shifting and Rotating the Coordinate System



rotated UCS

Moving the UCS can make it easier to work on particular sections of your drawing. Rotating the UCS helps you specify points in 3D or rotated views. Snap, Grid, and Ortho all rotate in line with the new UCS.

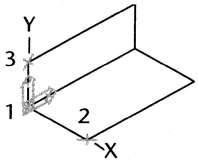
You can also set up a new angled baseline and draw lines relative to it. Snap, Grid, and Ortho rotate in line with the new UCS orientation.

With a customized UCS, you can rotate the *X,Y* plane and change the origin point of the coordinate system. This feature is particularly useful for working on sections where the baseline deviates from a horizontal or vertical orientation.

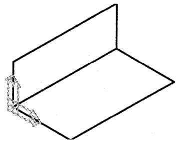
You can relocate the user coordinate system using several methods. Two methods, specifying a new *XY* plane and specifying a new origin, are described in this chapter. The other methods are more appropriate for working with 3D viewpoints.

Shifting the *XY* Plane

One way to relocate a UCS is to specify a new UCS origin and the direction of its positive *X* and *Y* axes.



points specified for new UCS



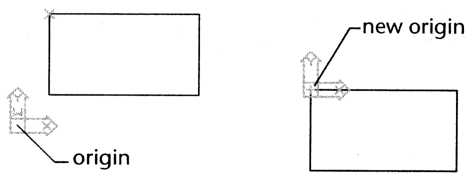
new UCS

To relocate the UCS

UCS Toolbar	Menu	Command
	Tools ➤ UCS ➤ 3 Point	UCS

Locating a New UCS Origin

By locating a new origin, you can adjust coordinate entry to be relevant to a specific area or object in your drawing. For example, you might relocate the origin point to the corner of a building or to serve as a reference point on a map.



Example showing location of new UCS origin

To locate a new UCS origin

UCS Toolbar	Menu	Command
	Tools ➤ UCS ➤ Origin	UCS

ONLINE HELP To control the UCS when working in 3D, see "Setting Drawing Planes in 3D (UCS)."

5

Creating Objects

With the AutoCAD LT drawing tools, you can create a range of objects, from simple lines and circles to spline curves, ellipses, and associative hatch areas. In general, you draw objects by specifying points with the pointing device or by entering coordinate values on the command line.

In this chapter

- Drawing line objects, polylines, double lines, and polygons
- Drawing curved objects such as circles, arcs, ellipses, donuts, and spline curves
- Creating solid-filled areas and hatched areas

In online Help

- For information about drawing 3D objects, see "Working in Three Dimensions."

Drawing Objects

AutoCAD LT drawings are made up of objects—lines, circles, splines, polygons, and other geometric shapes that you use to create the final composition. Each object you create is an individual entity: that is, you can move, copy, resize, measure, and otherwise manipulate it separately from all the other objects in your drawing.

In general, you draw objects by specifying points in the graphics area using your pointing device, or by entering coordinate values on the command line. To gain accuracy without entering coordinates, you can use Snap, Grid, Ortho, and PolarSnap drawing aids. You can also use object snaps to snap to specific points on an existing object. For more information about using drawing aids, see chapter 6, "Drawing with Precision." For more information on Object Snaps, see "Snapping to Geometric Points on Objects" on page 124.

Certain properties are associated with all objects that you draw in AutoCAD LT:

- ***Linetype*** The type of line assigned to an object can be set to many different styles, such as solid, center, dotted, and hidden.
- ***Color*** The color you assign to objects helps you visually distinguish them. When you print or plot your drawing, you can map a color to a specific pen or line weight on the plotter.
- ***Layer*** An object is associated with the layer on which you create it. You can create and name layers and assign a color and linetype to each one as dictated by industry, job, or company standards.

All objects in AutoCAD LT are drawn on layers. Objects can be made visible or invisible by turning their layers on and off. You can either associate the color and linetype properties of objects with layers, or you can set color and linetype explicitly for each object. You are not limited to the linetypes that are included with AutoCAD LT; you can easily create and save new linetypes.

ONLINE HELP See "Working with Layers," "Changing the Color of an Object," and "Changing the Linetype of an Object."

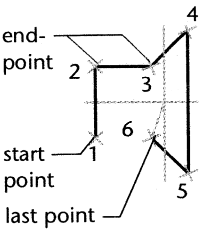
The properties assigned to objects are easily verified, and you can copy properties from one object to other objects. See "Matching Properties of Other Objects" on page 158.

You can control how some objects are displayed. Certain kinds of objects, including text, hatches, and wide polylines, can be displayed in a simplified form for faster display and preliminary test plots. See "Turning Visual Elements On and Off" on page 215. When objects overlap, you can control the order in which they are displayed. See "Changing the Drawing Order of Objects" on page 116.

Drawing Line Objects

The line is the most basic object in AutoCAD LT. You can create a variety of lines—single segment lines, multiple-segment lines with and without arcs, and double lines. In general, you draw lines by specifying points. See "Using a Coordinate System to Specify Points" on page 84.

Drawing Lines



A line can be one segment or a series of connected segments, but each segment is a separate line object. Use lines if you want to edit individual segments. If you need to draw a series of line segments as a single object, use a polyline. You can close a sequence of lines so that the first and last segments are joined.

To draw a line segment, you specify a start point (1) and an endpoint (2). To continue drawing line segments, you continue to specify points in your drawing (3,4,5,6). To finish drawing line segments, press ENTER, or enter **c** (Close) to connect the start point of the first segment (1) with the endpoint (6) of the last segment.

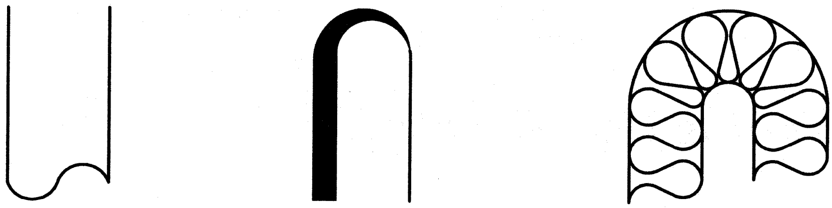
To draw a line

Draw Toolbar	Menu	Command
	Draw ➤ Line	LINE

ONLINE HELP To draw lines, see "Creating Single-Segment Lines."

Drawing Polylines

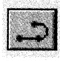
A polyline is a connected sequence of line or arc segments created as a single object. Use polylines if you want to edit all segments at once, although you can also edit them singly. You can set the width of individual segments, make segments taper, and close the polyline. When you draw arc segments, the first point of the arc is the endpoint of the previous segment. You can specify the angle, center point, direction, or radius of the arc. You can also complete the arc by specifying a second point and an endpoint.



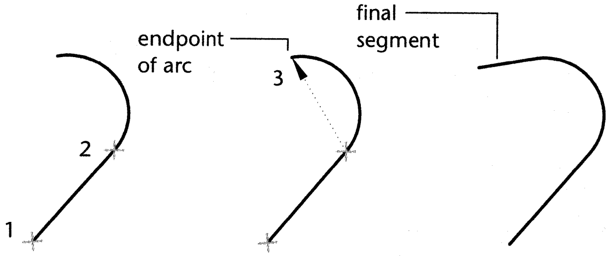
pipe symbol differing widths an insulated wall

Polylines with arc segments

To draw a polyline

Draw Toolbar	Menu	Command
	Draw ➤ Polyline	PLINE

The following example shows how you draw a polyline line segment (1,2). Switch to Arc mode by entering **a**, and continue with an arc segment (3). Enter **L** to return to Line mode, then draw another line segment in a tangential direction.



After you've created a polyline, you can edit it with PEDIT or use EXPLODE to convert it to individual line and arc segments. When you explode a wide polyline, the line width reverts to 0 and the resulting polyline is positioned along the center of what was the wide polyline.

Creating Wide Polylines

You can draw polylines of various widths by using the Width and Halfwidth options. These options become available after you start the PLINE command and specify a starting point.

Polylines with various widths

- ***Width*** Sets the width of the next polyline segments you draw. Zero (0) width produces a thin line. Widths greater than zero produce wide lines. Wide lines are filled if the FILLMODE system variable is on and outlined if FILLMODE is off.

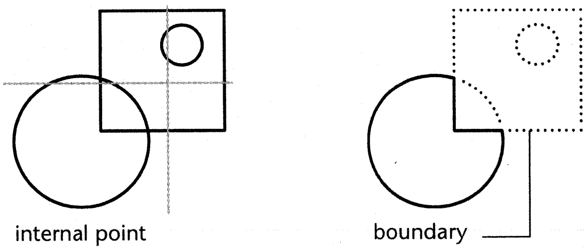
  When you use this option, AutoCAD LT prompts for both a starting and an ending width. By entering different values, you can taper the polyline. The starting and ending points of wide polyline segments are in the center of the line. Intersections of adjacent wide segments are usually beveled. However, AutoCAD LT does not bevel nontangent arc segments, acute angles, or segments that use a dash-dot linetype.
- ***Halfwidth*** Sets width by specifying the distance from the center of the wide polyline to an outside edge.

ONLINE HELP See "Creating Wide Polylines."

Creating Polylines from the Boundaries of Objects

You can create a polyline from the boundaries of overlapping objects that form a closed area.

A polyline created using the boundary method is a separate object, distinct from the objects used to create it. You can edit it using the same methods used to edit other polylines.



To expedite the boundary selection process, you can specify a group of boundary candidates, called a boundary set (see "Defining Hatch Boundaries" on page 111). Use boundary sets when you work in large or complex drawings. You create this set by selecting the objects you want AutoCAD LT to examine as it defines the boundary.

Because a boundary polyline overlays the objects used to create it, it may not be visible. However, you can move, copy, or modify it just as you can any other polyline.

To draw a boundary polyline

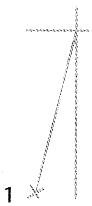
Menu	Command
Draw ➤ Boundary	BOUNDARY

ONLINE HELP See "Creating Multisegmented Lines (Polylines)."

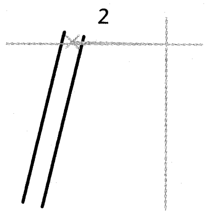
Drawing Double Lines

To create offset lines simultaneously, draw double lines. They provide a good way to draw walls in floor plans, for example. You can draw double lines as straight segments or as *arcs*. Each double-line segment and endcap, a line that connects the double-line endpoints, is actually a separate line object that can be individually edited.

The following example shows how you draw a double-line segment, specifying the start point (1), and the end point (2).

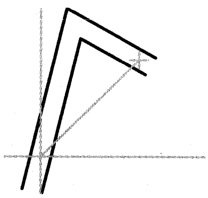


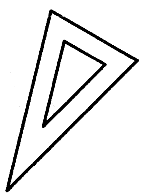
start point



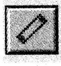
endpoint

As with polylines, you can continue to add new segments until you exit the command. To close a double line object so that the last segment is connected to the starting point, you enter **cl** (Close). The following example shows a new segment, with the last segment closing the double line.


new segment


polyline closed

To draw a double line

Draw Toolbar	Menu	Command
	Draw ➤ Double Line	DLINE

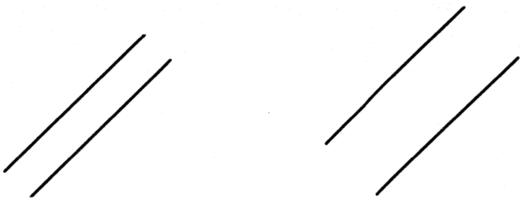
Related OFFSET creates concentric circles, parallel lines, and parallel curves.

ONLINE HELP See "Creating Double Lines."

The following sections describe the options available to control the placement and appearance of double lines. Access these options on the command line.

Setting Double-Line Width

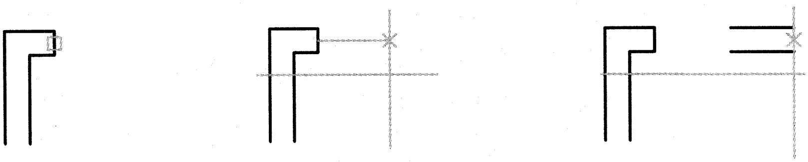
You can specify the width before or after you specify the first point. The width determines the amount of offset between the lines.



width = 1　　　　width = 2

Offsetting the Start of the Double Line

To start a double line a specific distance and angle from another point, use the Offset option.

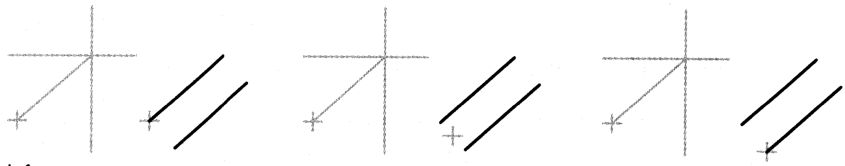


reference point　　　　direction and distance specified　　　　result

Controlling Placement of Double Lines

The *dragline* is an imaginary line that determines the placement of a double line in relation to the points you specify. You can control whether the double line is centered on or offset from the dragline.

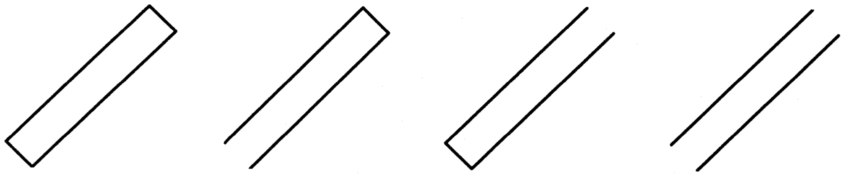
A negative offset value positions the offset side of the double line to the left of the specified points, while a positive number positions the offset side to the right. You can also specify that the dragline coincides with either the left or the right lines. Determine the left and right by imagining that you are standing at the start point of the line and looking at the endpoint.



left center right

Setting the Appearance of Endpoints

The ends of double lines can be open, closed, or different on each end. When the Endcaps option is turned on, AutoCAD LT connects, or caps, the specified ends. An exception to this rule is the Auto option, which automatically caps only the ends that are not snapped to an object.



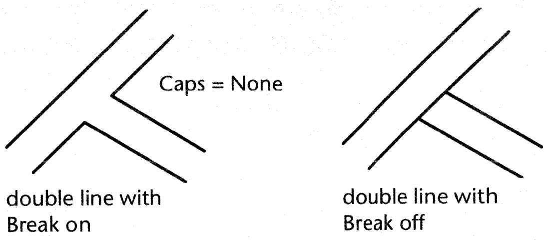
both endpoint start point none

Snapping Double Lines to Objects

You can start or end a double line by snapping to an existing object. To speed up the snap process, you can specify the pixel size of the area where AutoCAD LT searches for object snap candidates. The double line starts or ends by snapping to the object it finds in this search area. Each of the double-line legs adjusts to form a clean junction with the object. For more information on snapping to objects, see "Snapping to Geometric Points on Objects" on page 124.

Controlling the Appearance of Intersections

You can determine whether AutoCAD LT creates a gap in a line when you snap a double line to it. If the Caps option is on, the gap may not be visible.

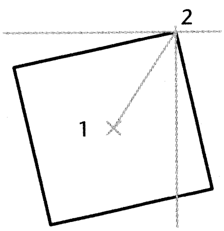


Drawing Polygons

Polygons are closed polylines with between 3 and 1,024 equal-length sides. You draw a polygon by specifying the center point and radius length or by specifying the endpoints of one of the edges of the polygon. Because polygons always have equal-length sides, they provide a simple way to draw squares and equilateral triangles.

The following illustrations show polygons drawn using the three methods. In the first two illustrations, (1) is the center of the polygon and (2) defines the radius length, which is being specified with the pointing device.

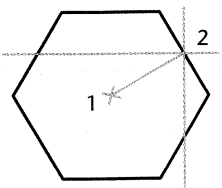
Drawing Inscribed Polygons



Use inscribed polygons when you want to specify the distance between the center of the polygon and each vertex. This distance is the radius of the circle within which the polygon is inscribed.

As illustrated in the example at the left, to draw an inscribed square, you specify the number of sides, then specify the center of the polygon (1). Enter **i** (Inscribed in Circle), and then specify the radius length (2).

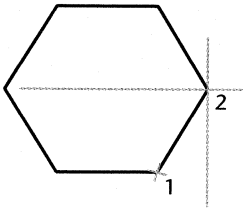
Drawing Circumscribed Polygons



Use circumscribed polygons when you want to specify the distance between the center of the polygon and the midpoint of each side. This distance is the radius of the circle the polygon circumscribes.

As illustrated in the example at the left, to draw a circumscribed hexagon, you specify the number of sides, and then specify the center of the polygon (1). Enter **c** (Circumscribed about Circle), and then specify the radius length (2).

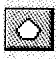
Specifying Polygon Edge Length



Use the Edge option when you want to draw a polygon by specifying the end-points of one of the edges of the polygon.

Another method is to draw your polygon based on a circle that you create first. By specifying the center of the polygon as the center of the circle, you can create a polygon inside or outside the circle. The circle can later help you find the center point of the polygon, if needed. When the drawing is complete, you can erase the circle or place it on a turned-off or frozen layer.

To draw a polygon

Draw Toolbar	Menu	Command
	Draw ➤ Polygon	POLYGON

Related RECTANG creates polyline rectangles.

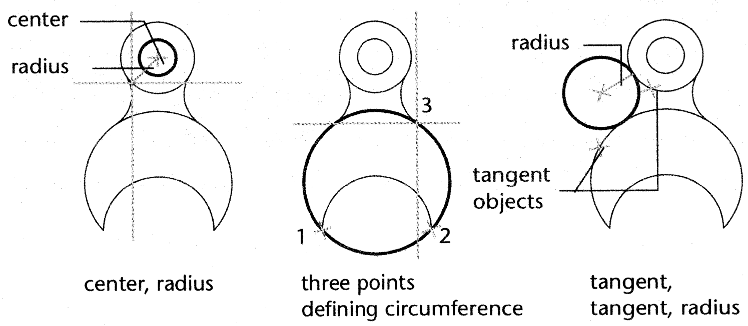
ONLINE HELP See "Creating Polygonal Objects."

Drawing Curved Objects

You can create a variety of curved objects with AutoCAD LT, including circles, arcs, ellipses, donuts, and spline curves.

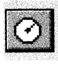
Drawing Circles

You can create circles in several ways. The default method is to specify the center and radius. You can also specify the center and diameter or define the circle's circumference with three points. You can also create the circle tangent to three existing objects or create it tangent to two objects and specify a radius. In the following illustrations, the darker circles are the ones being drawn.



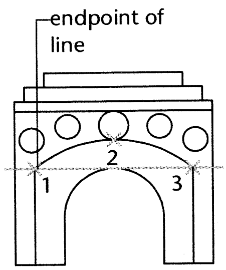
Three methods of drawing circles

To draw a circle

Draw Toolbar	Menu	Command
	Draw ➤ Circle	CIRCLE

ONLINE HELP See "Creating Circles."

Drawing Arcs



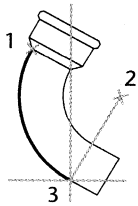
arc drawn using default 3-point method

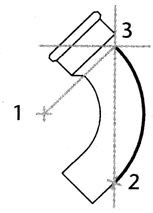
You can create arcs in many ways. The default method is to specify three points—a starting point, a second point on the arc, and an endpoint. You can also specify the included angle, radius, direction, and chord length of arcs. The chord of an arc is a straight line between the endpoints. By default, AutoCAD LT draws arcs counterclockwise.

This example illustrates how you draw an arc by specifying three points, making use of object snaps for precision drawing. The start point of the arc (1) snaps to the endpoint of a line. The second point of the arc (2) snaps to the quadrant of the middle circle. You specify the endpoint of the arc (3). For more information, see "Snapping to Geometric Points on Objects" on page 124.

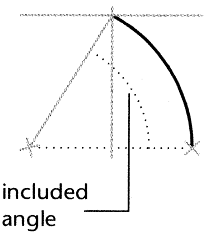
Specifying Three Points

Use the Start, Center, End or Center, Start, End method when you know the start point, center point, and endpoint. You draw an arc by specifying either the start point or the center point first. The center point is the center of the circle of which the arc is a segment.



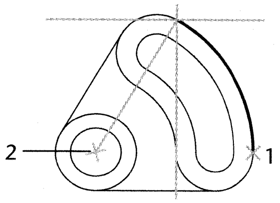


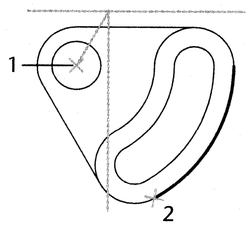
Arcs drawn by specifying either the start point or the center point first

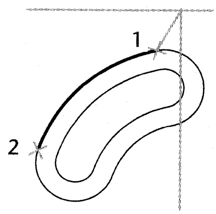


Specifying Two Points and an Angle

Use the Start, Center, Angle or Center, Start, Angle method when you have a start point and a center point you can snap to. The included angle determines the endpoint of the arc. Use the Start, End, Angle method when you have both endpoints but no center point to snap to.



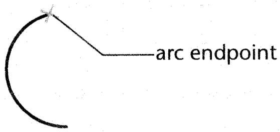


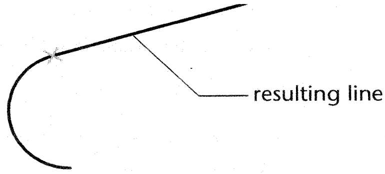


Arcs drawn using angle methods

Connecting Arcs and Lines

Immediately after you complete an arc, you can start a line at the arc endpoint by starting the LINE command and pressing ENTER at the Start Point prompt. The arc's endpoint defines the start point and the direction of the new line. You need to specify only the line length.

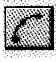




You can connect sequentially drawn arcs in the same way. To create connected arcs using a menu, choose Arc from the Draw menu. Then choose Continue.

In both cases, the resulting object is tangent to the previous one.

To draw connected arcs

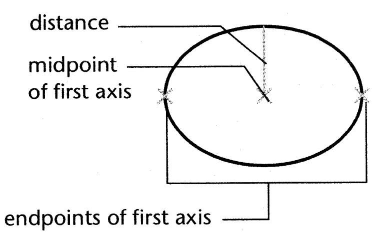
Draw Toolbar	Menu	Command
	Draw ➤ Arc ➤ Continue	ARC

Related ELLIPSE creates elliptical arcs. PLINE creates arc segments within 2D polylines.

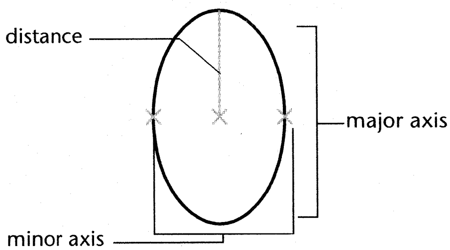
ONLINE HELP See "Creating Arcs."

Drawing Ellipses

You can create full ellipses and elliptical arcs, both of which are exact mathematical representations of ellipses. The longer axis of an ellipse is called the *major* axis, and the shorter one is the *minor* axis. The order in which you define the axes does not matter.



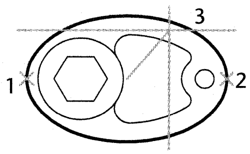
Major axis endpoints defined



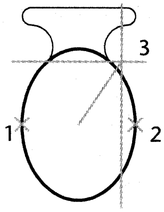
Minor axis endpoints defined

Using Endpoints and Distance

The default method of drawing an ellipse is to specify the endpoints of the first axis (1,2), and the distance (3), which is half the length of the second axis. The illustrations below show ellipses created with the major or the minor axis as the first axis.

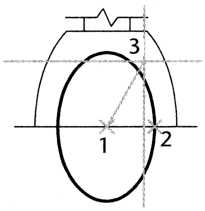


first axis as major axis



first axis as minor axis

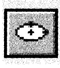
Ellipses created by specifying axis endpoints and distance



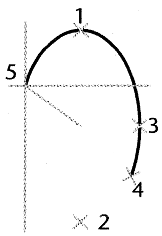
The ellipse at the left is drawn by specifying the center (1) and two axes. The endpoint of the first axis is at (2), and (3) defines half the length of the second axis.

An alternate method of drawing an ellipse is to use a rotation angle instead of a distance. You draw the ellipse by specifying either a center point and an endpoint of one axis or both endpoints of an axis, and then you provide a rotation angle by moving the pointing device or by entering an angle value on the command line.

To draw an ellipse

Draw Toolbar	Menu	Command
	Draw ➤ Ellipse	ELLIPSE

Drawing Elliptical Arcs

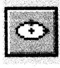


The default method of drawing elliptical arcs uses the first axis endpoints and the second axis distance, as for full ellipses. You then specify start and end angles. These angles are measured from the center of the ellipse and the direction of the major axis. This point is defined as zero degrees. The start angle defines the start point of the elliptical arc. The end angle defines the endpoint, and the arc is drawn between these points in the direction set by the ANGDIR system variable. If ANGDIR is 0, the angles are measured in a counterclockwise direction. If ANGDIR is 1, they are measured in a clockwise direction.

If the start and end angles are the same, you create a full ellipse. You can also specify a start angle and an included angle. The included angle is measured relative to the start point instead of from zero degrees.

In the example, start and end angles are measured from (1), which is the start point of the first axis. ANGDIR is set to 0, so the angles are measured counterclockwise from (1). Point (2) is the second endpoint of the first axis, and point (3) is the distance of the second axis. The start angle (4) is 230 degrees and the end angle (5) is 50 degrees.

To draw an elliptical arc

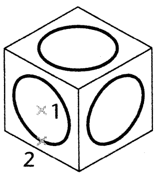
Draw Toolbar	Menu	Command
	Draw ➤ Ellipse ➤ Arc	ELLIPSE

ONLINE HELP See "Creating Ellipses."

Drawing Isometric Circles

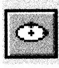
If you are drawing on isometric planes to simulate 3D, you can use ellipses to represent circles viewed from an oblique angle. First you need to turn on an isometric plane (see "Using Isometric Snap and Grid" on page 119).

In the following illustration, (1) is the specified center of the isometric circle, and (2) is the specified radius or diameter.



isometric circles

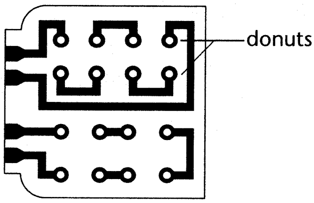
To draw an isometric circle

Draw Toolbar	Menu	Command
	Draw ➤ Ellipse ➤ Axis, End	ELLIPSE

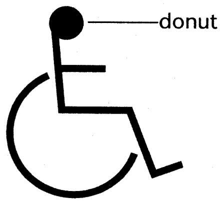
ONLINE HELP See "Drawing Isometric Circles."

Drawing Donuts

Drawing donuts is a quick way to create filled rings or solid-filled circles. Donuts are actually closed polylines having width. To create a donut, specify its inside and outside diameters and its center. You can continue creating multiple copies with the same diameter by specifying different centers until you press ENTER to complete the command. To create solid-filled circles, specify an inside diameter of 0.



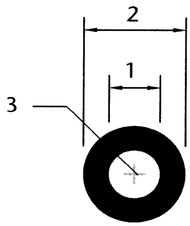
filled rings



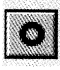
solid-filled circle

Donuts created as filled rings and solid-filled circles

In the following illustration, dimension (1) is the specified inside diameter, dimension (2) is the specified outside diameter, and point (3) is the specified center of the donut.



To draw a donut

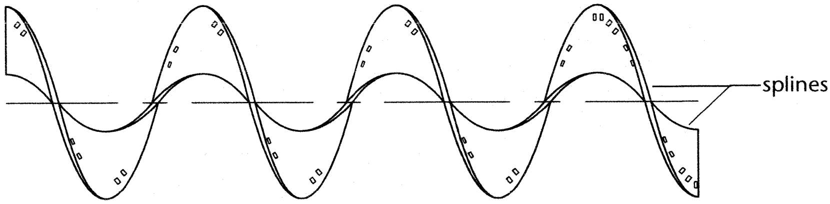
Draw Toolbar	Menu	Command
	Draw ➤ Donut	DONUT

After you've created a donut, you can edit it with PEDIT or convert it to two arcs with EXPLODE. If you explode a donut, its line width reverts to 0.

ONLINE HELP See "Creating Donuts."

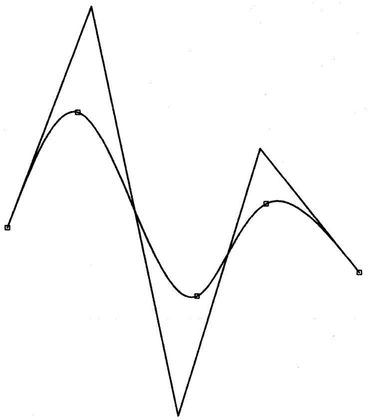
Drawing Spline Curves

A spline is a smooth curve passing through or near a given set of points. AutoCAD LT uses a particular type of spline known as a nonuniform rational B-spline (NURBS) curve. A NURBS curve produces a smooth curve between *control points,* a framework of points that AutoCAD LT uses to define the curve. Splines are useful for creating irregular-shaped curves, for example, drawing complex surfaces for aircraft or automobile design.

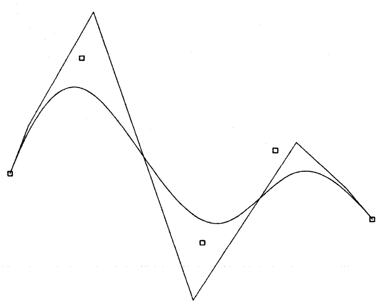


NURBS curves, created with the SPLINE command, are "true" splines. You can also create linear approximations of splines by smoothing polylines with the PEDIT command. You can convert 2D and 3D smoothed polylines to splines with SPLINE.

You create splines by specifying coordinate points. You can close the spline so the start point and endpoint are coincident and tangent. You can also change the spline-fitting *tolerance* while drawing the spline so that you see the effect. Tolerance refers to how closely the spline fits the set of fit points you specify. The lower the tolerance, the more closely the spline fits the points. At zero tolerance, the spline passes through the points.

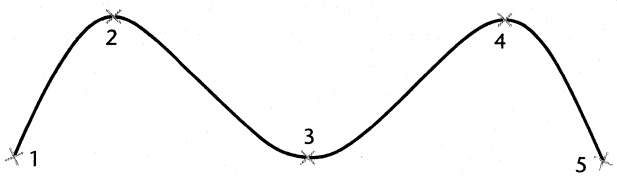


Fit Tolerance = 0

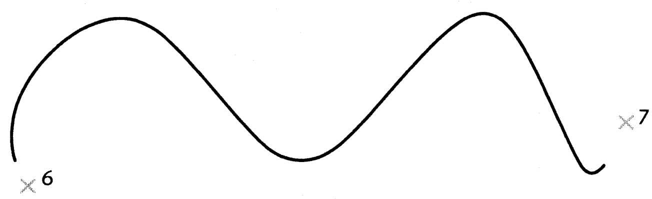


Fit Tolerance = 0.5

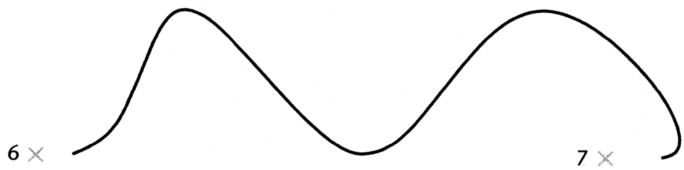
The following example shows how you draw a spline curve by specifying the fit points of the curve (1, 2, 3, 4, 5).



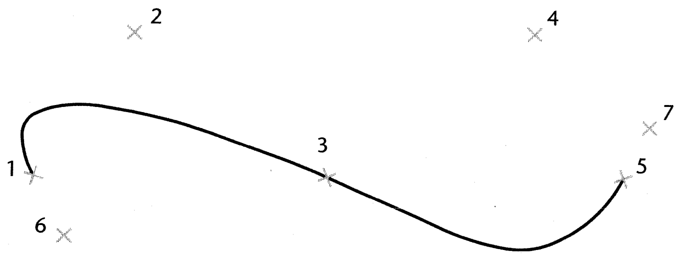
The following illustration is drawn using specified start point and endpoint tangents (6, 7).



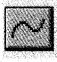
The following illustration shows the result when you use the same points but different start and end tangents.



The following spline is drawn using the same points but a higher tolerance and different start and end tangents.



To draw a spline

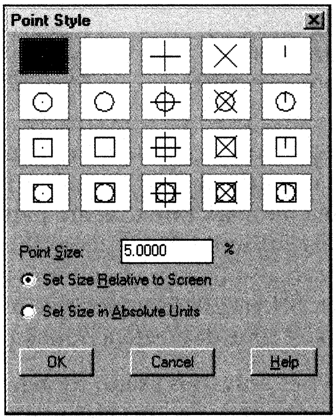
Draw Toolbar	Menu	Command
	Draw ➤ Spline	SPLINE

Related SPLINEDIT edits a spline object. PLINE draws polyline line and arc segments that form a single object. PEDIT can change a polyline into an approximation of a spline.

ONLINE HELP See "Creating Splines."

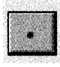
Creating Point Objects

Point objects can be useful, for example, as node or reference points that you can snap to and offset objects from. You can set the style of the point and its size relative to the screen or in absolute units.



From the Format menu, choose Point Style, then make your selections from the Point Style dialog box.

To draw a point

Draw Toolbar	Menu	Command
	Draw ➤ Point	POINT

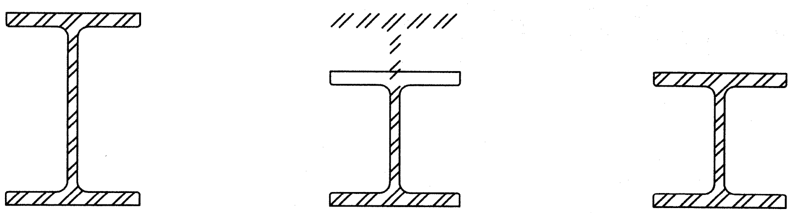
Related DDPTYPE sets the style and size of point objects.

ONLINE HELP See "Creating Points."

Hatching Areas

Hatching fills a specified area in a drawing with a pattern. You can use hatching to highlight or differentiate between areas of your drawing. You can hatch an enclosed area or a specified boundary using the BHATCH and HATCH commands.

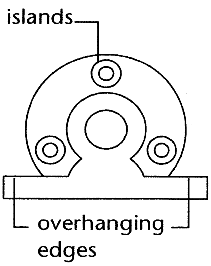
BHATCH creates associative or nonassociative hatches. *Associative* hatches are linked to their boundaries and update when the boundaries are modified. *Nonassociative* hatches are independent of their boundaries.



hatched object

result of editing boundary with nonassociative hatch

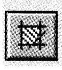
result of editing boundary with associative hatch



BHATCH defines boundaries automatically when you specify a point within the area to be hatched. Any whole or partial objects that are not part of the boundary are ignored and do not affect the hatch. The boundary can have overhanging edges (sections of the boundary line that are part of other objects) and islands (enclosed areas within the hatch area) that you choose to hatch or leave unhatched. You can also define a boundary by selecting objects. Hatched areas created with BHATCH are associative by default. You can remove hatch associativity at any time or create a nonassociative hatch.

HATCH creates nonassociative hatches only. It is useful for hatching areas in your drawing that do not have closed boundaries, see "Using Points to Define a Hatch Boundary" on page 112. HATCH is available only on the command line.

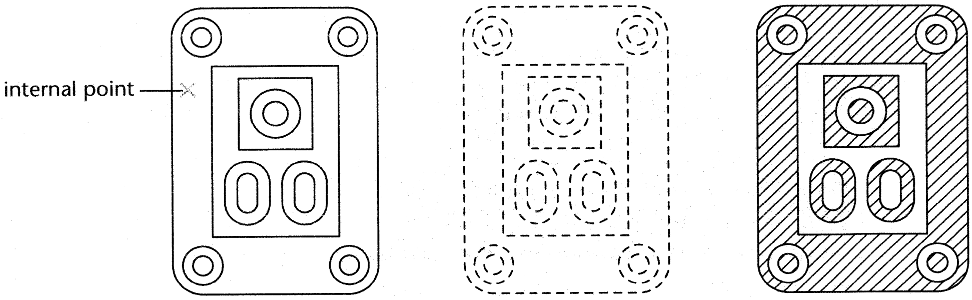
To hatch an enclosed area or selected objects

Draw Toolbar	Menu	Command
	Draw ➤ Hatch	BHATCH

ONLINE HELP To create associative and nonassociative hatches, see "Hatching and Filling Areas."

Defining Hatch Boundaries

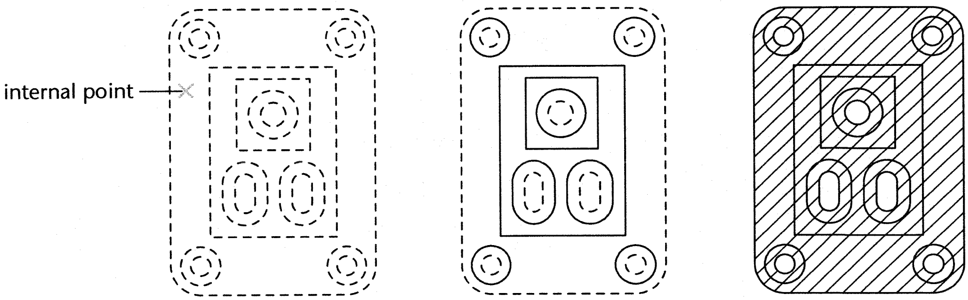
Boundaries can be any combination of lines, arcs, circles, 2D polylines, ellipses, splines, blocks, and paper space viewports. Each boundary component must be at least partly within the current view. By default, AutoCAD LT defines the boundary by analyzing all the closed objects in the current view. If you hatch using the Normal style, islands remain unhatched and islands within islands are hatched, as in the mechanical part shown here.



internal point selected — boundaries detected — result

Hatching Islands

Once all the islands have been detected, you can remove any islands from the hatch area by selecting their boundaries. The following illustration shows how you remove islands so that the part is completely hatched.

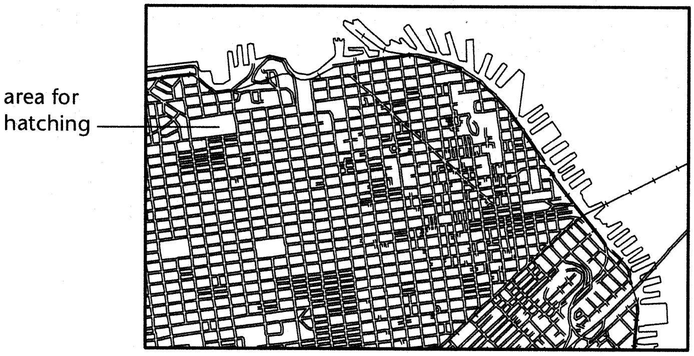


boundaries detected — islands to remove (solid boundaries) — result

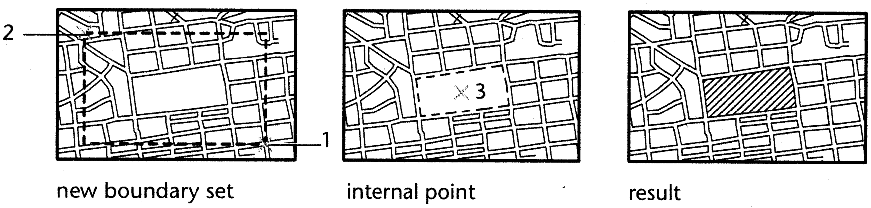
Using Boundary Sets

In a complex drawing, full or partial analysis of all potential boundaries can be time-consuming. To hatch a small area of a complex drawing, you can define a boundary set. A boundary set is a set of objects in the drawing. When you specify a point within the boundary set, AutoCAD LT ignores objects not included in the boundary set. Boundary sets are also useful for applying different hatch styles in different sections of a drawing.

You can define a boundary set within a map and hatch only part of the area. For clarity, first zoom into the area you want to hatch.



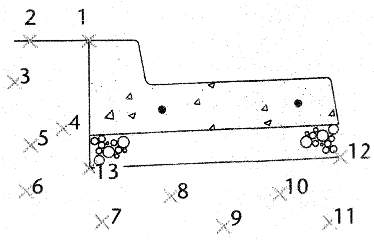
In the following illustration, (1) and (2) are the specified corner points of the boundary set; (3) is the internal point selected. AutoCAD LT analyzes only the portion of the drawing within the boundary set to apply the correct hatch area.



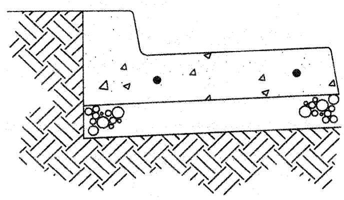
Using Points to Define a Hatch Boundary

Use the HATCH command to define a hatch boundary by specifying points directly. For example, you might want to illustrate that a whole area of a drawing is filled with a pattern by filling just a small section of that area, as shown in the next illustration.

In the following example, the hatch area was defined by specifying points directly. The hatch pattern is EARTH, and it is rotated 45 degrees. Once the hatch was created, the boundary polyline was discarded.



points specified to define hatch boundary



result

To define a boundary by specifying points

1. At the Command prompt, enter **hatch**, and then enter **earth** to select the Earth pattern.
2. Specify the scale and angle (in this case 1.0 for scale and 45 degrees for angle) for the pattern.
3. At the Select Objects prompt, press ENTER.

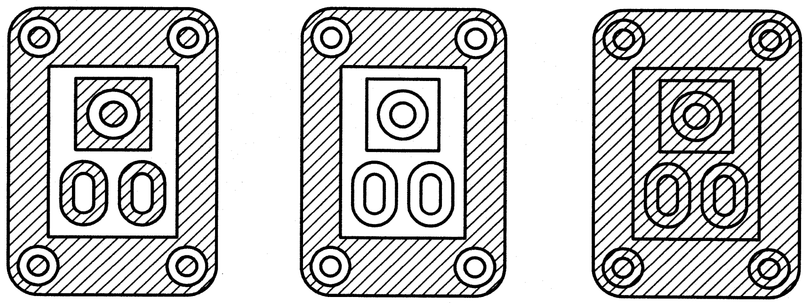
   AutoCAD LT creates a boundary from the points you specify.
4. Enter **n** to discard the polyline boundary once it has been defined. Specify the points that define the boundary (1–13). Then, enter **c** to close the boundary and press ENTER.

ONLINE HELP See "Defining Hatch Boundaries."

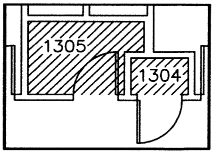
Using Hatch Styles

You can further determine how islands are hatched using the three hatching styles: Normal, Outer, and Ignore.

- Normal style hatches inward from the outer boundary. If it encounters an internal intersection, it turns off hatching until it encounters another intersection. Thus, areas separated from the outside of the hatched area by an odd number of intersections are hatched, and areas separated by an even number are not.
- Outer style hatches inward from the outer boundary and stops at the next boundary.
- Ignore style hatches the entire enclosed area, ignoring internal boundaries.



Hatch styles



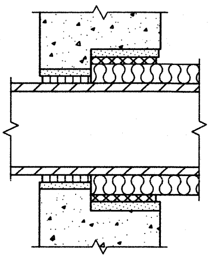
default hatching of text

If a hatch line encounters a text, attribute, shape, trace, or solid-fill object, and if the object is selected as part of the boundary set, AutoCAD LT does not hatch through the object. You can draw a pie slice, for example, label it with text, and hatch it, and the text will remain readable. If you want to hatch through such objects, use the Ignore style.

ONLINE HELP See "Controlling How Islands Are Hatched."

Using Hatch Patterns

AutoCAD LT supplies a solid fill and more than 50 industry-standard hatch patterns. Hatch patterns highlight a particular feature or area of a drawing. For example, patterns can help differentiate the components or represent the materials that make up an object. AutoCAD LT supplies 14 hatch patterns that conform to the ISO (International Standardization Organization) standards. These patterns are listed at the end of the pattern list in the Boundary Hatch and Hatch Pattern Palette dialog boxes. When you select an ISO pattern, you can also specify a pen width, which determines the line weight in the pattern.



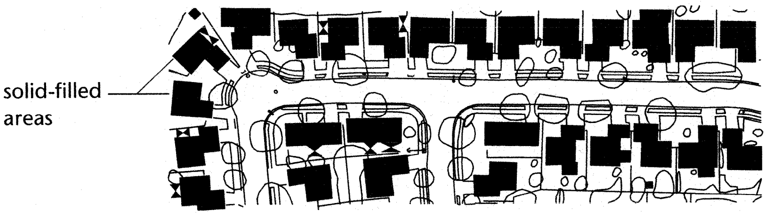
industry-standard hatch patterns

You can use a pattern supplied with AutoCAD LT or one from an external pattern library. These patterns are also listed by name and shown in the Hatch Pattern Palette dialog box. You can define a simple line pattern using the current linetype with the User-Defined Pattern option, or you can create more complex hatch patterns. To reduce file size, a hatch pattern is defined in the drawing database as a single graphic object.

ONLINE HELP See "Using Hatch Patterns."

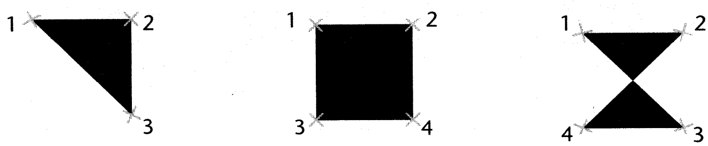
Creating Solid-Filled Areas

As an alternative to solid hatch patterns, AutoCAD LT provides 2D polygon solids. You can create triangular and quadrilateral areas filled with a color. For quicker results, you can create these areas with the FILLMODE system variable off, and then turn on FILLMODE to fill the finished area. You don't see the area outline until it is complete.



Solid-filled areas used to depict buildings

When you create a quadrilateral solid-filled area, the sequence of the third and fourth points determines its shape. Compare the following illustrations:



To create a solid-filled area

Command
SOLID

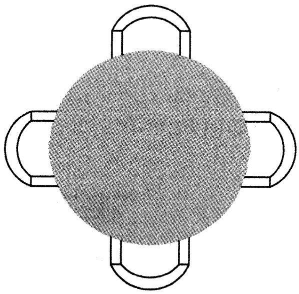
ONLINE HELP See "Creating Solid-Filled Areas."

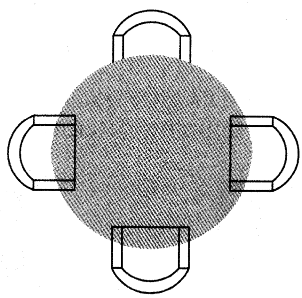
Changing the Drawing Order of Objects

By default, objects are drawn in the order they are created. DRAWORDER changes the display order of objects, moving one in front of another, for example.

Ordering ensures proper display and plotting output when two or more objects overlay one another. An example of when ordering may be necessary is when a raster image is drawn over existing objects, obscuring them from view.

Use Bring to Front or Send to Back to specify the drawing order of the current objects. Use Bring Above Object or Send Under Object to select the object you want to move above or below.


Draworder with hatch in front

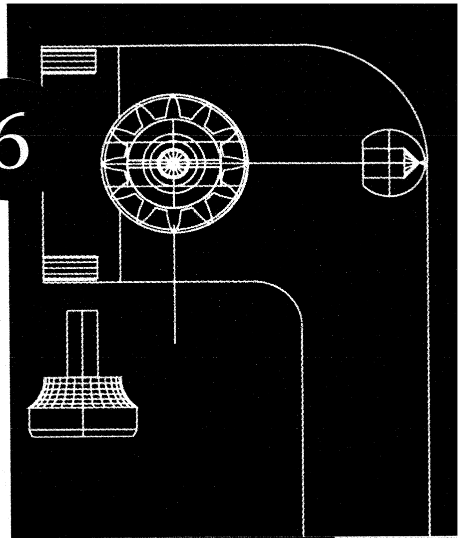

Draworder with hatch in back

To change the display order of objects

Standard Toolbar	Menu	Command
	Tools ➤ Display Order	DRAWORDER

ONLINE HELP See "Changing the Display Order of Overlapping Objects."

6



Drawing with Precision

With AutoCAD LT you don't need to perform tedious calculations to create precise drawings. You can use drawing aids to specify precise points without knowing the coordinates. You can also display various types of object property and status information.

In this chapter

- Changing the grid and snap alignment
- Using Ortho and PolarSnap to restrict point selection
- Using direct distance entry
- Snapping to geometric points on objects
- Using point filters and the tracking feature to enter coordinates
- Creating construction lines
- Calculating distances, angles, and areas
- Listing object information

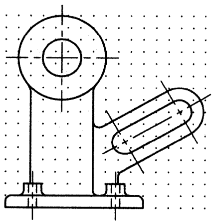
Adjusting Snap and Grid Alignment

You can use the grid as a visual guideline and turn on Snap to restrict cursor movement. For more information, see "Setting the Grid" on page 71, and "Setting Grid and Snap Spacing" on page 73.

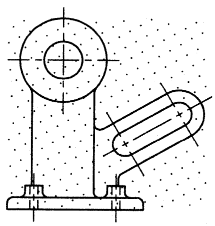
In addition to setting the spacing, you can adjust the snap and grid alignment. You can rotate the alignment, or you can set it for use with isometric drawings.

Changing the Snap Angle and Base Point

If you need to draw along a specific alignment or angle, you can rotate the snap angle. This rotation realigns the crosshairs and constrains the cursor to the new alignment when Snap or Ortho is on. In the following example, the snap angle is adjusted to match the 30-degree angle of the anchor bracket. With this adjustment, you can use the grid to draw objects at a 30-degree angle.



default snap angle



rotated snap angle

When you set the snap angle in the Drawing Aids dialog box, the grid angle also changes.

The center point of the snap angle rotation is the snap base point. If you need to align a hatch pattern, you can change this point, which is normally set to 0,0.

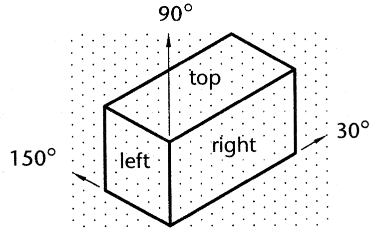
To change the snap angle and base point

Menu	Command
Tools ➤ Drawing Aids	DDRMODES

Using Isometric Snap and Grid

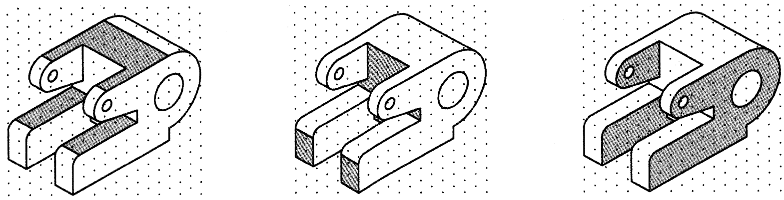
The Isometric Snap/Grid setting helps you create 2D drawings that represent 3D objects, such as cubes. Isometric drawings are not true 3D drawings, because they cannot be viewed in perspective or from another angle. However, you can simulate 3D by aligning each isometric plane along three major axes. If the snap angle is 0, the axes of the isometric planes are 30 degrees, 90 degrees, and 150 degrees. You can select from three planes, each with an associated pair of axes:

- Left orients the snap and grid alignment along the 90- and 150-degree axes.
- Top orients the snap and grid alignment along the 30- and 150-degree axes.
- Right orients the snap and grid alignment along the 90- and 30-degree axes.



isometric drawing planes

Choosing an isometric plane causes the snap intervals, grid, and crosshairs to realign along the corresponding isometric axes. AutoCAD LT restricts certain point selections to two of three axes under certain conditions. For example, when Ortho is on, the points you select when drawing objects will align along the plane on which you are drawing. Therefore, you can draw the top plane of a model, switch to the left plane to draw another side, and switch to the right plane to complete the drawing.


top plane left plane right plane

Planes of a model

To turn on isometric snap and grid

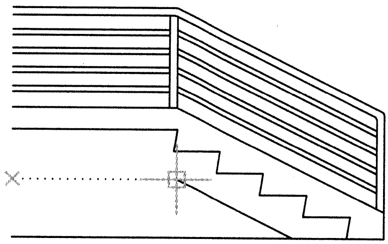
Menu	Command
Tools ➤ Drawing Aids	DDRMODES

Related You can cycle through the planes by pressing F5 or CTRL+E.

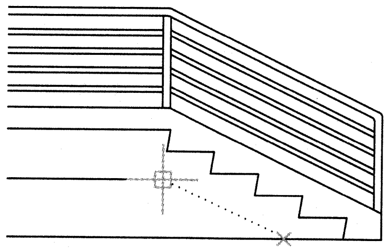
ONLINE HELP See "Creating Realistic Images."

Using Ortho and PolarSnap

AutoCAD LT provides drawing and editing tools similar to the drafter's parallel rule and triangle. You can use Ortho to restrict rubber-band cursor movement to the horizontal or vertical axis, as with the parallel rule, or use PolarSnap to restrict rubber-band cursor movement to specified angles, as with the triangle. You can also use PolarSnap to specify snap distances.



Ortho restricts the rubber-band cursor movement to horizontal

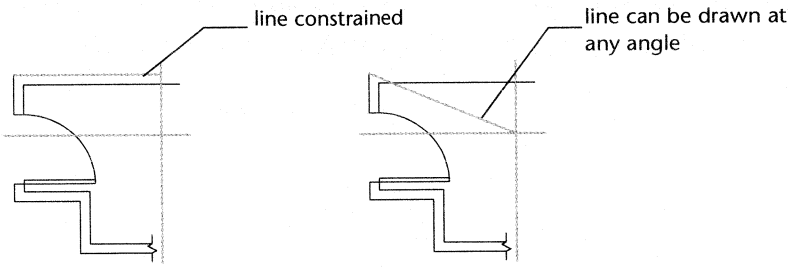


PolarSnap restricts the rubber-band cursor movement to a specific angle

Using Ortho

As you draw lines or move objects, you can use Ortho to restrict the movement of the rubber-band cursor to the horizontal or vertical axis. (The orthogonal alignment depends on the current snap angle or UCS.) Ortho works with activities that require you to specify a second point. You can use Ortho not only to establish vertical or horizontal alignment but to enforce parallelism or create regular offsets.

By setting orthogonal restraints, you can draw more quickly. For example, you can create a series of perpendicular lines by turning on Ortho before you start drawing. Because the lines are constrained to the horizontal and vertical axes, you can draw faster, knowing that the lines are perpendicular.



Ortho on Ortho off

As you move the cursor, a rubber-band line that defines the displacement follows the horizontal or vertical axis, depending on which axis is nearest to the cursor. AutoCAD LT ignores Ortho in perspective views and when you enter coordinates on the command line or specify an object snap.

For drawing or editing objects at angles other than right angles, use PolarSnap. See the following section, "Using PolarSnap" for more information.

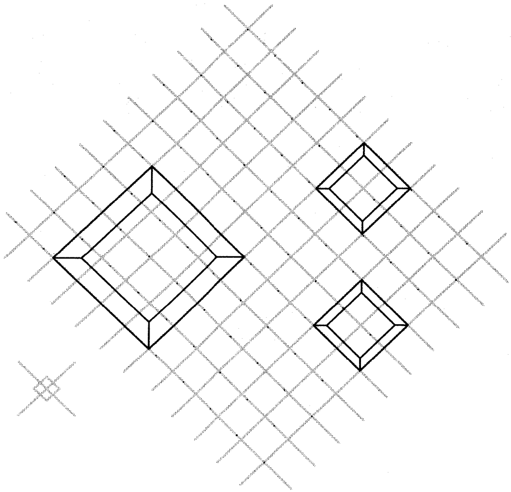
To turn Ortho on or off

Status Bar	Menu	Command
ORTHO	Tools ➤ PolarSnap Settings	ORTHO

Related You can turn Ortho on and off by pressing CTRL+L or F8.

Using PolarSnap

When you are creating or modifying objects, you can use PolarSnap to restrict the rubber-band cursor movement to precise angles and distances. You can use PolarSnap to specify snap angle, snap distance, or both. Also, you can choose to activate polar tooltips that display the distance and angle of the cursor from the last selected point. PolarSnap is active only when Ortho is on.



Use PolarSnap to control an object's length and angle.

AutoCAD LT overrides PolarSnap when you enter coordinates on the command line, use perspective views, specify an object snap, or use direct distance entry. For direct distance entry, AutoCAD LT overrides only the distance setting of PolarSnap, not the angle setting.

Using Angle Snap

Angle snap restricts drawing or editing cursor movement to angular increments you specify in the Ortho/Polar Settings dialog box. Once you set the snap angle, the cursor snaps at 0 degrees and at increments of the specified angle. For example, if the snap angle value is 15 degrees, the cursor snaps at 0, 15, 30, 45, and so on. The specified angle is relative to the UCS and the current grid rotation angle.

The orientation of 0 is dependent on the snap angle that you set when specifying angle conventions, and the direction of snap (clockwise or counterclockwise) is dependent on the units direction you specify when setting units of measurement. See "Setting Units Style" on page 69.

Using Distance Snap

Distance snap restricts cursor movement to increments of a length you specify. For example, if you specify a length of 4 units, the cursor snaps from the first point specified to lengths of 4, 8, 12, 16, and so on. The minimum distance you can specify is zero.

To turn PolarSnap on and off

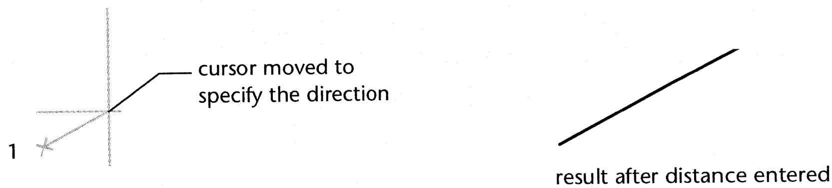
Standard Toolbar	Menu	Command
	Tools ➤ PolarSnap Settings	POLAR

Using Direct Distance Entry

Instead of entering coordinate values, you can use direct distance entry to specify a point by moving the cursor to indicate a direction and then entering the distance from the first point. This is a good way to specify a line length quickly.

You can use direct distance entry to specify points for all commands except those that prompt you to enter a single real value, such as ARRAY, MEASURE, and DIVIDE. When Ortho is on, this method is an efficient way to draw perpendicular lines. In the following example, you draw a line 25 units long using direct distance entry.

Command: *Enter* **line**
From point: *Specify the first point, and then move the pointing device in the direction you want to draw*
To point: *Enter* **25**
To point: *Press* ENTER



The line is drawn at the length and angle you specified.

Snapping to Geometric Points on Objects

Object snaps specify exact locations, such as a midpoint, endpoint, or an intersection, on existing objects. Using object snaps is a quick way to locate an exact position on an object without having to know the coordinate or draw construction lines. For example, you can use an object snap to draw a line to the center of a circle, to the midpoint of a polyline segment, or to an imaginary intersection.

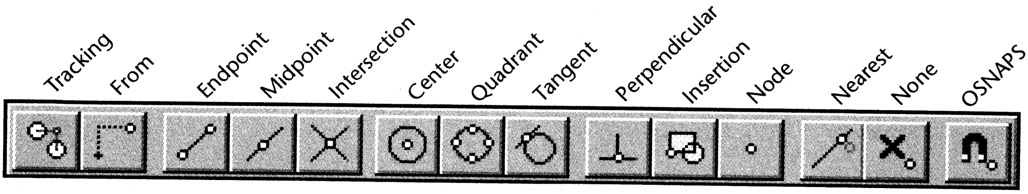
You can specify an object snap whenever AutoCAD LT prompts for a point. One of the easiest ways to specify an object snap is to use the cursor menu. When AutoCAD LT prompts for a point, hold down SHIFT, and then right-click. Choose the object snap from the displayed cursor menu.

Single object snaps affect only the next object you select. You also can turn on one or more running object snaps. Running object snaps stay active until you turn them off.

Most of the object snaps that are described here affect only objects visible on the screen, including objects on locked layers, floating viewport boundaries, solids, and polyline segments. You *cannot* snap to objects on turned-off or frozen layers or to the blank portions of dashed lines.

You can use the From object snap to establish a temporary reference point for use with relative offsets. For more information, see "Offsetting from Temporary Reference Points" on page 131.

Using Object Snap for a Single Point



Object Snap flyout accessed from the Standard toolbar

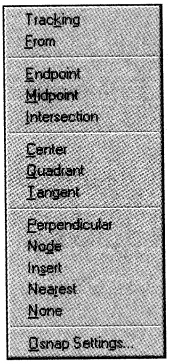
AutoCAD LT provides several visual cues that make specifying an object snap easy and accurate. By default, AutoSnap locates an object snap with a marker when you move your cursor over an object. For more information about Autosnap, see "Setting Running Object Snaps" on page 132.

Each object snap has a unique abbreviation that you can enter on the command line. Use commas to separate multiple object snaps on the command line.

Object snaps work only when AutoCAD LT prompts for a point. If you try to use an object snap at the Command prompt, AutoCAD LT displays an error message.

To choose object snaps from the cursor menu, hold down SHIFT and right-click. Then choose the object snap you want to use. To set running object snaps, choose Osnap Settings.

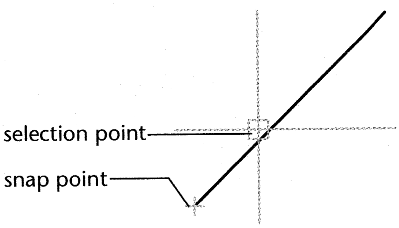
ONLINE HELP See "Using Object Snaps."



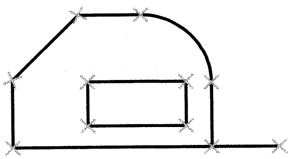
Cursor menu

Endpoint

Endpoint snaps to the closest endpoint of objects such as lines or arcs.



Endpoint snap



endpoints

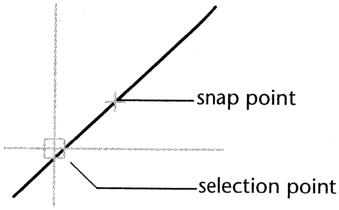
If you have given an object thickness, you also can snap to the projected edges of the object.

To snap to the endpoint of an object

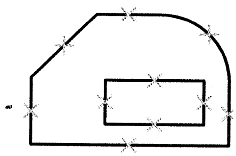
Standard Toolbar	Menu	Command
Object Snap Flyout	Cursor ➤ Endpoint	ENDP

Midpoint

Midpoint snaps to the midpoint of objects such as lines or arcs.



Midpoint snap



midpoints

On infinite lines, Midpoint snaps to the first point defined (the root). When you select a spline or an elliptical arc, Midpoint snaps to a point on the object that is halfway between the starting point and the endpoint.

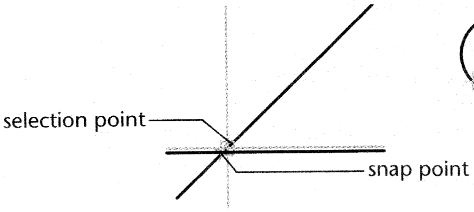
If you have given a line or arc thickness, you also can snap to the midpoints of the projected edges of the object.

To snap to the midpoint of an object

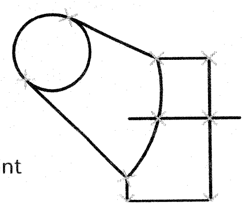
Standard Toolbar	Menu	Command
Object Snap Flyout	Cursor ➤ Midpoint	MID

Intersection

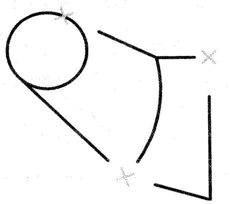
Intersection snaps to the intersection of objects such as lines, circles, arcs, and splines. If the target area only covers one object, AutoCAD LT prompts you to select a second object and then snaps to the imaginary intersection formed by extensions of those objects.



Intersection snap



intersections



extended intersections

Also, Intersection snaps to the corners of objects that have been given thickness. If two thickened objects extend in the same direction and have intersecting bases, you can snap to the intersection of the edges. If the objects have different thicknesses, the lesser thickness defines the intersection point.

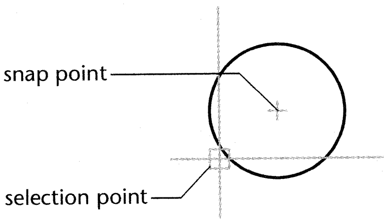
Intersections of arcs and circles that are part of a block (a group of objects that are treated as a single object) work only if the block has been uniformly scaled. Line intersections within blocks work normally.

To snap to an intersection of objects

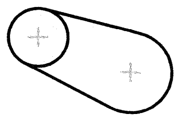
Standard Toolbar	Menu	Command
Object Snap Flyout	Cursor ➤ Intersection	INT

Center

Center snaps to the center of an arc, circle, or ellipse.



Center snap



centers

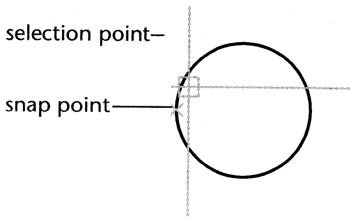
When you snap to the center, select a visible part of the arc, circle, or ellipse, not the actual center.

To snap to the center of a circle, arc, or ellipse

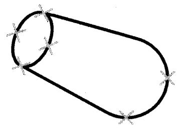
Standard Toolbar	Menu	Command
Object Snap Flyout	Cursor ➤ Center	CEN

Quadrant

Quadrant snaps to the closest quadrant of an arc, circle, or ellipse (the 0-, 90-, 180-, and 270-degree points).



Quadrant snap


quadrants

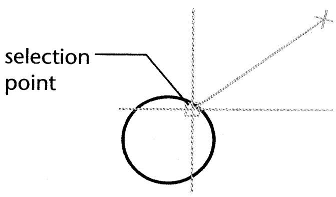
The location of the quadrant points for circles and arcs is determined by the current orientation of the UCS. If the arc, circle, or ellipse is a member of a rotated block, the quadrant points rotate with the block.

To snap to the quadrant of a circle, arc, or ellipse

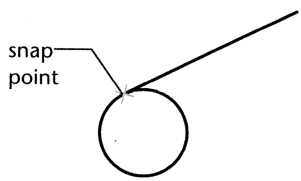
Standard Toolbar	Menu	Command
Object Snap Flyout	Cursor ➤ Quadrant	QUA

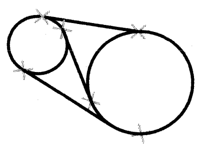
Tangent

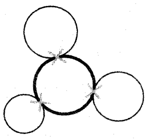
Tangent snaps to the point on a circle, arc, ellipse, or spline that, when connected to the last point, forms a line tangent to that object.



Tangent snap




tangents



circle tangent to three circles

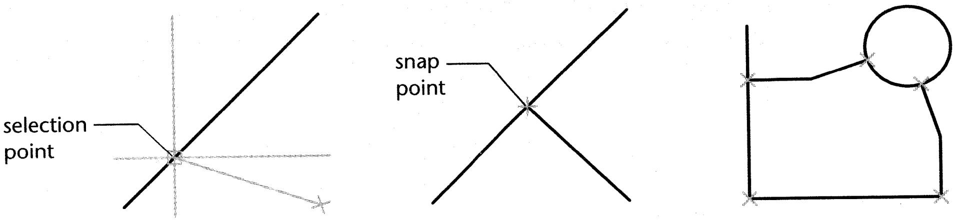
When drawing a circle with the three-point method, for example, you can use the Tangent object snap to construct a circle tangent to three other circles.

To snap to a tangent to a circle, arc, ellipse, or spline

Standard Toolbar	Menu	Command
Object Snap Flyout	Cursor ➤ Tangent	TAN

Perpendicular

Perpendicular snaps to the point on an object that forms a normal, or perpendicular, alignment with another object or with an imaginary extension of that object. You can use the Perpendicular object snap with objects such as lines, circles, ellipses, splines, or arcs.



Perpendicular snap

perpendiculars

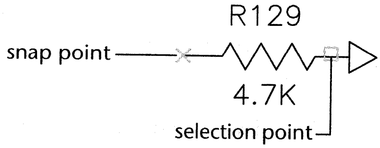
When you use the Perpendicular object snap to specify the first point of a line or circle, you construct a line or circle that is perpendicular to the object you select. When you use the Perpendicular object snap to specify the second point of a line or circle, AutoCAD LT snaps to a point that creates a normal from that object to the first point selected.

To snap to a point on an object that forms a perpendicular alignment with another object

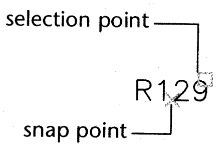
Standard Toolbar	Menu	Command
Object Snap Flyout	Cursor ➤ Perpendicular	PER

Insertion

Insertion snaps to the insertion point of a block, text, attribute (which contains information about a block), or attribute definition (which describes the characteristics of the attribute).



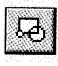
Insertion snap for block



Insertion snap for centered text

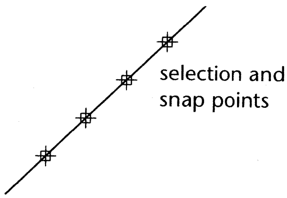
If you select an attribute within a block, AutoCAD LT snaps to the insertion point of the attribute, not the block. By pressing TAB, you can cycle through the insertion points of the block and any additional text, attributes, or attribute definitions.

To snap to the insertion point of a block, text, attribute, or attribute definition

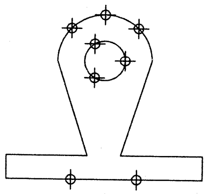
Standard Toolbar	Menu	Command
 Object Snap Flyout	Cursor ➤ Insert	INS

Node

Node snaps to a point object drawn with the POINT command.



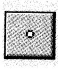
Node snap



nodes

Points included in a block can function as convenient snap points for attachment locations.

To snap to a point object

Standard Toolbar	Menu	Command
 Object Snap Flyout	Cursor ➤ Node	NODE

Nearest

Nearest snaps to a point object or to the location on another type of object that is closest to the specified point.

To snap to a point object or nearest location on another object type

Standard Toolbar	Menu	Command
Object Snap Flyout	Cursor ➤ Nearest	NEA

None

None turns off running object snaps for one point.

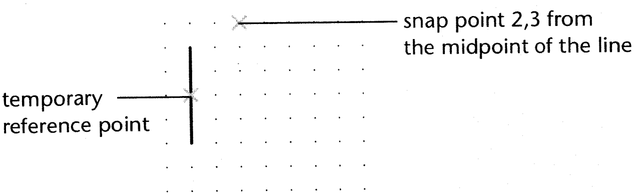
To turn off running object snaps

Standard Toolbar	Menu	Command
Object Snap Flyout	Cursor ➤ None	NONE

Offsetting from Temporary Reference Points

The From object snap differs from the other types of object snaps because it establishes a temporary reference point as a basis for specifying subsequent points. The From object snap is normally used in combination with other object snaps and relative coordinates.

For example, at a prompt for a point when drawing a polyline, you can enter **from mid**, select a line, and then enter **@2,3** to locate a point two units to the right and three units up from the midpoint of the line.



Command: *Enter* **pline**
From point: *Enter* **from**
Base point: *Enter* **mid**
of: *Select the line*
of <Offset>: *Enter* **@2,3**

You can specify an absolute coordinate for use as a base point; however, specifying an absolute coordinate for the offset essentially cancels From Object Snap and locates the point at the specified coordinate.

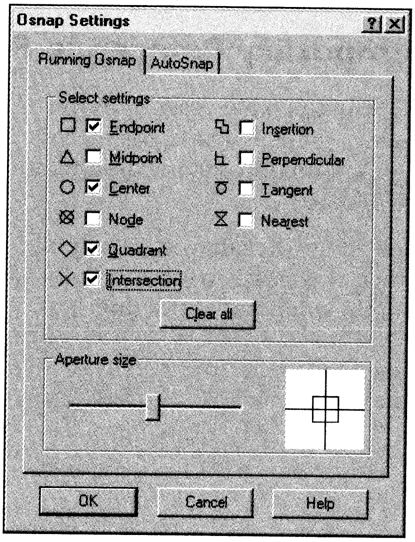
To specify a From object snap

Standard Toolbar	Menu	Command
Object Snap Flyout	Cursor ➤ From	FROM

ONLINE HELP See "Offsetting from Temporary Reference Points."

Setting Running Object Snaps

If you need to set the same object snap repeatedly, you can set it as a running snap, which means it stays active until you turn it off. For example, you might set Center as a running snap if you need to connect the centers of a series of circles with a line.



With running object snaps, you can set single as well as multiple location types, such as endpoints and center points. As with single object snaps, AutoSnap is on by default and displays a marker indicating the available object snaps.

When you turn on multiple object snaps, AutoCAD LT uses the object snap most appropriate to the object you select. If two potential snap points are located on the object you select, AutoCAD LT snaps to the eligible snap point closest to the cursor. By pressing TAB, you can cycle through the available snap points on the selected object.

NOTE The From object snap *cannot* be set as a running object snap.

You can quickly turn object snaps on and off without having to respecify your object snap settings by double-clicking Osnap on the status bar. Double-clicking Osnap when no object snaps are set displays the Osnap Settings dialog box. You can also switch running object snaps on and off by pressing CTRL+F or F3.

To set running object snaps

Status Bar	Menu	Command
OSNAP	Tools ➤ Object Snap Settings	DDOSNAP

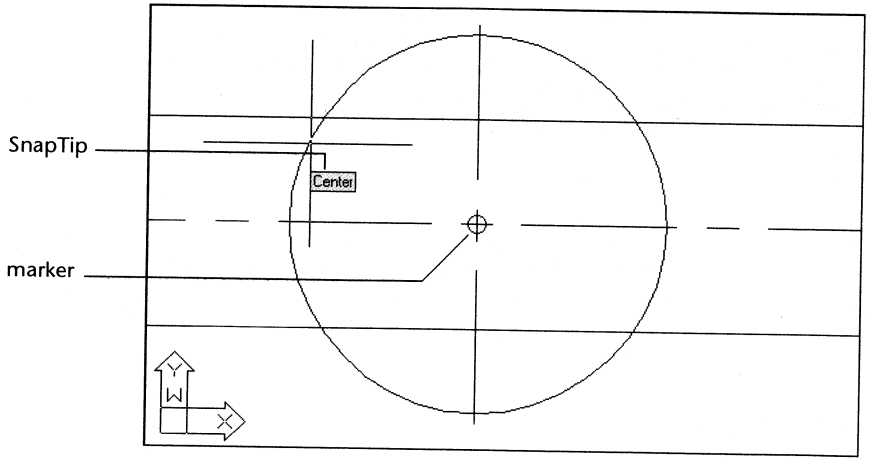
ONLINE HELP See "Setting Running Object Snaps."

Using AutoSnap

When you use any of the object snap settings, you may notice that a marker and a SnapTip are displayed when you move the cursor over a snap point. AutoSnap is automatically turned on when you specify a single object snap or turn on running object snaps in the Osnap Settings dialog box.

After you enter a drawing command, AutoSnap indicates the snap points as you move your pointing device over the object.

With the cycling feature, you can cycle through all the snap points available for a particular object by pressing TAB. For example, if you press TAB while the cursor is on the circle in the following illustration, AutoSnap gives you the option to snap to quadrant, intersection, and center.



AutoSnap displays visual cues when you move your cursor over an object.

- ***Marker*** A geometric symbol that displays the object snap location when the cursor moves over snap points of an object.
- ***SnapTip*** A small label near the cursor that describes which part of the object you are snapping to.
- ***Magnet*** An automatic movement of the cursor that locks the cursor onto the snap point.

An aperture box is displayed at the center of the crosshairs after you select one or more object snaps (off by default).

As an alternative to AutoSnap, turn on an aperture box to define the selection area. Specifying an object snap converts the cursor to an aperture box. When you select an object, AutoCAD LT snaps to the eligible snap point closest to the center of the aperture box.

To control AutoSnap and aperture box settings

Menu	Command
Tools ➤ Object Snap Settings ➤ AutoSnap tab	OSNAP

ONLINE HELP See "Using Object Snaps."

Using Point Filters

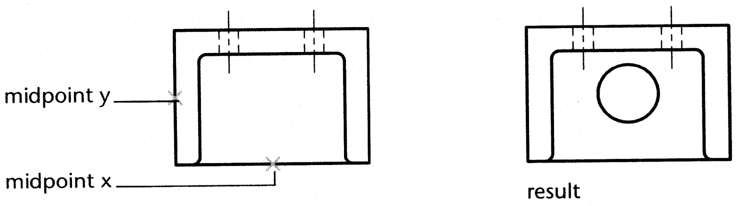
With point filters, you can specify one coordinate value at a time while temporarily ignoring the other coordinate values. Used with object snaps, point filters can extract coordinate values from an existing object so you can locate another point.

Specifying a point filter limits the next entry to a specific ordinate value, such as the *X* or the *Y* value, or even an *XY* coordinate value. (For 3D points, you also can specify *Z* values.) After you specify the first value, AutoCAD LT prompts for the remaining values.

At the prompt for a point on the command line, you can enter **.x** or **.y** or **.z** to indicate which value you want to specify.

Example

In the following illustration, the hole in the holding plate was centered in the rectangle by extracting the *X* and *Y* values from the coordinates of the midpoints of the plate's horizontal and vertical line segments.



The following example shows how to use point filters to create the hole.

Command: *Enter* **circle**
3P/2P/TTR/<Center point>: *Enter* **.x**
of: *Enter* **mid**
of: *Select the horizontal line on the lower edge of the holding plate*
of: (need YZ): *Enter* **mid**
of: *Select the vertical line on the left side of the holding plate*
of: Diameter/<Radius> *Specify the radius of the hole*

ONLINE HELP See "Using Coordinate Filters."

Using Tracking

In addition to using point filters, you can use tracking to visually locate points relative to other points in your drawing. Typically, you use tracking in combination with direct distance entry.

You can use tracking whenever AutoCAD LT prompts for a point. When you start tracking and specify a point, AutoCAD LT constrains the next point selection to an orthogonal path that extends vertically or horizontally from the first point.

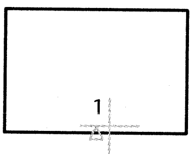
The orthogonal direction determines which of the old point's values, *X* or *Y*, is replaced with the new point's *X* or *Y* value. If the rubber-band line is constrained to horizontal, then the *X* value is replaced. If the rubber-band line is constrained to the vertical, then the *Y* value is replaced.

If you select a second point and press ENTER to end tracking, AutoCAD LT locates the new point at the intersection of an imaginary orthogonal path extending from the first two points.

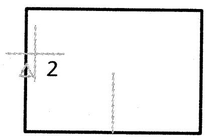
You can use tracking with direct distance entry to, for example, place text a specified distance from another object, to insert windows or doors in walls at a specified distance from a corner, or to break a wall.

The following example shows how to draw a line from the center point of a rectangle.

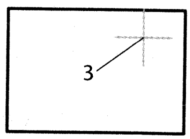
Command: *Enter* **line**
LINE From point: *Enter* **tk**
First tracking point: *Enter* **mid**
of: *Select the horizontal line of the rectangle* (1)
Next point (press ENTER to end tracking): *Enter* **mid**
of: *Select the vertical line of the rectangle* (2)
Next point (press ENTER to end tracking): *Press* ENTER *to end tracking*



first midpoint specified



second midpoint specified



line starts at the orthogonal intersection

Once you start tracking, AutoCAD LT does not place the point until you turn off tracking by pressing ENTER. Therefore, you can track as many points as you need.

To use tracking when AutoCAD LT prompts for a point

Standard Toolbar	Menu	Command
Object Snap Flyout	Cursor ➤ Tracking	TRACKING

Tracking can only be used when AutoCAD LT prompts for a point. If you try to use tracking at the Command prompt, AutoCAD LT displays an error message.

ONLINE HELP See "Using Tracking to Offset Points."

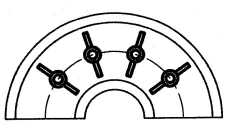
Specifying Measurements and Divisions

Sometimes you need to create points or blocks at specific intervals on an object. For example, you may need to snap to points at half-unit intervals or insert markers on an object to identify five equal segments. You can use one of the following commands:

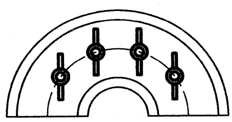
- To specify the length of each segment, use MEASURE
- To specify the number of equal segments, use DIVIDE

You can measure or divide lines, arcs, splines, circles, ellipses, and polylines. With both methods, you can identify the intervals by inserting either a point or a named set of objects known as a block.

By specifying points, you then can use the Node object snap to align other objects at even intervals on the measured or divided object. By specifying blocks, you can create precise geometric constructions or insert custom markers. You can rotate blocks at each insertion point.



blocks aligned



blocks not aligned

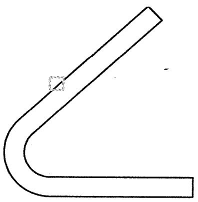
To be inserted, blocks must already be defined within the drawing. Any variable attributes within the block are excluded from the insertion. For more information about blocks, see "Working with Blocks" on page 289.

The starting point for measurements or divisions varies with the object type. For lines or polylines, segments start at the endpoint closest to the selection point. Segments in closed polylines begin at the polyline's starting point. Segments in circles start at the angle from the center point that is equivalent to the current snap angle. For example, if the snap angle is 0, the circle starts at the three o'clock position and continues in a counterclockwise direction.

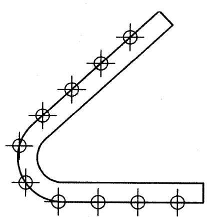
Specifying Measured Intervals on Objects

You use the MEASURE command to mark intervals on an object. You can mark the intervals with either points or blocks.

In the following illustration, an object has points inserted at measured intervals.



object selected



points at measured intervals

To mark intervals on an object

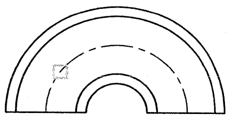
Menu	**Command**
Draw ➤ Point ➤ Measure	MEASURE

ONLINE HELP See "Specifying Intervals on an Object."

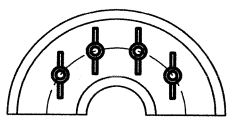
Dividing Objects into Segments

Use the DIVIDE command to create points or blocks on an object at a specific number of equal intervals. This command does not actually break the object into individual objects; it only identifies the location of the divisions so that you can use them as geometric reference points.

In the following illustration, an object is divided into equal segments.



object selected



points indicating divisions

To divide an object into equal segments

Menu	Command
Draw ➤ Point ➤ Divide	DIVIDE

ONLINE HELP See "Dividing an Object into Equal Segments." To use DIVIDE and MEASURE to place blocks, see "Specifying Intervals on an Object."

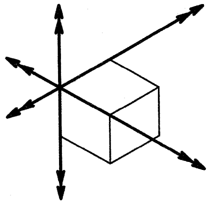
Drawing Construction Lines

You can create construction lines that extend to infinity in one or both directions. Construction lines that extend in one direction are known as *rays*. Construction lines that extend in both directions are known as *xlines*. These construction lines can be used as a reference for creating other objects. For example, you can use construction lines to find the center of a triangle, prepare multiple views of the same item, or create temporary intersections that you can use for object snaps.

Construction lines do not change the total area of the drawing; therefore, their infinite dimensions have no effect on zooming or viewpoints. You can move, rotate, and copy construction lines just as you move, rotate, and copy other objects. You may want to create construction lines on a construction line layer that can be frozen or turned off before plotting. See "Freezing Layers" on page 77.

Creating Construction Lines (XLines)

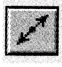
A construction line, or *xline*, can be placed anywhere in 3D space and extends to infinity in both directions. You can specify its orientation in several ways. The default method for creating the line is the two-point method: you select two points to define the orientation. The first point, the *root*, is considered the midpoint of the construction line.



construction lines

Commands that display the drawing extents ignore construction lines.

To create a construction line

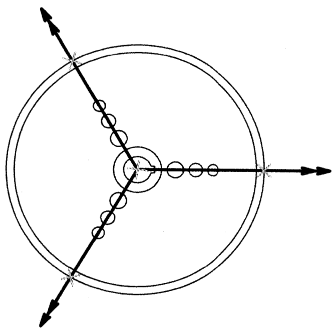
Draw Toolbar	Menu	Command
	Draw ➤ Construction Line	XLINE

You can create construction lines in several other ways, including specifying an angle from the horizontal axis along which the construction line is drawn and offsetting the construction line from a baseline you specify.

ONLINE HELP See "Creating Construction Lines."

Creating Rays

A ray is a line in 3D space that starts at a point you specify and extends to infinity. Unlike xline construction lines, which extend in two directions, rays extend in only one direction. As a result, rays help reduce the visual clutter caused by numerous construction lines. Like construction lines, rays are ignored by commands that display the drawing extents.



three rays

To create a ray

Menu	Command
Draw ➤ Ray	RAY

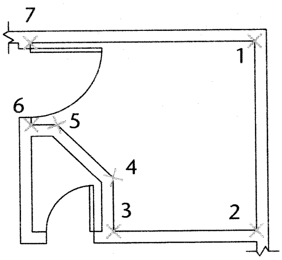
ONLINE HELP See "Creating Rays."

Calculating Areas

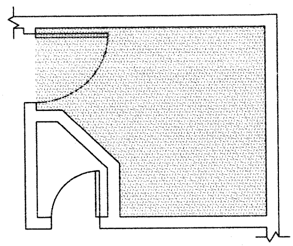
You can display the area and perimeter of several types of objects or of a sequence of points. If you need to calculate the combined area of more than one object, you can keep a running total as you add or subtract areas from the selection set.

Calculating a Defined Area

You measure an arbitrary closed region defined by the points you specify. The points must lie on a plane parallel to the *XY* plane of the current UCS.



points specified


arbitrary closed region

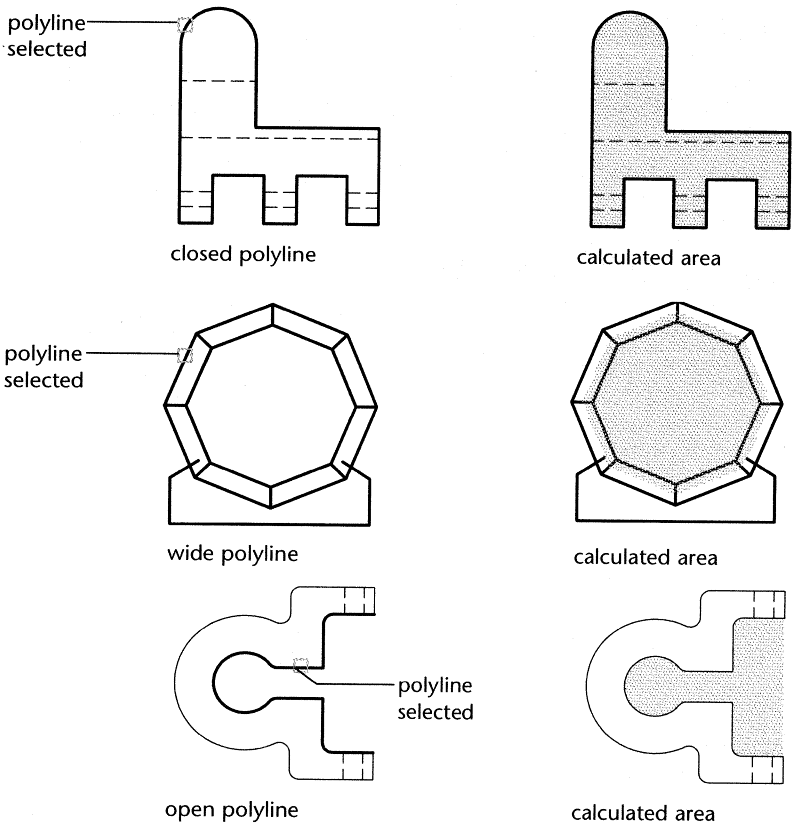
To measure an area

Standard Toolbar	Menu	Command
Inquiry Flyout	Tools ➤ Inquiry ➤ Area	AREA

Calculating the Area Enclosed by an Object

You can calculate the enclosed area and perimeter, or circumference, of circles, ellipses, polylines, and polygons. The displayed information differs according to the type of object you select:

- ***Circles, ellipses, and planar closed spline curves*** The area and circumference or perimeter are displayed.
- ***Closed polylines and polygons*** The area and perimeter are displayed. For wide polylines, this area is defined by the center of the width.
- ***Open objects, such as open spline curves and open polylines*** The area and length are displayed. The area is computed as though a straight line connected the starting point and endpoint.

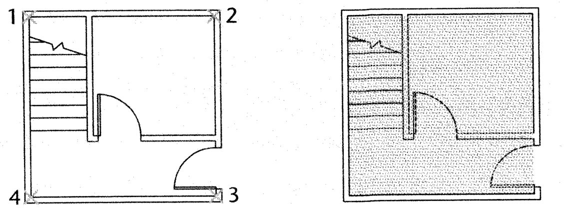


How areas are calculated for various types of objects

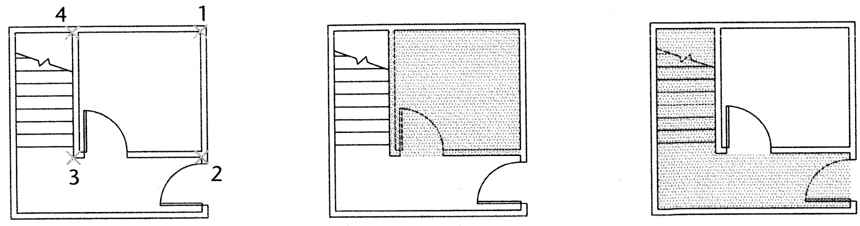
Working with Combined Areas

You can measure more than one area either by specifying points or by selecting objects. For example, you can measure the total area of rooms in a floor plan.

You can also subtract one or more areas from a combined area that you have already calculated. In the following example, the area of the floor plan is first measured, and then a room is subtracted.



original calculated area



subtracted area

total remaining area

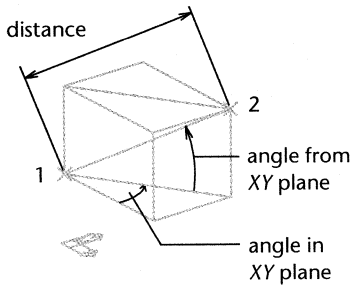
ONLINE HELP See "Calculating Areas."

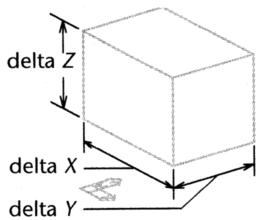
Calculating Distance and Angle

You can quickly display the following information for two points you specify:

- Distance between them in drawing units
- Angle between the points in the *XY* plane
- Angle of the points from the *XY* plane
- Delta *X*, *Y*, and *Z* distance between the points

You can use this information to quickly determine the relationship between the two points.





To calculate the distance and angle between two specified points

Standard Toolbar	Menu	Command
Inquiry Flyout	Tools ➤ Inquiry ➤ Distance	DIST

ONLINE HELP See "Calculating Distance and Angle."

Listing Object Information

You can display a variety of information regarding your drawing and the objects it contains, including:

- Database information for selected objects (LIST command)
- Coordinate values of a designated point (LIST command)
- Visual location of a point whose coordinate you specify (ID command)
- Blocks inserted and external drawings referenced (BLOCK command)
- Time spent working on the drawing (TIME command)

The following example shows the information typically listed for a circle using LIST.

```
Command: list

Select objects: 1 found

Select objects:
                  CIRCLE    Layer: 0
                            Space: Model space
                   Handle = 1E
           center point, X=   6.2174  Y=   4.9861  Z=   0.0000
                 radius    1.7247
          circumference   10.8369
                   area    9.3455
```

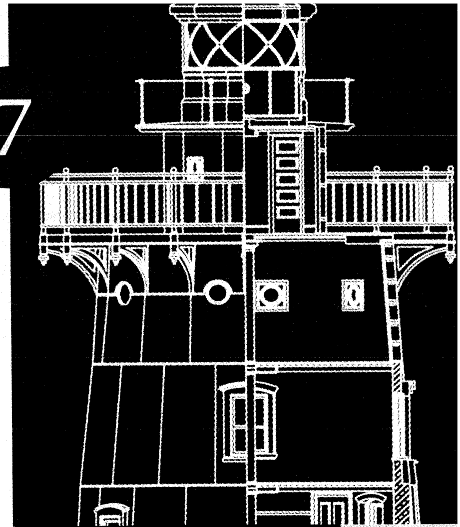
To display object information

Standard Toolbar	Menu	Command
Inquiry Flyout	Tools ➤ Inquiry ➤ List	LIST

ONLINE HELP See "Displaying the Properties of Objects."

7

Editing Your Drawing



AutoCAD LT provides powerful tools for editing objects: you can change an object's size, shape, and location, or you can convert an object into its individual components. You can also display, modify and apply properties to selected objects. This chapter describes how to edit two-dimensional (2D) objects.

In this chapter

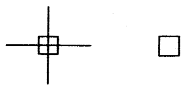
- Selecting objects individually and in groups
- Editing objects using grips
- Editing object properties
- Copying, moving, erasing, and resizing objects
- Converting objects into their components
- Modifying polylines, splines, and hatches
- Chamfering and filleting objects

Selecting Objects

Before you can edit objects, you need to create a *selection set* of the objects. A selection set can consist of a single object, or it can be a more complex grouping, for example, the set of objects of a certain color on a certain layer. You can make several changes to the same selection set, including adding or removing objects from a selection set, or selecting all objects or the most recent selection set specified.

You can create a selection set either before or after you choose the editing command.

Choosing the Command First

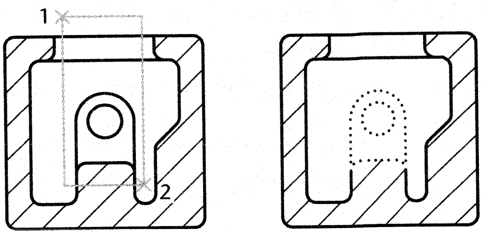


crosshairs pickbox

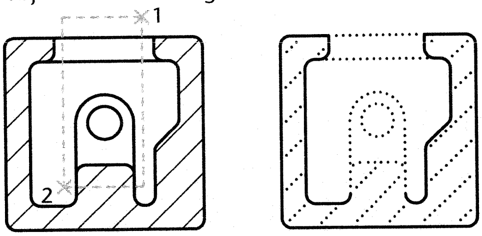
When you choose an editing command, AutoCAD LT prompts you to select objects and replaces the crosshairs with a pickbox. You can respond to the Select Objects prompt either by selecting individual objects by clicking them with the pointing device, by specifying a selection area graphically, or by entering an option at the Select Objects prompt. The most common methods are described in the following sections.

Using Selection Areas

You can select objects by enclosing them in a rectangular selection area. A selection area is a rectangular area that you define in the graphics area by specifying two corner points at the Select Objects prompt. The order in which you specify the points makes a difference. Dragging from left to right (*window* selection) selects only objects entirely within the selection area. Dragging from right to left (*crossing* selection) selects objects within and crossing the selection area. Objects must be at least partly visible to be selected.



objects selected using window selection

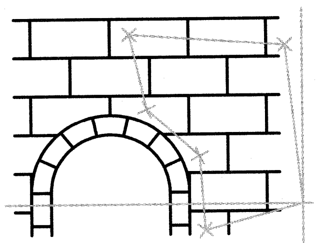


objects selected using crossing selection

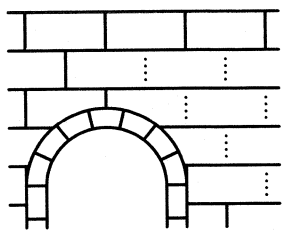
Using a Polygon Selection Area

To select objects within an irregularly shaped area, you enclose them in a polygon selection area. You create the polygon selection area by specifying points to enclose the area. You specify whether to use a window or a crossing polygon by entering **wp** or **cp** at the Select objects prompt. The direction in which you specify points to create a polygon selection area does not affect the selection method. A *window* polygon selection area selects only objects it encloses entirely, and a *crossing* polygon selection area selects objects it encloses and crosses.

The following illustration shows the result of selecting an irregular set of bricks using a window polygon selection area.

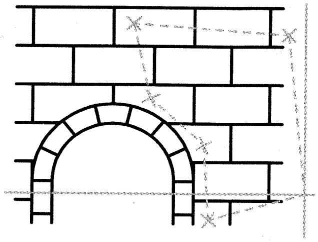


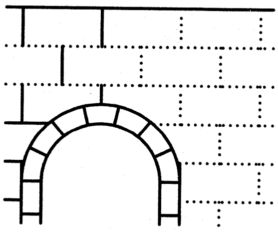
window polygon selection area



result

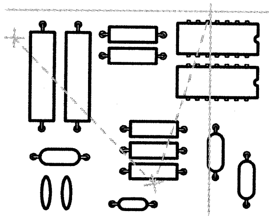
The following illustration shows the result of specifying the same selection area as a crossing polygon selection area.

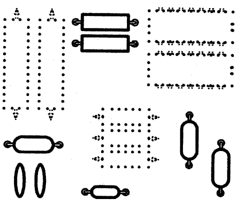

crossing polygon selection area


result

Using Selection Fences

You can select nonadjacent objects in a complex drawing most easily with a selection *fence*. A fence is a line or polyline that selects all the objects it passes through. This circuit board illustration shows a fence selecting several components.


selection fence


result

To select nonadjacent objects with a fence

1 At the Select Objects prompt, enter **f** (Fence).

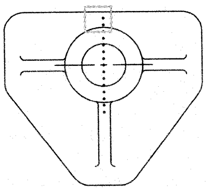
2 Specify the fence points.

3 Press ENTER to select the objects.

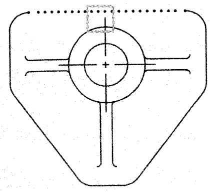
Selecting Objects Close Together

It is difficult to select objects that are close together or lie directly on top of one another. You can keep pressing the pick button to cycle through these objects, one after the other, until you have selected the one you want.

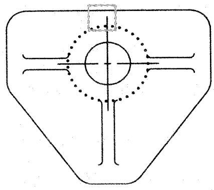
In the following example, two lines and a circle all lie within the scope of the selection pickbox.



first selected object



second selected object



third selected object

To cycle through objects for selection

1. At the Select Objects prompt, hold down CTRL and select a point as near as possible to the object you want.
2. Keep pressing the pick button of your pointing device until the object you want is highlighted.
3. Press ENTER to select the object.

Removing Objects from a Selection Set

After you create a selection set, you can remove individual objects from that set. For example, you can select an entire set of densely grouped objects and then remove specific objects within the set, leaving only the objects you want to be in the set. You can choose to remove objects only while object selection is already in progress or when objects in a selection set are highlighted.

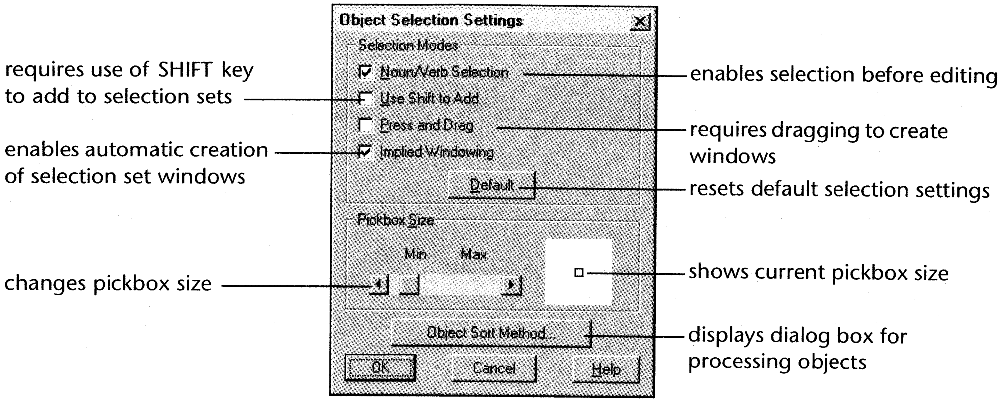
To remove objects while creating a selection set

1. Select some objects.
2. At the Select Objects prompt, enter **r** (remove).
3. At the Remove Objects prompt, select the objects you want to remove from the selection set.
4. To return to adding objects to the selection set, enter **a** (add).

NOTE You can also remove an object from a selection set by pressing the SHIFT key as you select the object.

Customizing Object Selection

You can control how objects are selected by choosing the selection setting, pickbox size, and object sorting method. Use the Object Selection Settings dialog box to control object selection.



To use the object selection settings

Menu	**Command**
Tools ➤ Selection	DDSELECT

ONLINE HELP To change selection methods and set selection system variables, see "Customizing Object Selection."

Selecting Objects First

There are several ways you can select objects before you start an editing command. You can use the SELECT command, which provides all the options you can use at the Select Objects prompt. You can also change the default setting so you always select objects before editing them.

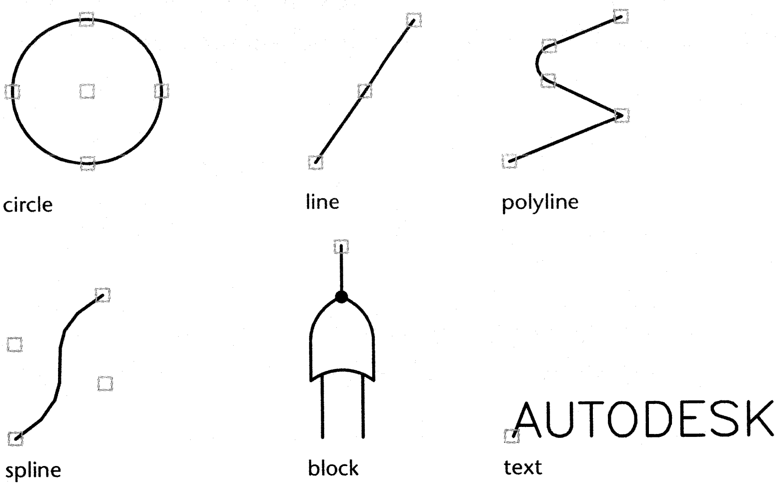
To select objects first

Menu	**Command**
Tools ➤ Selection, then turn on Noun/Verb Selection	DDSELECT

You can also select objects with the pointing device so that they are marked with grips, if grips are turned on.

Editing with Grips

Grips mark control locations on a selected object, as shown in the illustration. Selecting a block turns on a grip at its insertion point.



Examples of grip locations

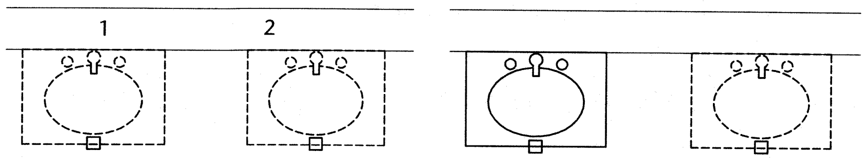
With grips you can use the pointing device to combine command and object selection and therefore edit more quickly. When grips are turned on, you select the objects you want *before* editing. You can then manipulate the objects with the pointing device and use the shortcut menu commands.

To turn grips on and off

Menu	Command
Tools ➤ Grips, then select Enable Grips	DDGRIPS

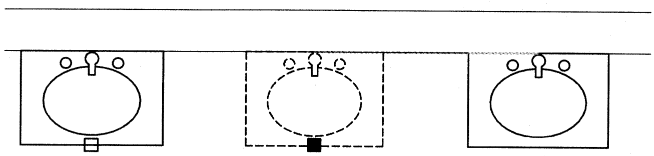
The graphics cursor snaps to any grip over which it is moved. Objects removed from a grip selection set are no longer highlighted, but their grips remain active.

Remove grips from the selection set by pressing ESC. To remove a specific object from the gripped selection set, hold down the SHIFT key as you select the object.



both blocks selected

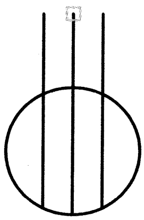
first block removed from selection set



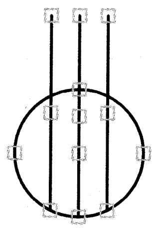
second block copied using grips

Besides turning grips on or off, you can also control the colors assigned to grips, set the grip size, and determine whether to activate all grips within a block.

The two illustrations show the difference between a block with only a single grip active and a block with the grips of all its component objects active.



grips within blocks turned off



grips within blocks turned on

To change grip settings

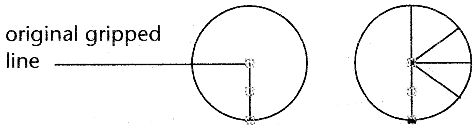
Menu	Command
Tools ➤ Grips	DDGRIPS

ONLINE HELP To control grip settings, see "Controlling Grips in Blocks."

When you use grips for editing, you select a grip to act as the base point or *base grip*. Then you select one of the grip commands. You can access these commands by right-clicking to open the shortcut menu, or you can cycle through them with the SPACEBAR or shortcut keys. For descriptions of these editing functions, see the sections on copying, offsetting, stretching, moving, rotating, scaling, and mirroring in this chapter.

Multiple Copy

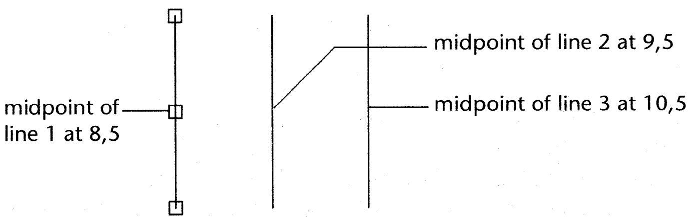
If you press SHIFT as you select the first new coordinate location, Multiple Copy is activated. For example, with Stretch, Multiple Copy stretches the object, such as a line, and copies it to wherever you specify in the graphics area. Another way to activate Multiple Copy is to specify the Copy option at the Command prompt and select a point or enter a coordinate for each copy's destination. Multiple Copy remains active until you select another option from the current grip command or press ENTER to end the command.



Stretch with multiple copy (grips and objects are not selected)

Multiple Copy Mode and Offset Snap Locations

If you hold down SHIFT continuously while you select multiple copy points, the graphics cursor snaps to offset points based on the first two points you selected. For example, in the following illustration, the midpoint of line 1 is at the coordinate 8,5. Based on that midpoint, line 2 was copied using SHIFT + Stretch; its midpoint is at 9,5. Line 3 snaps to an offset based on these two coordinates, 10,5.



Stretch with multiple copy and automatic offset snap

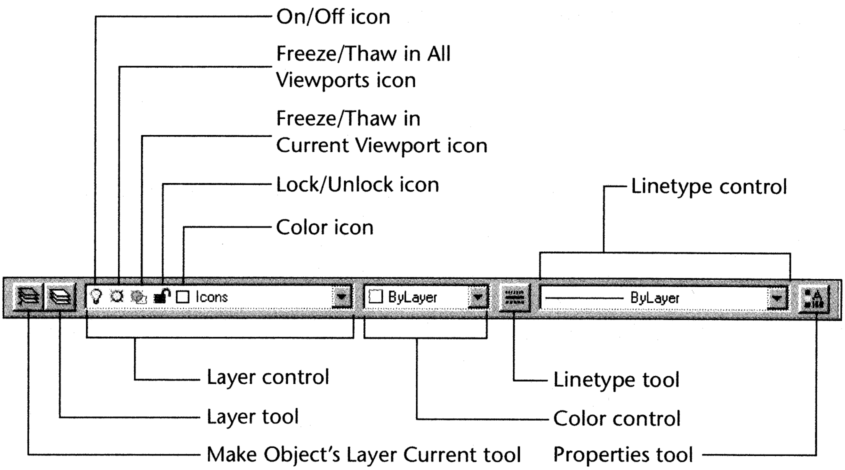
Multiple Grips as Base Grips

You can use multiple grips as the base grips. This allows you to keep the geometry intact between the selected grips. To select multiple grips, hold down the SHIFT key as you select the grips.

ONLINE HELP To edit using grips, see "Using Grips to Edit Objects."

Editing Object Properties

All the objects you draw are created on a layer. Because a layer has a linetype and color associated with it, you can specify that any object created on or moved to a layer inherits that layer's properties. An object's properties can also be set according to the "current" linetype and color settings, overriding the layer's properties, or by assigning properties to individual objects. These methods, which provide great flexibility and control of object properties, are described in detail in the following sections.



The Object Properties toolbar

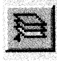
The Object Properties toolbar provides the easiest way to quickly view or change an object's layer, color, and linetype. The layer, color, and linetype controls on the Object Properties toolbar consolidate the commands needed to view and edit an object's properties. Selecting any object when no command is active dynamically displays its layer, color, and linetype in the controls on the toolbar.

Editing Layers

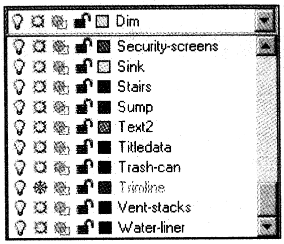
With the Layer control, you can make a layer current, view the layer of a selected object, and change an object's layer. You can also change a layer's properties by clicking the icons in the Layer control. You can make an object's layer the current layer by choosing Make Object's Layer Current.

To make an object's layer the current layer

Object Properties Toolbar



If one object is selected and no command is active, the Layer control displays the layer assigned to that object. If you've selected multiple objects on a variety of layers, the control is blank. Selecting a layer from the list changes all selected objects to that layer.



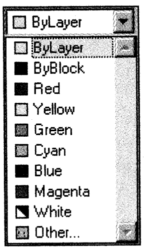
Edit a layer's properties directly in the Layer control by clicking the property icons.

The Layer control

ONLINE HELP See "Changing Layer Assignments and Layer Properties."

Editing Colors

Colors can be assigned to individual objects as well as layers in a drawing. With AutoCAD LT, the objects you create are assigned the current color. If the current color is set to BYLAYER, objects are created in the color assigned to the current layer. You can override the color of the current layer by editing an object's color or by specifying a new color for all newly created objects. Use the Color control on the Object Properties toolbar to edit the colors of individual objects.



Click Other to open the Select Color dialog box and specify a custom color.

The Color control

With the Color control, you can make a color current, and view and change the color of a selected object.

The Color control displays seven standard colors and the four most recently used colors.

ONLINE HELP To assign color to objects and layers, see "Working with Color."

Assigning Color to New Objects

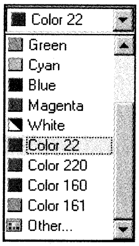
You can apply color to new objects in one of three ways.

Applying color to new objects

If you choose...	New objects assume...
a color from the Color control	the current color specified in the Color control, overriding the current layer color
BYLAYER	the color of the layer upon which they are drawn
BYBLOCK	the default color until they are grouped into a block; when the block is inserted into a drawing, it inherits the current color setting

Changing the Color of an Object

To change the color of existing objects, you first select objects whose color you want to change, and then you select a color from the Color control. If the color you want is not present in the list, you specify the color in the Select Color dialog box.



Viewing the Color Property of an Object

If one object is selected and no command is active, the Color control displays the color assigned to that object. If you select multiple objects with a variety of colors, the display window of the Color control is blank. Selecting a color from the list changes all selected objects to that color.

When you select multiple objects from different layers, and the colors of the selected objects are set to the BYLAYER option, the word BYLAYER is displayed in the Color control without a color swatch if the color assigned to each layer varies.

Editing Linetypes

Linetypes can be assigned to individual objects as well as layers in a drawing. With AutoCAD LT, the objects you create are assigned the current linetype. If the current linetype is set to BYLAYER, objects are created in the linetype assigned to the current layer. You can override the linetype of the current layer by editing an object's linetype or by specifying a new linetype for all newly created objects. Use the Linetype control on the Object Properties toolbar to edit the linetypes of individual objects.

With the Linetype control, you can make a linetype current, view and change the linetype of a selected object.

Linetypes in external references cannot be made current or assigned to objects, so their names are not displayed in the Linetype control.

ONLINE HELP For more information, see "Working with Linetypes."

Assigning Linetypes to New Objects

When you select a linetype, you can apply the linetype to new objects in one of three ways.

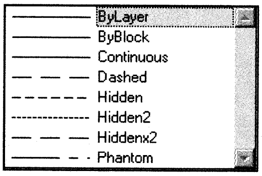
Applying linetypes to new objects

If you choose...	New objects assume...
a linetype from the Linetype control	the current linetype specified in the Linetype control, overriding the current layer linetype
BYLAYER	the linetype of the layer upon which they are drawn
BYBLOCK	the CONTINUOUS linetype until they are grouped into a block; when the block is inserted into a drawing, it inherits the current linetype setting

ONLINE HELP To set the current linetype for new objects and layers, see "Setting the Current Linetype."

Changing the Linetype of an Object

To change the linetype of existing objects, you select the objects whose linetype you want to change, and then select a linetype from the Linetype control.



The Linetype control

ONLINE HELP To change the linetype of objects and layers, see "Changing the Linetype of an Object."

Viewing the Linetype Property of an Object

If one object is selected and no command is active, the Linetype control displays the linetype assigned to that object. If you select multiple objects with a variety of linetypes, the display window of the Linetype control is blank. Selecting a linetype from the list changes all selected objects to that linetype.

When you select multiple objects from different layers, and the linetypes of the selected objects are set to the BYLAYER option, the word BYLAYER is displayed in the Linetype control. No example linetype is displayed if the linetype assigned to each layer varies.

Matching Properties of Other Objects

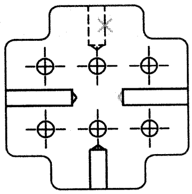
You can copy some or all properties of one object to one or more objects using the Property Painter. Properties that can be copied include color, layer, linetype, linetype scale, thickness, and in some cases, dimension, text, and hatch properties. For example, if you insert a symbol from a manufacturer's catalog, you can easily apply color, linetype, and linetype scale from another object in your current drawing, ensuring consistency of object properties.

To copy properties from an object to one or more objects

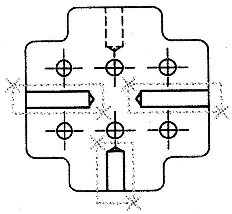
Standard Toolbar	Command
	PAINTER

1 Select the objects whose properties you want to copy.
2 Choose the Property Painter icon on the Standard toolbar, and then enter **settings** on the command line. In the Property Settings dialog box, select or clear boxes to turn properties on or off.
3 Select the destination objects and choose Apply to apply the specified properties.

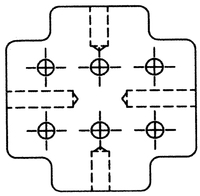
NOTE The property settings are maintained for the duration of the AutoCAD LT session or until you change them.



source object selected



destination object(s) selected



result

ONLINE HELP See "Copying Properties between Objects."

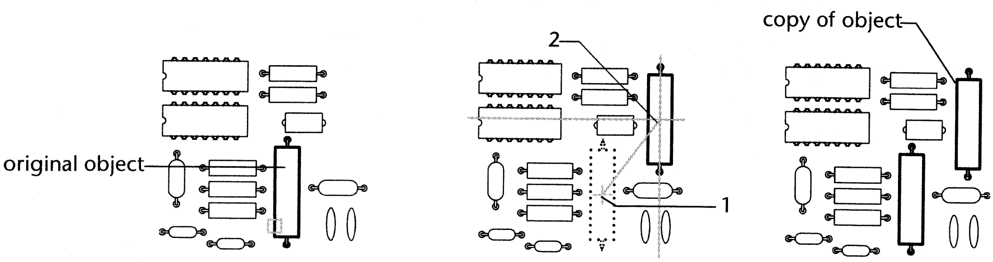
Copying Objects

With AutoCAD LT, there are several ways to copy objects. Objects can be copied to any location in a drawing, or to a location that is relative to the original object, as with mirroring, for example. *Offsetting* creates new objects at a specified distance from selected objects, or through a specified point. *Mirroring* creates a mirror image of objects in a specified mirror line. *Arraying* creates sets of copies in a rectangular or circular pattern. You can copy single or multiple objects within the current drawing, and you can copy between drawings or applications.

Copying within a Drawing

To copy objects within a drawing, you create a selection set and specify a start point and an endpoint for the copy. These points are called the base point and the second point of displacement, respectively, and can be anywhere within the drawing.

The following example illustrates the base point (1), the second point of displacement (2), and the resulting copy of the selected object.

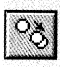


object selected   point of displacement   result

You can copy a selection set multiple times by specifying multiple points of displacement.

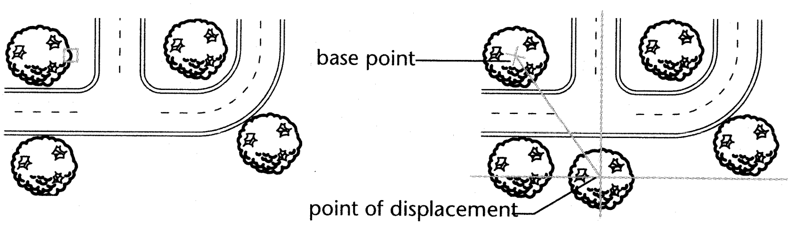
ONLINE HELP See "Copying Objects Between Points."

To copy a selection set multiple times

Modify Toolbar	Menu	Command
	Modify ➤ Copy	COPY

1 Select the objects to copy and press ENTER.

2 Enter **m** (Multiple).

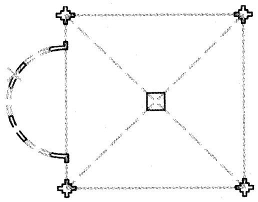


selected object   result

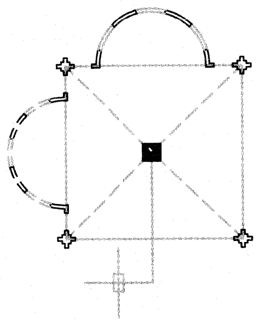
To arrange multiple copies in a rectangular or circular pattern, see "Arraying Objects" on page 167.

Multiple Copying Using Grips

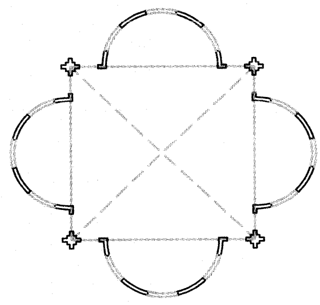
You can create multiple copies using any of the grip editing commands. For example, you can rotate a selection set, leaving copies of the set at each location you specify with the pointing device. You can also make multiple copies of selection sets, which is a quick and simple way to create small arrays.



object selected

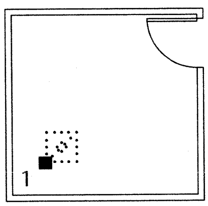


first copy rotated around base grip

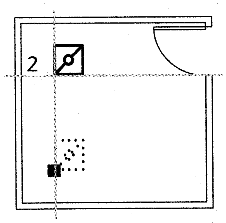


additional rotated copies

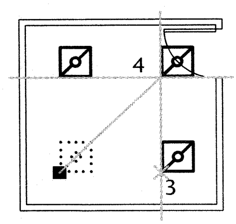
You can place multiple copies at regularly spaced intervals by creating an offset snap. The offset is defined by the distance between the original object and the first copy. In the following lighting layout, the first copy of the light fixture symbol is placed at an offset of two units. All subsequent copies are then placed two units apart.



base grip selected



copy offset defined



result

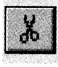
Copying with the Clipboard

When you want to use objects from another AutoCAD LT drawing or from a file created with another application, move or copy these objects to the Clipboard and then paste them from the Clipboard into your drawing. Copying to and pasting from the Clipboard is different from copying objects from one location to another within a single drawing.

Cutting to the Clipboard

Cutting deletes the selected objects from the drawing and stores them on the Clipboard.

To cut a selected object to the Clipboard

Standard Toolbar	Menu	Command
	Edit ➤ Cut	CUTCLIP

Related Select the objects and press CTRL + X.

Copying to the Clipboard

Copying to the Clipboard retains a copy of the selected objects in the drawing and also stores them on the Clipboard. The objects can then be pasted into another drawing or into another application.

NOTE Updating the original object does *not* update the inserted copy. Compare *copying* with *linking*, which is discussed in the next section, "Copying Views." With AutoCAD LT, you can also use Microsoft's OLE (object linking and embedding).

ONLINE HELP For information on using object linking and embedding (OLE), see "Linking and Embedding Data (OLE)."

To copy a selected object to the Clipboard

Standard Toolbar	Menu	Command
	Edit ➤ Copy	COPYCLIP

Related Select the objects and press CTRL + C.

ONLINE HELP To cut or copy objects to the Clipboard and then paste from the Clipboard, see "Using Cut, Copy, and Paste."

Copying Views

The COPYLINK command copies the current view rather than selected objects to the Clipboard. The view can then be pasted into a document within another application. Unlike embedding, linking creates a reference to the source. If the source view is updated, you have to update only the link to update all linked copies of the view.

If you select a viewport, AutoCAD LT copies the viewport contents. Otherwise, it copies the displayed graphics area.

To link an AutoCAD LT view to another document

Menu	Command
Edit ➤ Copy Link	COPYLINK

1 Save the AutoCAD LT drawing you want to link first, so that it has a name.
2 If multiple viewports are displayed, select a viewport.
3 From the Edit menu, choose Copy Link.
   AutoCAD LT copies the view to the Clipboard.
4 Open a document in another application. You can then paste the Clipboard contents, following the application's procedures for inserting linked data so that it can be edited from AutoCAD LT through the application.

ONLINE HELP To copy and link views, see "Exporting OLE Objects from AutoCAD LT."

Pasting Objects into AutoCAD LT

Applications use different internal formats to store information. When you copy objects to the Clipboard, AutoCAD LT stores information in all available formats. When you paste the Clipboard contents into an AutoCAD LT drawing, AutoCAD LT uses the format that retains the most information. However, you can override this setting and convert pasted information to AutoCAD LT format.

Pasting AutoCAD LT Objects

The AutoCAD LT format is the preferred format for copying objects to and from AutoCAD LT, because it is the easiest format to edit. It retains all relevant object information, including block references and 3D aspects. You can also paste AutoCAD objects into AutoCAD LT without losing any object information.

Pasting Text

ASCII text (in TXT files) is copied to AutoCAD LT as paragraph, or multiline, text. Paragraph breaks are retained, and the text can be edited using the multiline text editor. To retain the font and style characteristics of formatted text, specify the Windows metafile (WMF) format.

Pasting Metafiles

The Windows metafile format, or picture format, contains screen vector information and can be scaled and printed without loss of resolution. Use this format to paste objects into Windows applications that support WMF files. You can convert metafiles to AutoCAD LT objects using the IMPORT command.

Pasting Bitmaps

Bitmapped images are raster images consisting of a pattern of pixels and are commonly used by paint applications. Bitmaps may lose clarity when scaled, so this format is pasted only if AutoCAD LT cannot correctly handle the format in another way. AutoCAD LT recognizes the standard Windows device-independent bitmap format (BMP files).

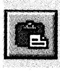
Pasting Text to the Command Line

You can paste single-line or multiline text from the Clipboard to the command line. For example, you might copy and paste a long command or text string to avoid entering it again.

Editing Pasted Information

If you convert pasted information to AutoCAD LT format, the pasted information becomes a true AutoCAD LT object and can be edited as such. The same is true for unformatted text and objects already in AutoCAD LT format. You cannot use all of AutoCAD LT's editing commands when modifying information in other formats.

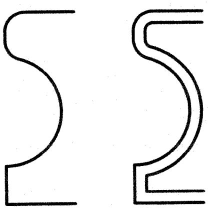
To paste objects from the Clipboard

Standard Toolbar	Menu	Command
	Edit ➤ Paste	PASTECLIP

Related Press CTRL + V to paste.

Offsetting Objects

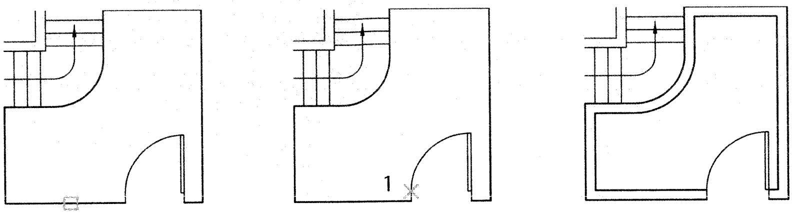
Offsetting creates a new object that is similar to a selected object but at a specified distance. You can offset lines, arcs, circles, 2D polylines, ellipses, elliptical arcs, xlines, rays, and planar splines. Offsetting circles creates larger or smaller circles depending on the offset side.



original object

object with offset

The following example shows how the polyline border of the room is offset so that the wall aligns with the door frame. You select the polyline border, and then specify the offset point (1).



selected object — through point — result

For information about offsetting splines, see "Editing Splines" on page 183.

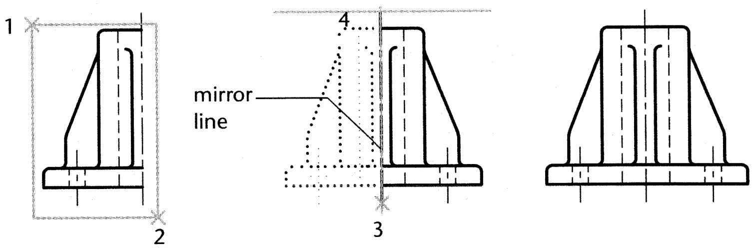
To offset an object

Modify Toolbar	Menu	Command
	Modify ➤ Offset	OFFSET

ONLINE HELP See "Offsetting an Object."

Mirroring Objects

You mirror objects around a *mirror line*, which you define with two points.



object selected with window — mirror line defined with two points — result with original object retained

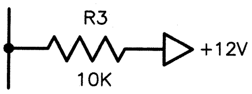
You can delete or retain the original objects. Mirroring works in any plane parallel to the *XY* plane of the current UCS.

Mirroring is useful for creating symmetrical objects. You can quickly draw half the object and mirror it rather than drawing the whole object.

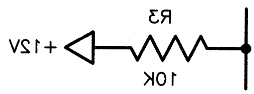
Mirroring Text and Attributes

When you mirror text, attributes, and attribute definitions, they are reversed or turned upside down in the mirrored image. These mirrored objects are true mirror images of the original section of the object and follow the mathematical rules for reflection.

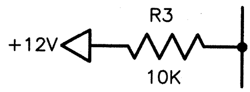
To prevent mirrored text from being reversed or turned upside down, you set the MIRRTEXT system variable to 0, or off. By default, MIRRTEXT is on. If you turn it off, the text has the same alignment and justification as before the mirroring. Compare the following illustrations.



before MIRROR

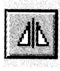


after MIRROR
(MIRRTEXT = 1)



after MIRROR
(MIRRTEXT = 0)

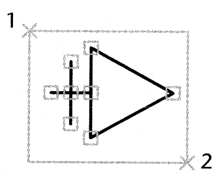
To mirror an object

Modify Toolbar	Menu	Command
	Modify ➤ Mirror	MIRROR

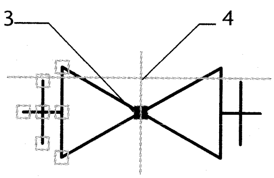
Mirroring with Grips

One quick way to mirror is with grips. The following example shows how you draw a valve by mirroring one half of the valve and retaining the original. Turning Ortho on helps you draw a vertical mirror line so that the object is mirrored horizontally.

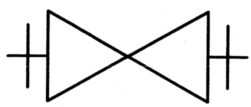
You select the objects to mirror using a window, (1) and (2), and then select the base grip (3). Choose Mirror from the shortcut menu, enter **c** on the command line to copy the mirrored object, and then specify the mirror line (4).



object to mirror selected with window selection



base grip and second point selected with Ortho on for mirror line



result with original object retained

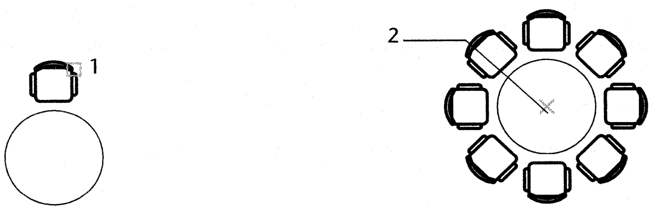
ONLINE HELP See "Mirroring Objects."

Arraying Objects

An *array* is a set of identical objects in a rectangular or circular pattern. You can copy an object or selection set in polar or rectangular arrays. For polar arrays, you control the number of copies of the object and whether the copies are rotated. For rectangular arrays, you control the number of rows and columns and the distance between them.

Creating Polar Arrays

The following example shows a circular table with chairs created by making a polar array of the original chair and rotating the copies as they are arrayed. The illustration shows the selected object (1) and the center of the polar array (2). For polar arrays, you specify the center, the number of objects in the array, and the angle the array is to fill.

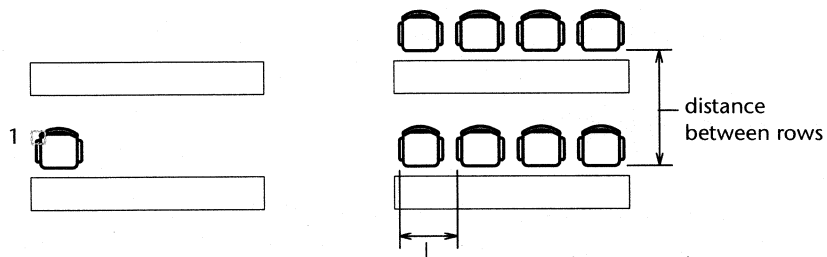


object selected

results of polar array of 8 objects in 360°

Creating Rectangular Arrays

This example shows a rectangular array of the chair.



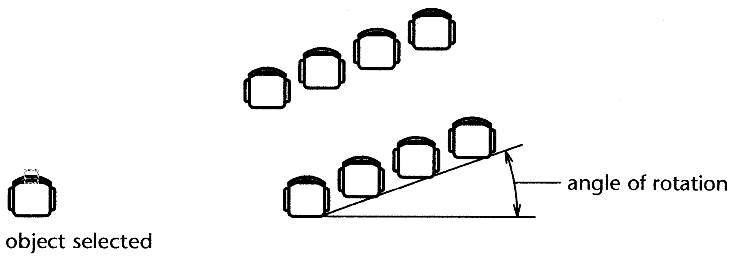
object selected

The array has two rows and four columns. The illustration shows the object selected (1) and the resulting array. For rectangular arrays, you specify the number of rows and columns and the distance between the rows and columns. The direction of the array can be controlled by specifying positive or negative values for distances.

Creating Rotated Rectangular Arrays

AutoCAD LT builds a rectangular array along a baseline defined by the current snap rotation angle. This angle is zero by default, so the rows and columns of a rectangular array are orthogonal with respect to the *X* and *Y* drawing axes. However, you can change this angle and create a rotated array. Setting the snap rotation angle to a nonzero value rotates the screen crosshairs accordingly. All rectangular arrays are constructed parallel to the crosshairs; rows align with the *X* crosshair, and columns align with the *Y* crosshair.

The following example shows a rotated rectangular array of chairs.



For rotated rectangular arrays, you set the *snap angle* to the angle of rotation for the array.

To create an array

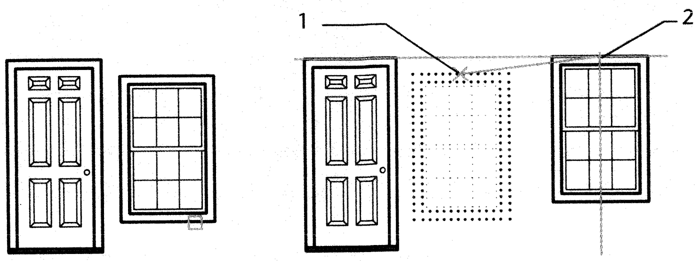
Modify Toolbar	Menu	Command
	Modify ➤ Array	ARRAY

ONLINE HELP See "Creating an Array of Objects."

Moving Objects

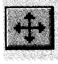
When you move objects, you can rotate or move them without changing orientation or size. You use snap, coordinates, grips, and object snaps to move objects with precision.

The following example shows how you move the window. You select the object to move, specify the base point (1), and then specify the point of displacement (2).



object selected   result

To move an object

Modify Toolbar	Menu	Command
	Modify ➤ Move	MOVE

You can also move objects by stretching them. See "Moving by Stretching" on page 172.

Moving with Grips

One quick way to move objects is with grips. You select the objects to move, and then select the base grip. Choose Move from the shortcut menu, and then move the selected objects using the pointing device.

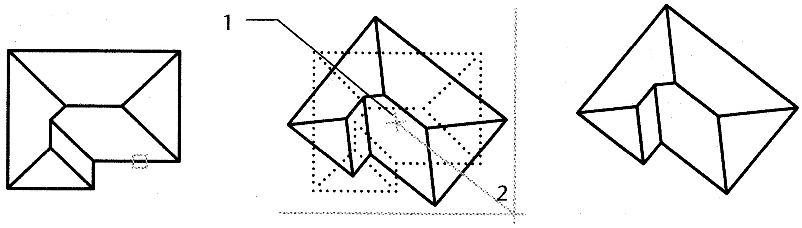
ONLINE HELP See "Moving Objects."

Rotating Objects

You rotate objects by choosing a base point and a *relative* or *absolute* rotation angle. You can control whether objects are rotated counterclockwise or clockwise.

Rotating by Relative Angle

The following example shows how you rotate the plan view of a house using the default relative angle method. The illustration shows the base point (1) and the angle of rotation (2).



object selected

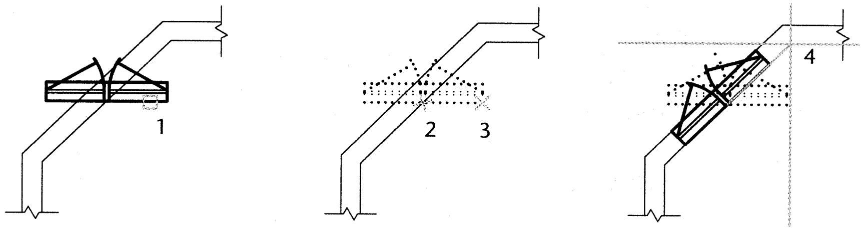
base point and angle of rotation

result

Rotating by Reference Angle

Sometimes it's easier to rotate with absolute angles. For example, to align two objects when you know the absolute angles of both, use the current angle of the object to be rotated as the reference angle, and use the angle of the other object as the new angle. An easier way is to use the pointing device to select the object that you want to rotate and the object you want to align it with.

The following example shows how you specify the reference angle by selecting two points on the object to rotate. You then specify the new angle by selecting the object you want to align with.



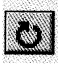
object selected

reference angle specified

new angle specified—object rotates

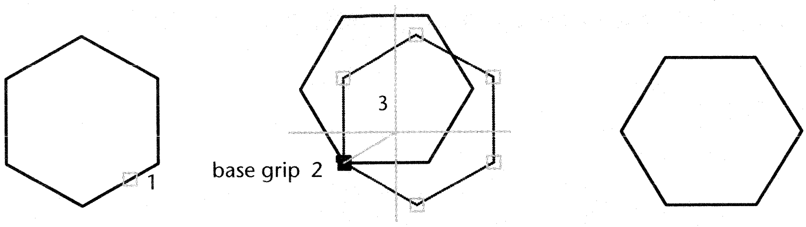
The illustration shows the selected object (1), the intersection point (2) and the object endpoint (3) to define the reference angle, and the new angle specified (4) to complete the rotation.

To rotate an object

Modify Toolbar	Menu	Command
	Modify ➤ Rotate	ROTATE

Rotating with Grips

You can use grips to rotate objects around a base grip in either direction of rotation, specifying the angle of rotation with your cursor or entering an angle. The following example shows how you rotate a polygon using grips.



object selected | polygon rotated counter-clockwise | result

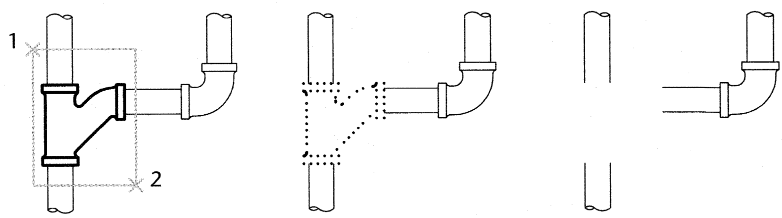
Select the object to rotate (1), and then highlight the base grip (2). Choose Rotate from the shortcut menu, and then use your pointing device and rotate the object to a new position (3), which results in a counterclockwise rotation.

ONLINE HELP See "Rotating Objects."

Erasing Objects

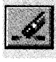
You delete unwanted objects from your drawing by erasing them. You can use all the available selection methods with the ERASE command (see "Selecting Objects" on page 146).

The following example shows how you use window selection to erase a section of piping.



objects selected with window selection | selected objects | result

To erase an object

Modify Toolbar	Menu	Command
	Modify ➤ Erase	ERASE

ONLINE HELP See "Erasing Objects."

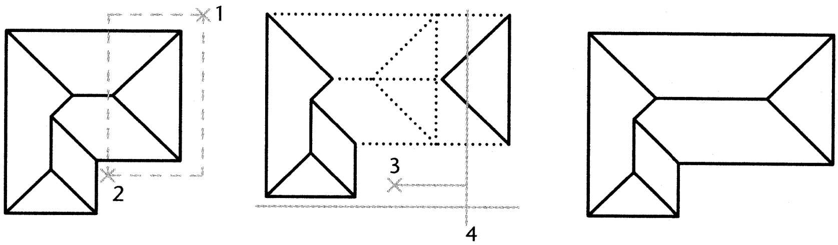
Resizing Objects

You can resize objects by stretching, scaling, extending, lengthening, and trimming them.

Stretching Objects

To stretch an object, you specify a base point for the stretch and then a point of displacement. You select the object with a crossing selection. You can also combine grip editing with object snaps, grid snaps, and relative coordinate entry to stretch with greater accuracy.

This example shows how you stretch the plan view of a house. Turning on Ortho helps you move the object in a straight line.

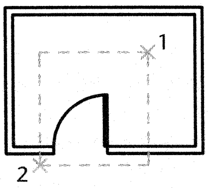


objects selected with crossing selection | points specified for stretch with Ortho on | result

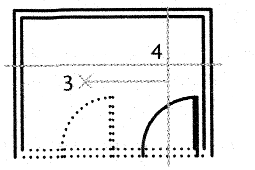
The illustration shows the crossing selection (1, 2), the base point (3), and the point of displacement (4).

Moving by Stretching

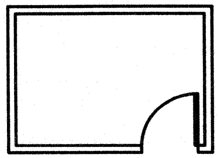
The following example shows how you move a door from one part of a wall to another by stretching.



objects selected with
crossing selection



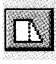
objects dragged with
Ortho on



result

The illustration shows the crossing selection (1, 2), the base point (3), and the point of displacement (4). The door and the door endpoint lie entirely within the selection window and thus move to the new location. The wall lines, on the other hand, merely cross the selection window and stretch in accordance with the movement of the door.

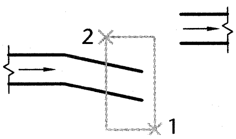
To stretch an object

Modify Toolbar	Menu	Command
	Modify ➤ Stretch	STRETCH

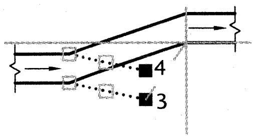
Stretching with Grips

You can stretch an object by moving selected grips to new locations. Some grips move the object rather than stretching it. This is true of grips on text objects, blocks, midpoints of lines, centers of circles, centers of ellipses, and point objects.

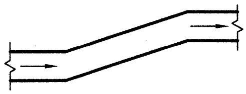
The following example shows how you stretch two lines that represent a pipe.



object selected with
crossing selection



two base grips selected
and dragged



result

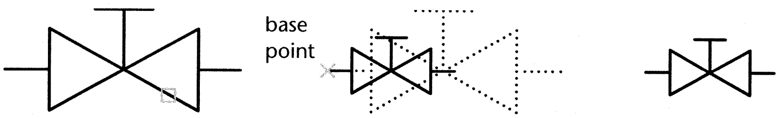
The illustration shows the crossing selection (1, 2), the end grips highlighted (3, 4), and the result of the stretch.

Scaling Objects

You scale selection sets using the same scaling factor in the *X* and *Y* directions. You scale selection sets using the same scaling factors in the *X* and *Y* directions. Scaling makes an object larger or smaller, but does not alter its aspect ratio. You can scale an object in several ways.

Scaling by Scale Factor

Scaling by a scale factor changes all dimensions of the selected object. A scale factor greater than 1 enlarges the object. A scale factor less than 1 shrinks the object. In the following example, you decrease the size of the block by half, scaling it by a factor of 0.5.

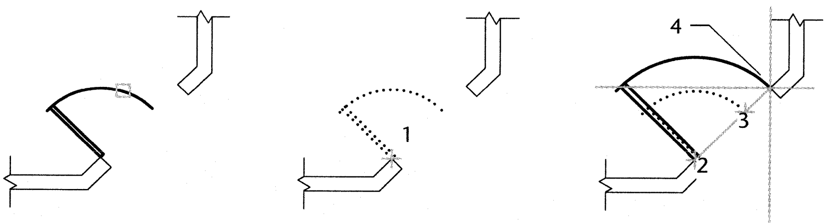


object selected | object scaled by factor of 0.5 | result

Scaling by Reference

When you scale by reference, you use the size of an existing object as a reference for the new size. To scale by reference, specify the current scale and then the new scale length. For example, if one side of an object is 4.8 units long and you want to expand it to 7.5 units, use 4.8 as the reference length and 7.5 as the new length. A quick way to change the length of open objects, such as lines, arcs, polylines, elliptical arcs, and splines, is to lengthen them. See "Changing the Length of Objects" on page 180.

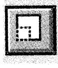
You can also specify the reference length by selecting a base point (1) and two reference points (2, 3) and dragging to specify the new scale (4).



object selected | base point specified | first and second reference points and reference length

You can use the Reference option to scale an entire drawing. For example, use this option when the original drawing units are inappropriate. Select all objects in the drawing. Then use Reference to select two points and specify the intended distance. All the objects in the drawing are scaled accordingly.

To scale an object

Modify Toolbar	Menu	Command
	Modify ➤ Scale	SCALE

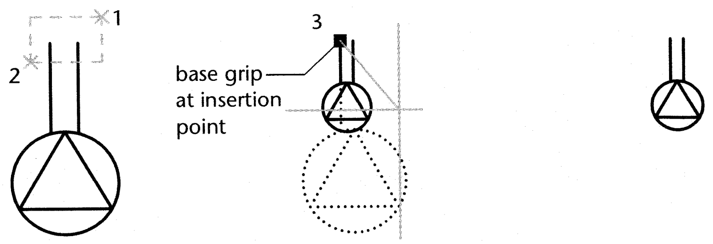
1 Select the objects to scale, and then specify the base point.

2 To scale the selected objects, do *one* of the following:

- Move your pointing device toward or away from the base point to scale the selected object.
- Enter the scale factor.
- Enter **r** (Reference). Select the first and second reference points, or enter a value for the reference length. Drag the object to scale and select a point, or enter a value for the new length.

Scaling with Grips

You can use the Scale grip command to scale objects. For example, you can increase the size of a circle by dragging outward from the base grip or decrease the size by dragging inward. Alternatively, you could enter a value for relative scaling. In this example, the outlet symbol, which is defined as a block, is scaled down. When selected, blocks have a single grip at the insertion point.



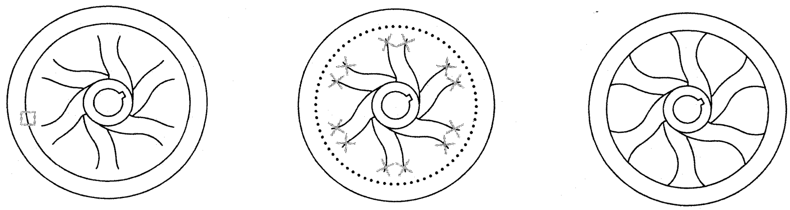
object selected with crossing selection | cursor moved to reduce size | result

To scale the block, you select it using a crossing selection (1, 2), and then select the insertion point as the base grip (3). Choose Scale from the shortcut menu, and then move the cursor toward the center of the block, thus decreasing its size.

ONLINE HELP To stretch and scale objects, see "Resizing or Reshaping Objects."

Extending Objects

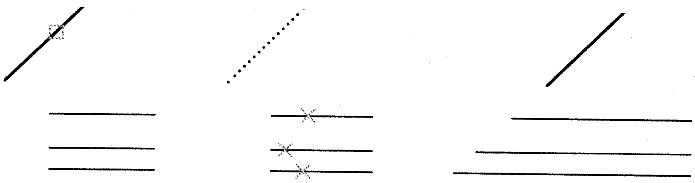
You can extend objects so that they end precisely at a boundary defined by other objects. You can also extend objects to where they *would* intersect a boundary. This is called extending to an *implied* boundary. This example shows how you extend the lines precisely to a circle, which is the boundary.



boundary selected — objects to extend selected — result

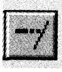
Extending to an Implied Boundary

This example shows how you extend the three horizontal lines to an implied boundary, which is where they would intersect the single line if it were extended.



implied boundary selected — objects to extend selected — result

To extend an object

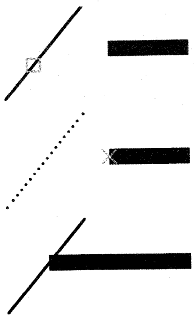
Modify Toolbar	Menu	Command
	Modify ➤ Extend	EXTEND

To extend to an implied boundary

1 From the Modify menu, choose Extend.
2 Select the object for the implied boundary and press ENTER.
3 Enter **e** (Edge).
4 Enter **e** (Extend), or press ENTER if Extend is the current option.
5 Select the lines to be extended and press ENTER.

If you want to extend an object without selecting objects as defined boundary edges, press ENTER to select all objects as potential boundaries.

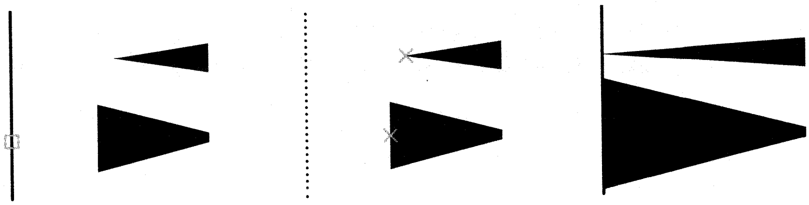
Extending Polylines



extending a wide polyline

You can extend only open polylines. Either the first or the last edge extends as if it were a line or arc object.

Wide polylines extend so the centerline intersects the boundary. Because the ends of wide polylines are at a 90-degree angle, part of the end extends past the boundary if the boundary is not perpendicular to the extended segment. If you extend a tapered polyline segment, the width of the extended end is corrected to continue the original taper to the new endpoint. If this correction gives the segment a negative ending width, the ending width is forced to 0.



boundary edge selected — objects to extend selected — result

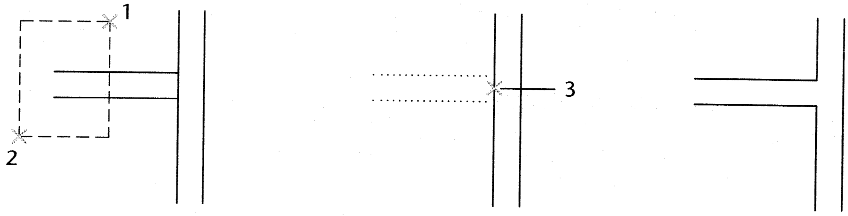
Extending Infinite Lines

You can extend rays, but you *cannot* extend an xline. Like a circle, an xline is unbounded and has no start point and no endpoint. A ray is semibounded and has a start point but no endpoint; therefore, a ray can be extended to a new start point.

Trimming Objects

You can cut an object precisely at an edge defined by one or more objects. Objects you define as cutting edges do not have to intersect the object being trimmed; you can trim back to an implied intersection. Cutting edges can be lines, arcs, circles, polylines, ellipses, splines, xlines, rays, and viewports in paper space. Wide polylines are cut along their centerline.

The following example shows how you join two walls smoothly by trimming the section where they intersect.

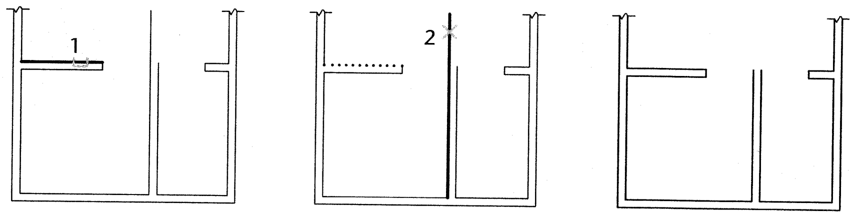


cutting edges selected with a crossing selection | object to trim selected | result

Trimming to an Implied Intersection

An implied intersection is the point where two objects would intersect if they were extended. You can trim objects using their implied intersection as the cutting edge.

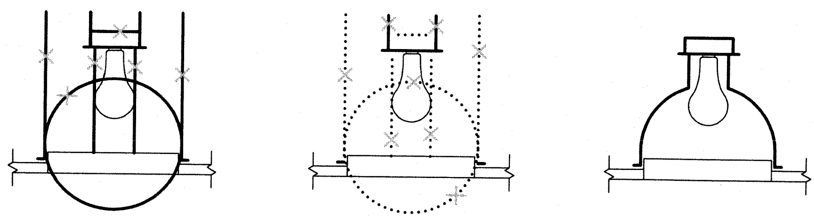
The following example shows how you trim the vertical wall back to its implied intersection with the horizontal wall.



cutting edge selected | object to trim selected | result

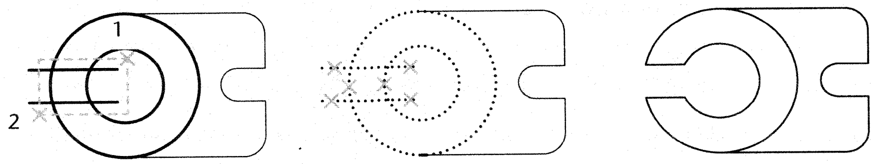
Trimming Complex Objects

An object can be one of the cutting edges *and* one of the objects being trimmed. For example, in the light fixture illustrated, the circle is a cutting edge for the construction lines and is also being trimmed.



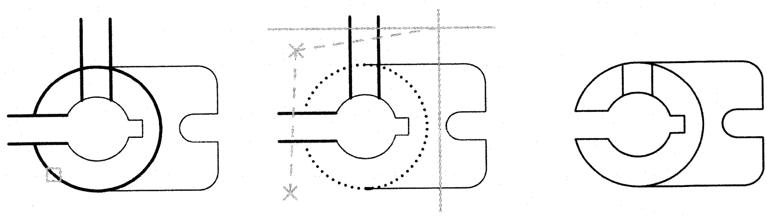
cutting edges selected | objects to trim selected | result

When trimming complex objects, using different selection methods can help you choose the right cutting edges and objects to trim. In the following example, the cutting edges are selected with crossing selection.



edges selected with crossing selection | objects to trim selected | result

In the following example, a selection fence is used to select a series of objects for trimming.



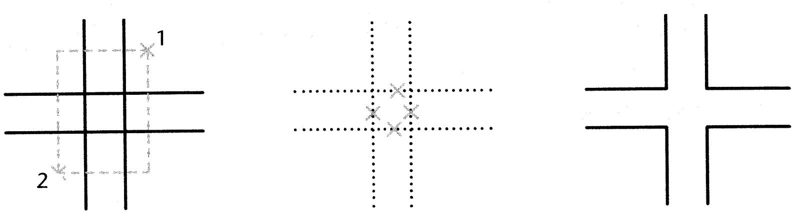
cutting edge selected | objects to trim selected with a selection fence | result

In paper space, you can use viewport borders as cutting edges; however, you cannot trim the viewports themselves.

If you want to trim an object without selecting objects as defined cutting edges, press ENTER to select all objects as potential cutting edges.

Trimming to the Nearest Intersection

You can trim objects to their nearest intersection with other objects. When you select the objects to trim, AutoCAD LT automatically chooses the nearest selected objects as cutting edges. In this example, the walls are trimmed so that they intersect smoothly.



cutting edges selected with crossing selection | objects to trim selected | result

To trim an object

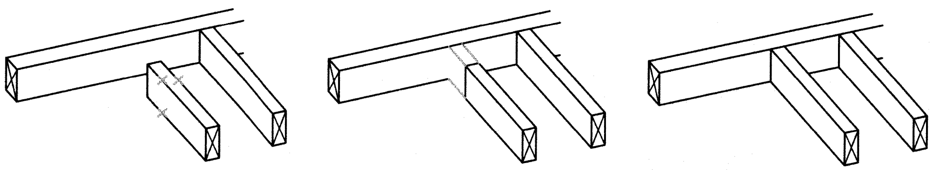
Modify Toolbar	Menu	Command
	Modify ➤ Trim	TRIM

ONLINE HELP To trim and extend objects, see "Trimming and Extending Objects."

Changing the Length of Objects

You can change the angle of arcs, and you can change the length of open lines, arcs, open polylines, elliptical arcs, and open splines. The results are similar to both extending and trimming. You can alter length in several ways:

- Specifying an incremental length or angle measured from an object's endpoint (Delta)
- Specifying a new length as a percentage of the total length or angle (Percent)
- Specifying the object's total absolute length or included angle (Total)
- Dragging an object's endpoint (Dynamic)



original line object — lengthening by percentage — result after moving edge lines and trimming

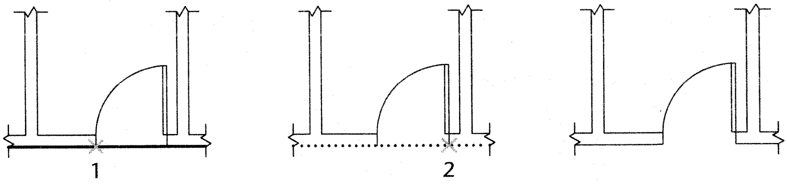
To lengthen an object

Modify Toolbar	Menu	Command
	Modify ➤ Lengthen	LENGTHEN

ONLINE HELP To lengthen objects, see "Changing the Size and Shape of Objects."

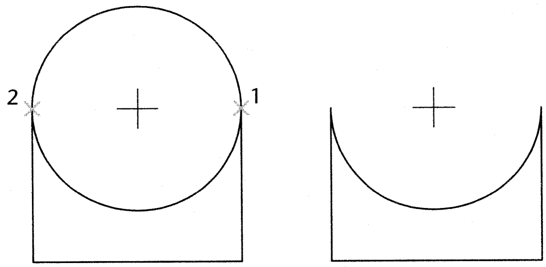
Breaking Objects

You can remove part of an object with the BREAK command. You can break lines, circles, arcs, polylines, ellipses, splines, xlines, and rays. When breaking an object, you can either select the object at the first break point and then specify a second break point, or you can select the entire object and then specify the two break points.

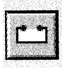


first break point   second break point   result

Breaks in enclosed objects such as circles, ellipses, polygons, and polylines always remove the object part in a counterclockwise direction.



To break an object

Modify Toolbar	Menu	Command
	Modify ➤ Break	BREAK

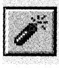
ONLINE HELP See "Creating Breaks in Objects."

Exploding Objects

Exploding an object converts it from a single object to its constituent parts. For example, exploding forms simple lines and arcs from polylines, rectangles, donuts, and polygons. It replaces a block reference or associative dimension with copies of the simple objects that compose the block or dimension.

An exploded object doesn't look any different, but exploding objects or blocks can modify the properties or attributes associated with them.

To explode an object

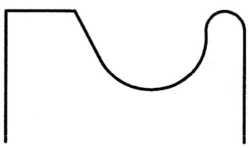
Modify Toolbar	Menu	Command
	Modify ➤ Explode	EXPLODE

ONLINE HELP See "Exploding Objects."

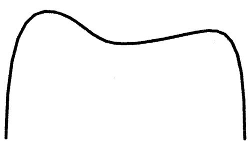
Editing Polylines

You can edit polylines by closing and opening them and by moving, adding, or deleting individual vertices. You can straighten the polyline between any two vertices and toggle the linetype so that it applies to each segment or to the polyline as a whole. You can set a uniform width for the entire polyline or control the width of each segment.

You can create a linear approximation of a spline curve from a polyline with the Fit and Spline options and then use Decurve to change the splined polyline back to a polyline. The illustration shows a polyline edited with the Spline option.



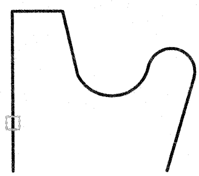
polyline



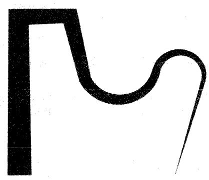
splined polyline

However, some editing actions, such as trimming, breaking, and any grip editing, remove the spline definition of a fit or splined polyline. For this reason, it is better to use a true spline, which maintains its spline definition. You create a true spline using SPLINE. See "Drawing Spline Curves" on page 107.

In this illustration, each polyline segment has a different starting and ending width, resulting in a taper.

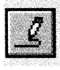


polyline selected



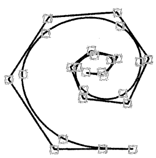
segments with different start and end widths

To edit a polyline

Modify II Toolbar	Menu	Command
	Modify ➤ Object ➤ Polyline	PEDIT

ONLINE HELP See "Editing Polylines."

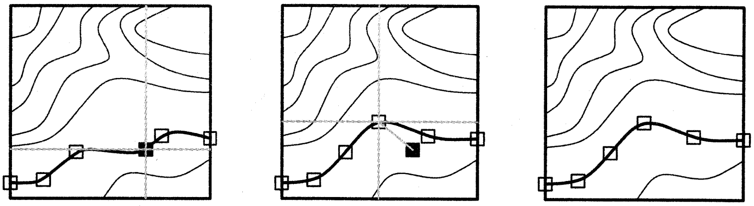
Editing Splines



Spline curves are defined by the *fit points* used to create them. You can modify a spline by adjusting its fit points: delete them, add more for greater accuracy, or move them to alter the shape of the spline. You can open or close a spline, edit its start and end tangents, or reverse its direction. You also can change the tolerance of the spline. Tolerance refers to how closely the spline fits the set of fit points you specify. The lower the tolerance, the more closely the spline fits the points.

Splines can be further refined by using the *control points*, or frame, of a curve. The more control points you specify in a particular part of the frame, the more pull they exert on the curve. Increasing the weight of a control point pulls the spline more towards that point. You can also refine a spline by changing its order. A spline's order is the degree of the spline polynomial plus 1. A cubic spline, for example, has order 4. The higher a spline's order, the more control points it has. This type of curve is called a *B-spline*. AutoCAD LT can generate quadratic and cubic spline-fit polylines.

The following example shows a spline that represents a geographic contour. You can use grips to move and refit the fourth or fifth fit point to increase the accuracy of the spline.



fit point selected fit point moved result

To edit a spline curve by moving fit points

Modify II Toolbar	Menu	Command
	Modify ➤ Object ➤ Spline	SPLINEDIT

ONLINE HELP See "Editing Splines."

Editing Hatches

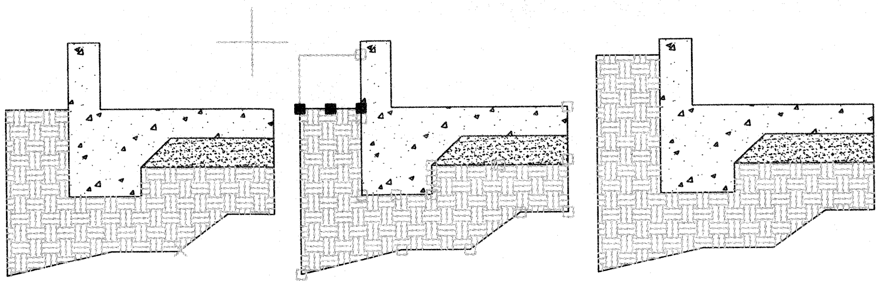
You can edit both hatch boundaries and hatch patterns. You can remove the associativity of a hatch, modify hatch patterns, or choose a new pattern for an existing hatch.

Editing Hatch Boundaries

Hatch boundaries can be copied, moved, stretched, and so on, using the editing commands described earlier in this chapter. If you edit the boundary of an associative hatch, the pattern is updated as long as the editing results in a valid boundary. Associative hatches are updated even if they're on layers that are turned off. Nonassociative hatches are not updated, even when their boundaries are changed. To change a nonassociative hatch, you need to recreate it.

Editing Hatches with Grips

Using grips, you can stretch, move, rotate, scale, and mirror hatch boundaries and their associated hatches just as you can other objects. If the editing results in a valid boundary, the associative hatch is updated. See "Hatching Areas" on page 110.

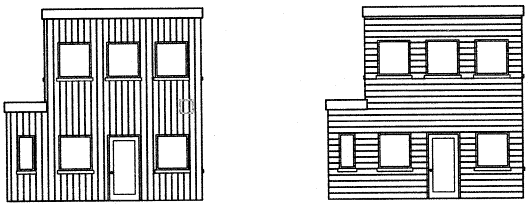


associative hatch using EARTH pattern | hatch boundary to move selected | result

Changing Existing Hatch Patterns

You can change the angle or spacing of an existing hatch pattern or replace it with a solid fill or a predefined pattern.

The following example shows the results of changing the angle of the hatch.

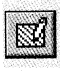


line hatch pattern before editing | line hatch pattern after editing

After editing, the hatch pattern is updated with a new angle.

To reduce file size, a hatch pattern is defined in the drawing as a single graphic object. However, you can also explode hatch patterns into their composite lines. Hatch patterns are defined in text files with the *.pat* file extension. By altering these files, or creating new ones, you can create new hatch patterns.

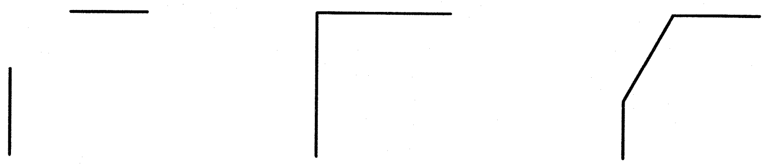
To edit existing hatch patterns

Modify Toolbar	Menu	Command
	Modify ➤ Object ➤ Hatch	HATCHEDIT

ONLINE HELP See "Modifying Hatches and Solid-Filled Areas."

Chamfering Objects

Chamfering connects two nonparallel objects by extending or trimming them to intersect or to join with a beveled line. You can chamfer lines, polylines, xlines, and rays.



original objects

chamfer distance zero

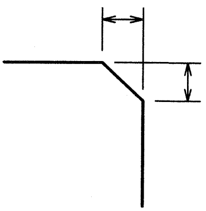
chamfer distance nonzero

If both objects being chamfered are on the same layer, the chamfer line is drawn on that layer. Otherwise, the chamfer line is drawn on the current layer. The same rules apply to chamfer color and linetype.

If no intersection point is within the drawing limits and if limits checking is turned on, AutoCAD LT rejects the chamfering. For information about drawing limits, see "Setting Grid Limits" on page 72.

With the *distance method*, you specify the amount that each line should be trimmed or extended. With the *angle method*, you also specify the length of the chamfer and the angle it forms with the first line. You can retain the objects as they were before the chamfer or trim or extend them to the chamfer line.

Chamfering by Specifying Distances

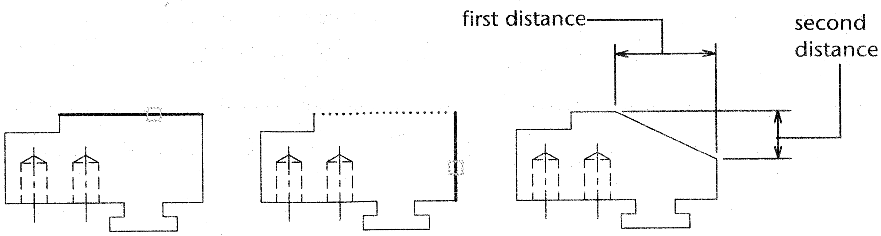


chamfer distances

The chamfer distance is the amount each object is trimmed or extended to meet the chamfer line or to intersect the other object. If both chamfer distances are 0, chamfering trims or extends the two objects until they meet but does not draw a chamfer line.

The default setting for the first distance is the last distance specified. The default setting for the second distance is whatever you choose for the first distance, because symmetrical distances are common. However, you can reset the chamfer distances.

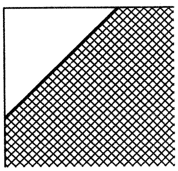
The following example shows the result of setting uneven chamfer distances.



first line selected second line selected result

Trimming Chamfered Objects

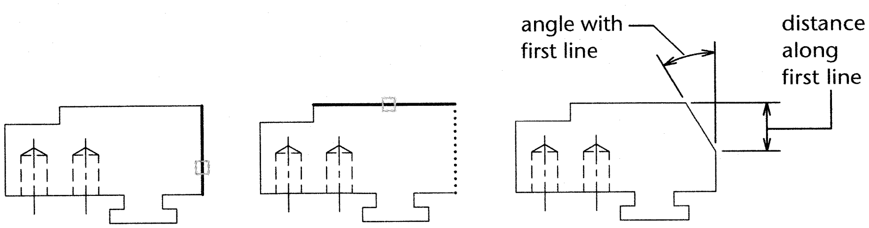
By default, objects are trimmed when chamfered, but you can specify that they remain untrimmed. The following illustration shows an untrimmed chamfer.



untrimmed chamfered objects

Chamfering by Specifying Distance and Angle

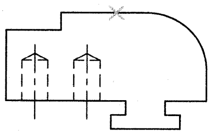
You can chamfer two objects by specifying where on the first selected object the chamfer line starts. Then you specify the angle the chamfer line forms with this object. The following example shows the chamfer resulting from specifying a distance and angle.

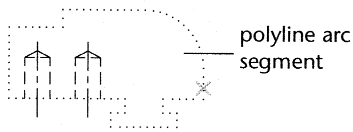


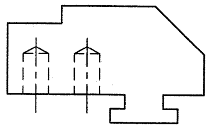
first line selected second line selected result

Chamfering Polyline Segments

If the two points you select are segments of a polyline, they must be adjacent or separated by one arc segment. If they are separated by an arc segment, as shown in the illustration, chamfering deletes the arc and replaces it with a chamfer line.





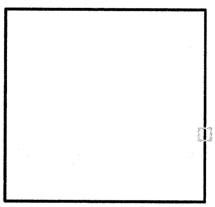


polyline with arc segment selected

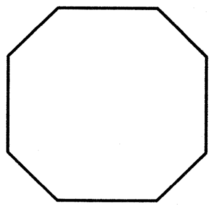
result—chamfer line replaces polyline arc

Chamfering an Entire Polyline

You can chamfer an entire polyline so that each intersection is chamfered. For best results, keep the chamfer distances equal. This example shows the result of chamfering a closed polyline by setting both chamfer distances to the same values.

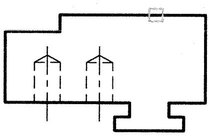


polyline selected when chamfer distances equal

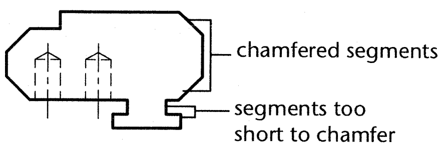


result

When you chamfer an entire polyline, AutoCAD LT chamfers only the segments that are long enough to accommodate the chamfer distance. In the following illustration, some polyline segments were too short to be chamfered.



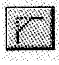
polyline selected



result

Chamfers added to a polyline become new segments of that polyline, even if the chamfer distance is 0.

To chamfer two nonparallel lines by specifying the distance

Modify Toolbar	Menu	Command
	Modify ➤ Chamfer	CHAMFER

1 Enter the first chamfer distance. Enter the second chamfer distance.

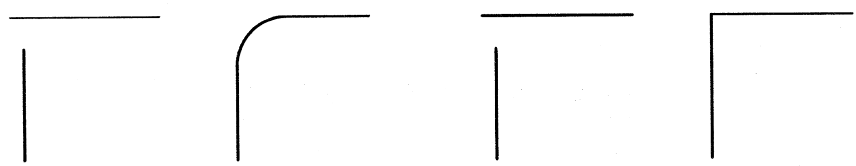
2 Press ENTER to reenter the CHAMFER command.

3 Select the lines for chamfering.

ONLINE HELP To chamfer by specifying distances, chamfer by specifying distance and angle, chamfer an entire polyline, or trim chamfered objects, see "Creating Chamfers."

Filleting Objects

Filleting connects two objects with a smoothly fitted arc of a specified radius. Although an inside corner is called a *fillet* and an outside corner is called a *round,* AutoCAD LT treats both as fillets.



two lines filleted with nonzero radius — two lines filleted with zero radius

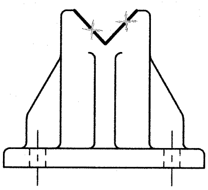
If both objects being filleted are on the same layer, the fillet line goes on that layer. Otherwise, the fillet line goes on the current layer. The same rules apply to fillet color and linetype.

You can fillet pairs of line segments, polyline line (not arc) segments, splines, xlines, rays, circles, arcs, and true (not polygon) ellipses. Lines, xlines, and rays can be filleted when parallel. You can fillet every vertex of a polyline at the same time. You can fillet a combination of lines and polylines.

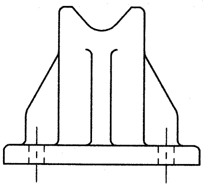
Setting the Fillet Radius

The fillet radius is the radius of the arc that connects filleted objects. By default, the fillet radius is 0.5 or the last radius set. Changing the radius affects subsequent fillets but not existing ones.

The following example shows the result when you fillet two line segments.



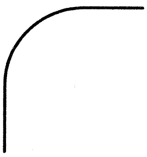
lines selected



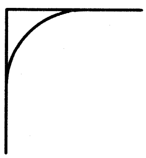
result

Trimming Filleted Objects

By default, all objects except circles, full ellipses, closed polylines, and splines are trimmed when filleted. You can specify that filleted objects remain untrimmed.



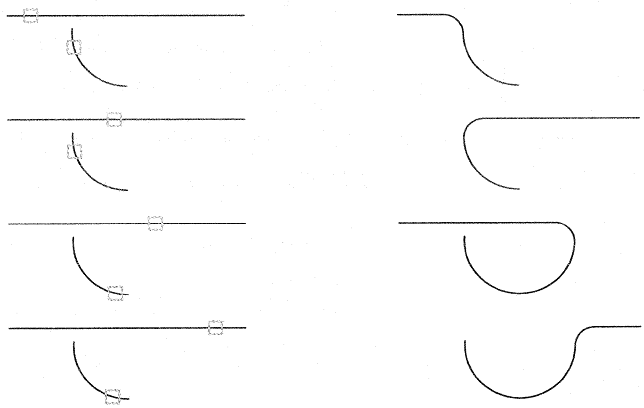
two lines filleted with the Trim option set



two lines filleted with the No Trim option set

Filleting Circles and Arcs

You fillet circles and arcs in the same way as lines. Depending on the points you specify, more than one possible fillet can exist between the objects. AutoCAD LT chooses the endpoints closest to the points you use to select the objects. Compare the following fillets:



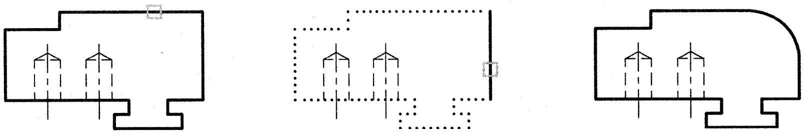
objects selected for filleting

results

Filleting Line and Polyline Combinations

To fillet line and polyline combinations, the line must intersect, or intersect when extended, one of the polyline's line segments. If the Trim option is turned on, the filleted objects and fillet arc join to form a single new polyline.

The illustration shows a fillet of a line and polyline with a fillet radius of 0.25 and the Trim option on.



polyline selected

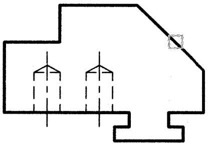
line selected

result

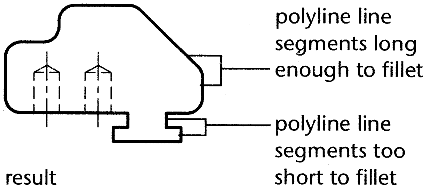
Filleting an Entire Polyline

You can fillet an entire polyline or remove fillets from a polyline. The polyline can contain arc segments that have the same fillet radius as line segments in the polyline.

If you set a nonzero fillet radius, AutoCAD LT inserts fillet arcs at each vertex where two line segments meet if the segments are long enough to accommodate the fillet radius.

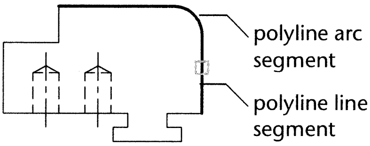


polyline selected for filleting

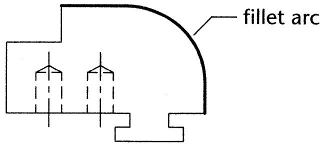


result

If two polyline line segments are separated by one arc segment and the two line segments converge as they approach the arc segment, AutoCAD LT removes the arc segment and replaces it with a fillet arc.

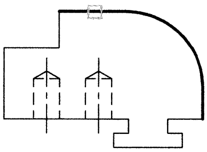


polyline selected for filleting

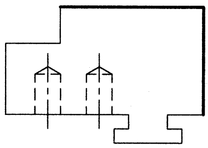


result—fillet arc replaces arc segment

If you set the fillet radius to 0, no fillet arcs are inserted. If two polyline line segments are separated by one arc segment, AutoCAD LT removes that arc and extends the lines until they intersect.



polyline selected for filleting

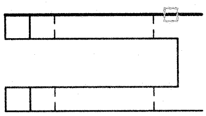


result—zero radius fillet replaces arc segment

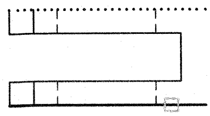
This method provides a quick way to join two endpoints. However, if the lines do *not* converge as they approach the arc segment, no change occurs.

Filleting Parallel Lines

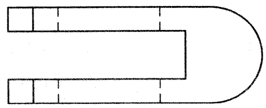
You can fillet parallel lines, xlines, and rays. Because two parallel lines uniquely determine a plane, the fillet arc is placed on that plane. The first selected object must be a line or ray, but the second object can be a line, xline, or ray. The fillet arc connects as shown here.



first parallel line selected



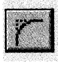
second parallel line selected



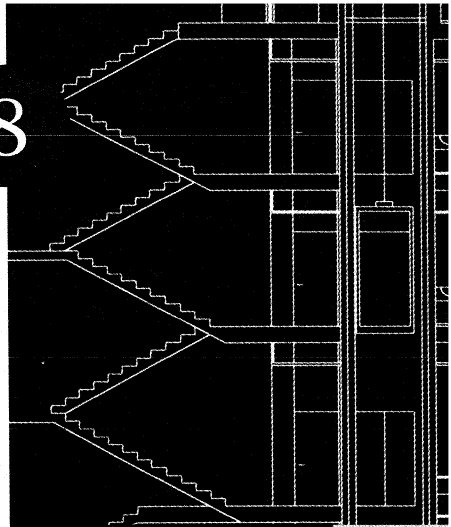
result

The diameter of the fillet arc is always equal to the distance between the lines. The current fillet radius is ignored and remains unchanged.

To fillet or round the edges of two selected objects

Modify Toolbar	Menu	Command
	Modify ➤ Fillet	FILLET

ONLINE HELP To set the filleting radius, trim filleted objects, fillet arcs and circles, fillet line and polyline combinations, fillet an entire polyline, and fillet parallel lines, see "Creating Fillets."



8

Controlling the Drawing Display

In this chapter

- Using Realtime to change the view area and magnification
- Using aerial views to zoom and pan the drawing
- Using tiled viewports
- Using named views

In online Help

- You can also specify a viewpoint, or location in 3D model space from which you view a 3D drawing. See "Changing 3D Viewpoints."

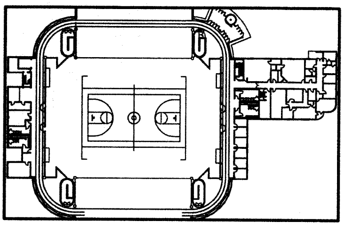
As you edit your drawing, you can control the drawing display to move quickly to different areas of your drawing while you track the overall effect of your changes. You can zoom to change magnification or pan to reposition the view in the graphics area; display several views at one time by splitting the screen into several tiled viewports; or save a view and then restore it.

The complexity of your drawing affects how fast AutoCAD LT refreshes the screen. You can improve program performance by turning off the display of text, fill, or selection highlighting.

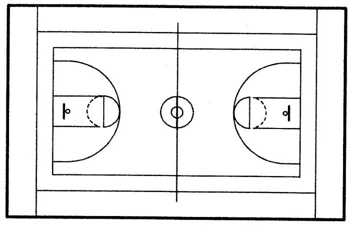
Using Zoom and Pan

A specific magnification, position, and orientation is known as a *view*. The most common way to change a view is to use one of AutoCAD LT's many *zoom* options, which increase or decrease the size of the image displayed in the graphics area.

Shrinking the image to see a larger portion of the drawing is called zooming out. Magnifying the image to view the details more closely is called zooming in.



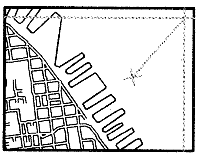
zoomed out



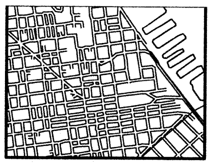
zoomed in

Zooming does not change the absolute size of the drawing. It changes the size of the view within the graphics area. AutoCAD LT offers several ways to change the view, including specifying a display window, zooming to a specific scale, and displaying the entire drawing.

You can move the current view of the drawing in any direction by *panning*. The drawing limits and the scale and zoom value of your view remain the same. Only the position changes.



before PAN



after PAN

You can also change the view by dragging the scroll bars in the graphics area, even while you are using a command.

Zooming and Panning Interactively

Along with the ability to pan and zoom the image incrementally, AutoCAD LT provides the Realtime option for interactive zooming and panning. With ZOOM Realtime, you can zoom in or out of the drawing by moving the cursor vertically up or down.

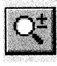
Using PAN Realtime, you can pan the drawing image to a new location by clicking the image with the pointing device and moving the cursor.

When you are using the Realtime option of ZOOM or PAN, you can right-click the pointing device and use the shortcut menu to move quickly between zooming and panning.

Using Real-Time Zooming

Realtime ZOOM provides interactive zooming capability. As you move the pointing device up or down, the drawing image zooms in or out.

To zoom in real time

Standard Toolbar	Menu	Command
	View ➤ Zoom ➤ Realtime	ZOOM

The Realtime option is the default setting for ZOOM. Pressing ENTER after entering **zoom** on the command line automatically starts ZOOM Realtime.

With ZOOM Realtime, you can zoom in or out of the drawing by moving the cursor vertically up or down. Hold down the pick button at the midpoint of the drawing and move the cursor vertically to the top (positive direction) of the window to zoom in to 100% (2x magnification). Hold down the pick button at the midpoint of the drawing and move the cursor vertically to the bottom (negative direction) of the window to zoom out 100% (.5x magnification).

When you have reached the zoom-in limit (the current view), the plus sign (+) is no longer displayed, indicating that you cannot zoom in any further. When you have zoomed out to the limit (the extents of the current view), the minus sign (–) is no longer displayed, indicating that you cannot zoom out any further. You cannot zoom out beyond the extents of the current view.

When you release the pick button, zooming stops. You can release the pick button, move the cursor to another location in the drawing, and then press the pick button again and continue zooming from that location.

By right-clicking, you can open a shortcut menu with additional viewing options, including Pan, Zoom Window, Zoom Previous, and Zoom Extents. To exit ZOOM Realtime, press ENTER or ESC.

NOTE If you are using a Microsoft IntelliMouse with ZOOM Realtime, you can zoom in by pushing the top of the wheel forward. To zoom out, pull the top of the wheel backward. You can set the IntelliMouse zoom factor on the System tab of the Preferences dialog box.

Using Real-Time Panning

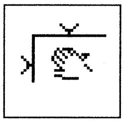
PAN Realtime provides interactive panning capability. As you move the pointing device, the drawing image shifts to a new location.

To pan in real time

Standard Toolbar	Menu	Command
	View ➤ Pan ➤ Realtime	PAN

Realtime is the default setting for PAN. Pressing ENTER after entering **pan** on the command line automatically starts PAN Realtime.

Hold down the pick button on the pointing device and move the hand cursor to pan the drawing.



When you reach a logical extent (edge of the drawing space), a bar is displayed on the hand cursor on the side where the extent has been reached. Depending upon whether the logical extent is at the top, bottom, or side of the drawing, the bar is either horizontal (top or bottom) or vertical (left or right side).

When you release the pick button, panning stops. You can release the pick button, move the cursor to another location in the drawing, and then press the pick button again to pan the display from that location.

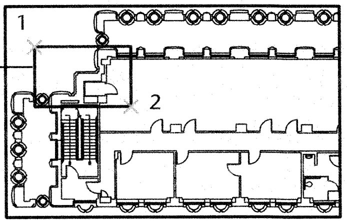
By right-clicking, you can open a shortcut menu with additional viewing options, including Zoom, Zoom Window, Zoom Previous, and Zoom Extents. To exit PAN Realtime, press ENTER or ESC.

NOTE If you are using a Microsoft IntelliMouse with PAN Realtime, you can pan by holding down the wheel and dragging the mouse. To stop panning, release the wheel. Use the MBUTTONPAN system variable to turn on pan.

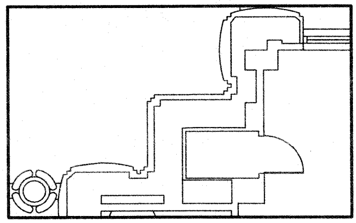
ONLINE HELP See "Changing Views."

Defining a Zoom Window

You can quickly zoom in on an area by specifying the corners that define it.



original view



new view

To specify a zoom window

Standard Toolbar	Menu	Command
Zoom Flyout	View ➤ Zoom ➤ Window	ZOOM Window

- Click to specify the corners of the area you want to zoom in on.

The region specified by the corners you select is centered in the new display. The shape of the zoom window you specify does not necessarily correspond to the new view, which fits the shape of the graphics area.

Displaying the Previous View

When you work with close-up details in your drawing, you may need to zoom out frequently to see an overview of your work. Use ZOOM Previous to return quickly to the prior view.

AutoCAD LT can restore up to 10 previous views in succession. These views include not only zoomed views but also views that have been panned, restored, or set to perspective or plan view. ZOOM Previous restores only the view magnification and position, not the previous content of an edited drawing.

To restore the previous view

Standard Toolbar	Menu	Command
	View ➤ Zoom ➤ Previous	ZOOM Previous

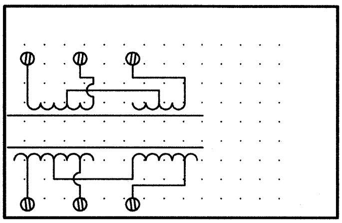
Shortcut From ZOOM Realtime, right-click, then choose Zoom Previous from the shortcut menu.

Scaling a View

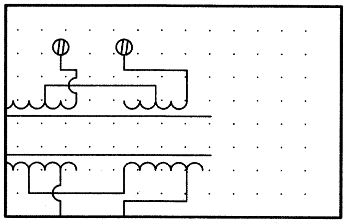
If you need to increase or decrease the magnification of the image by a precise scale, you can specify a zoom *scale* in three ways:

- Relative to the drawing limits
- Relative to the current view
- Relative to paper space units

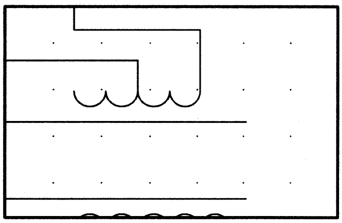
To scale a view relative to the drawing limits, enter a simple scale value. For example, enter **1** to display the limits as large as possible in the graphics area, centered at the center point of the previous view. To zoom in or out, enter a higher or lower number. As shown in the following example, you can enter **2** to display the image at twice the full size or **.5** to display the image at half the full size. The limits are shown by the grid.



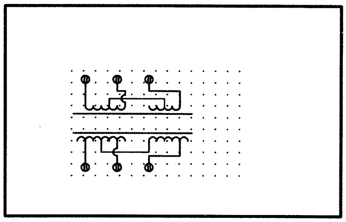
current view



zoomed to 1



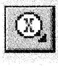
zoomed to 2



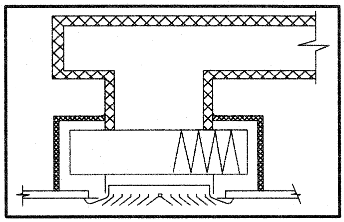
zoomed to .5

Scaling the view relative to the drawing limits

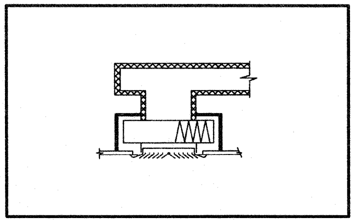
To zoom to a given scale factor

Standard Toolbar	Menu	Command
	View ➤ Zoom ➤ Scale	ZOOM Scale

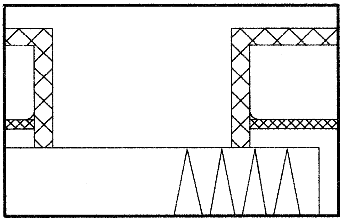
To scale a view relative to the current view, add **x** after the value you enter. As shown in the following example, you can enter **2x** to double the size of the current view or **.5x** to display a view half the size of the current view. Entering **1x** has no effect.



current view



zoomed to .5x



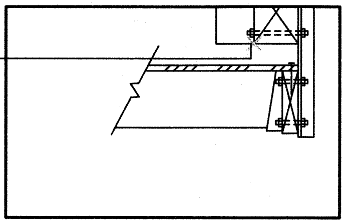
zoomed to 2x

Scaling the view relative to the current view

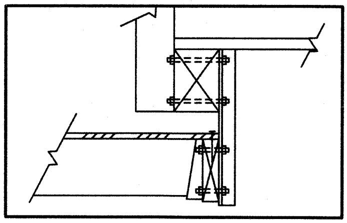
To scale a view relative to paper space units, add **xp** after the value you enter. This specification increases or decreases the view relative to the current paper space scale and is used to scale the viewport before plotting. See "Scaling Views Relative to Paper Space" on page 340.

Centering a View

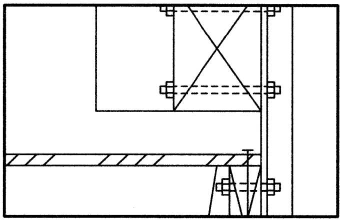
You can move a specific point in your drawing to the center of the graphics area. ZOOM Center is useful for resizing an object and bringing it to the center of the viewport. The following example shows the effects of using ZOOM Center to display a view at the same size and at twice the size.



current view



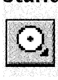
ZOOM Center with size unchanged



ZOOM Center using 2x scale

Centering a specific point

To zoom to a given center and magnification

Standard Toolbar	Menu	Command
 Zoom Flyout	View ➤ Zoom ➤ Center	ZOOM Center

With ZOOM Center, you can specify size by entering either the number of vertical drawing units or a magnification relative to the current view.

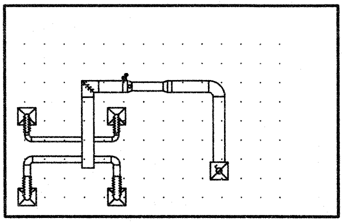
For example, to specify a height, enter **2** to display a view that is two drawing units high. Values smaller than the default value increase the size of the image. Larger values decrease the size of the image.

To specify relative magnification, enter a scale factor followed by **x**. For example, enter **2x** to display a view that is twice as large as the current view. If you are working with floating viewports, you can enter **xp** to scale the view relative to paper space. See "Scaling Views Relative to Paper Space" on page 340.

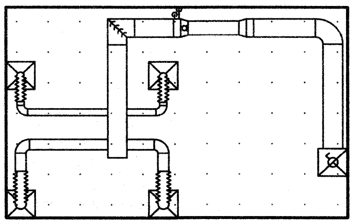
Displaying Drawing Limits and Extents

To display a view based on the drawing boundaries or the extents of the objects in the drawing, use ZOOM All or ZOOM Extents.

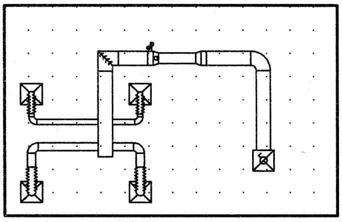
ZOOM All displays the entire drawing. If the objects extend beyond the limits, ZOOM All displays the extents of the objects. If the objects are drawn within the limits, ZOOM All displays the limits. The following illustrations show ZOOM All and ZOOM Extents when the objects are drawn within the limits.



current view



zoomed to extents



zoomed to show entire drawing (all)

Zoomed views with grid defining drawing limits

ZOOM Extents changes the view to encompass the object extents for the current drawing. In some cases, using ZOOM All and Extents may cause a regeneration. If the REGENAUTO system variable is on, the regeneration occurs automatically. If REGENAUTO is off, you are prompted to approve the regeneration. Regeneration will not occur on layers that are frozen. If your drawing has no objects, ZOOM Extents displays the drawing limits.

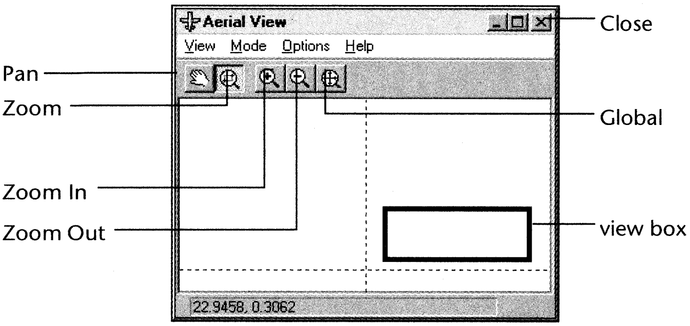
ZOOM Extents calculates zooms based on the extents of the active viewport, not the graphics area. However, when using the ZOOM command in model space while working in a floating viewport, if you are zoomed in beyond the floating viewport's borders, some of the area zoomed may not be visible.

To display the entire drawing or the extents

Standard Toolbar	Menu	Command
Zoom Flyout	View ➤ Zoom ➤ All View ➤ Zoom ➤ Extents	ZOOM All ZOOM Extents

Using Aerial View

Aerial View is a navigation tool that displays a view of the drawing in a separate window so that you can quickly move to that area. If you keep the Aerial View window open as you work, you can zoom and pan without choosing a menu option or entering a command.



The Aerial View window provides real-time zooming and panning of the AutoCAD LT window. Using the Aerial View window to zoom into and view a portion of the drawing does not force regeneration of the drawing.

Aerial View works in all model space and paper space views. You can easily move the Aerial View window by dragging it to another location. Also, you can resize the window by dragging its border.

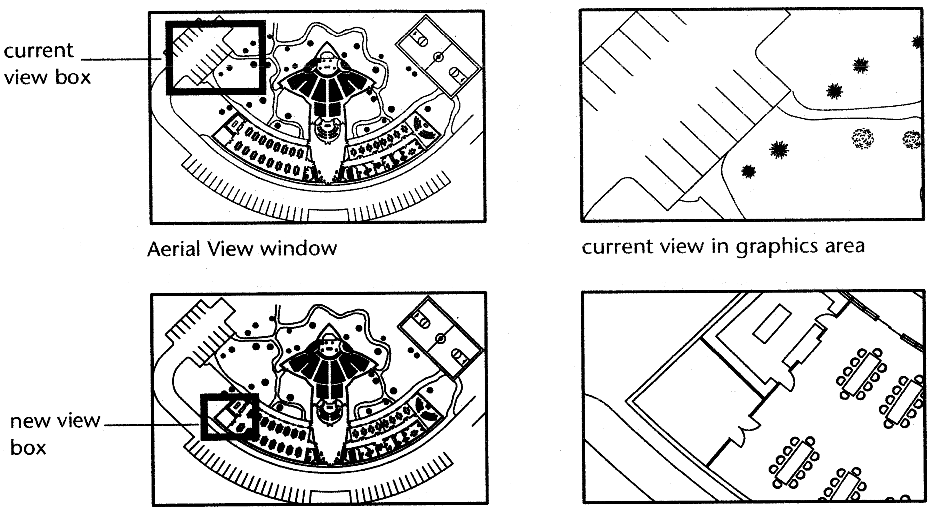
Once you open the Aerial View window, you can keep it visible as you work and then close it when you no longer need it.

To display the Aerial View window

Standard Toolbar	Menu	Command
	View ➤ Aerial View	DSVIEWER

Zooming with Aerial View

You can change the view by creating a new view box in the Aerial View window. To zoom in to the drawing, make the view box smaller. To zoom out of the drawing, make the view box larger. As you zoom in or out of the drawing, a real-time view of the current zoom location is displayed in the graphics area. The following illustration shows how the view box works.



Aerial View window — current view in graphics area

Aerial View window — new view in graphics area

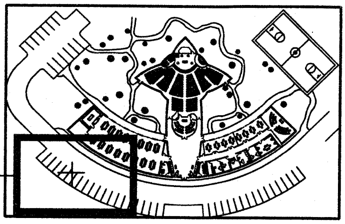
To zoom using Aerial View

1 From the Mode menu in the Aerial View window choose Zoom.
2 Click and drag your pointing device to define the zoom window you want.

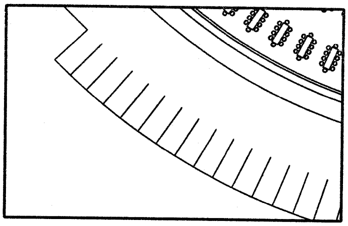
   When you release the pick button, the graphics area zooms along with the Aerial View window to reflect the current zoom view.

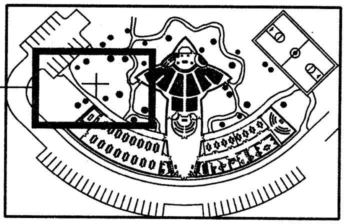
Panning with Aerial View

You can pan the drawing by moving the view box without changing its size. This method changes the view without changing the magnification.

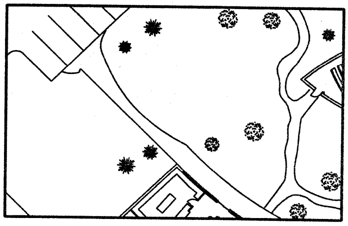


Aerial View window


current view in graphics area



Aerial View window


new view in graphics area

To pan using Aerial View

1 From the Mode menu in the Aerial View window choose Pan.
2 Move your pointing device to reposition the view box. When the view box is positioned where you want it, press the pick button.

   The graphics area shifts along with the Aerial View window to reflect the new position.

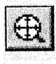
You can also interactively pan the drawing display by pressing and holding the pick button, and then moving the current view box. This will pan the drawing display in real time.

Changing the Size of the Aerial View Image

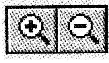
You can change the size of the image in the Aerial View window by displaying the entire drawing or by incrementally resizing the image. These changes do not affect the view in the graphics area.

When the entire drawing is displayed in the Aerial View, the Zoom Out menu item and the Zoom Out button are unavailable. When the current view box nearly fills the Aerial View window, the Zoom In menu item and button are unavailable. It is possible for both these conditions to exist at the same time, causing both to be unavailable, such as when the drawing extents are displayed.

To display the entire drawing in the Aerial View window

Aerial View Toolbar	Menu
	View ➤ Global

To increase or decrease the size of the Aerial View image

Aerial View Toolbar	Menu
	View ➤ Zoom In View ➤ Zoom Out

Changing the Aerial View Update

AutoCAD LT automatically updates the Aerial View window to reflect the changes you make in your drawing. When working on complex drawings, you may want to turn off this dynamic updating to improve program performance.

Similarly, if you work with multiple viewports, the Aerial View image changes as you select different viewports. You can turn off this feature so that AutoCAD LT updates the Aerial View window only when you activate it.

To turn dynamic updating on and off

Menu
Options ➤ Dynamic Update

A check mark indicates that the Aerial View window shows changes as they occur.

To turn viewport updating on and off

Menu
Options ➤ Auto Viewport

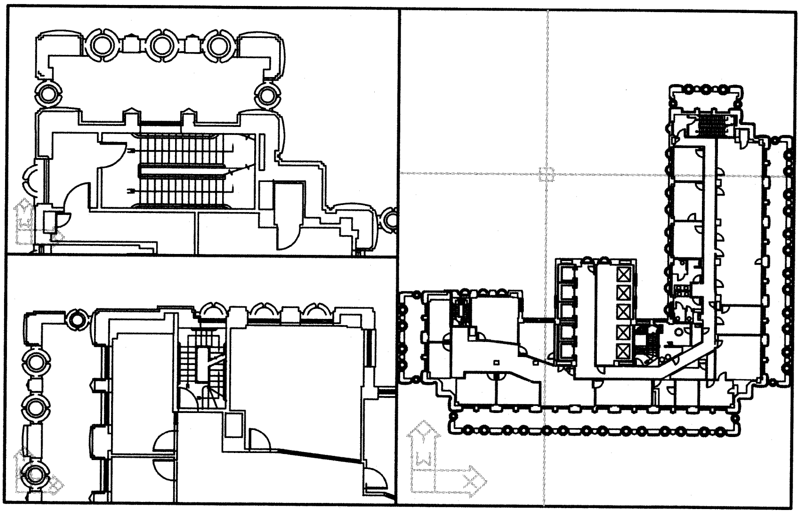
A check mark indicates that Aerial View displays the current viewport as you switch viewports.

ONLINE HELP See "Using the Aerial View Window."

Using Tiled Viewports

AutoCAD LT usually begins a new drawing using a single viewport that fills the entire graphics area. (A *viewport* is a rectangular region of the graphics area that displays all or a portion of the drawing.) You can split the graphics area to display several viewports simultaneously. For example, if you keep both the full and the detail views visible, you can see the effects of your detail changes on the entire drawing.

The following illustration shows a drawing with three tiled viewports. The crosshairs cursor is in the current viewport. The viewports completely fill the graphics area and do not overlap.



Tiled viewports

In each tiled viewport, you can

- Pan; zoom; set snap, grid, and UCS icon; and restore named views in individual viewports
- Draw from one viewport to another when executing a command
- Name a configuration of viewports so that you can reuse it

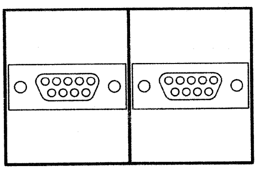
As you draw, changes made in one viewport are instantly reflected in the others. You can switch among these viewports at any time, including in the middle of a command.

Tiled viewports differ from the viewports arranged in paper space; only the currently selected tiled viewport can be printed or plotted. Paper space viewports, also known as floating viewports, are used to establish a final layout for a drawing. They can overlap and be printed or plotted at the same time. For more information about paper space viewports, see "Preparing a Layout" on page 334.

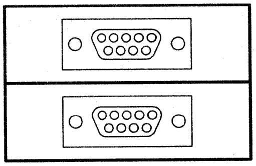
Displaying Multiple Tiled Viewports

You can display tiled viewports in various configurations. How you display the viewports depends on the number and size of the views you need to see. If you're not familiar with the available configurations, you can select a display by choosing its picture.

The following illustrations show the default viewport configurations.

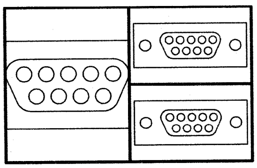


vertical

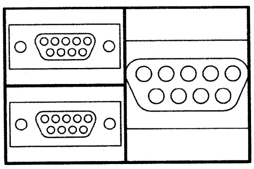


horizontal

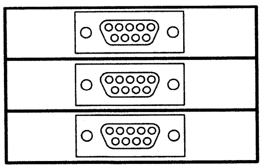
Two viewports



left

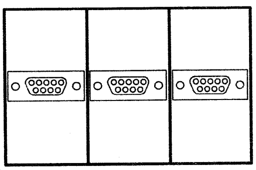


right

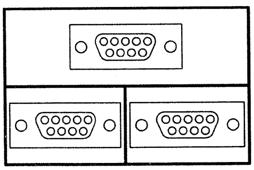


horizontal

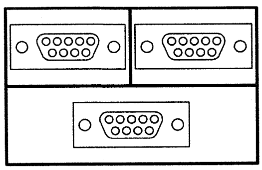
Three viewports



vertical

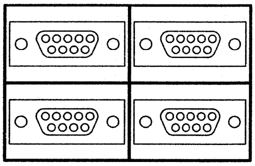


above



below

Three viewports



Four viewports

To display a tiled viewport configuration

Menu	Command
View ➤ Tiled Viewports ➤ Layout	VPORTS

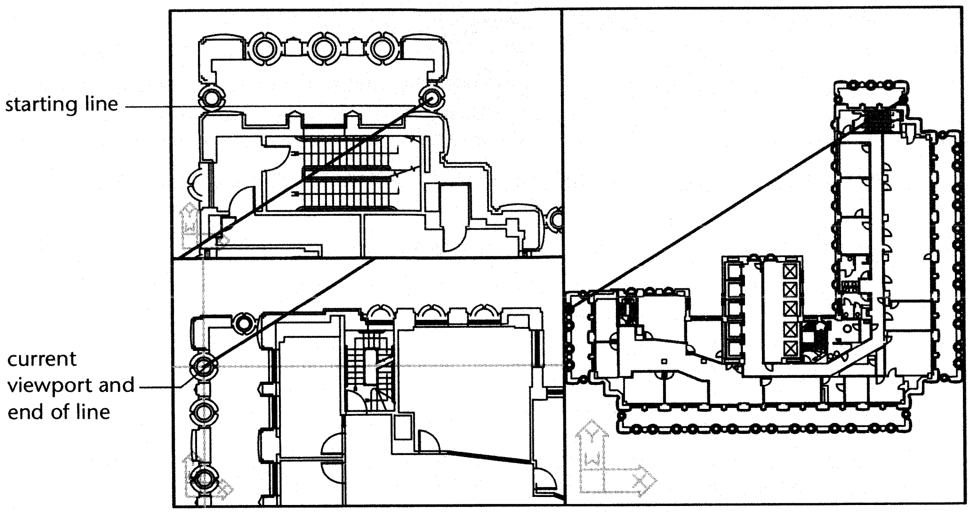
To display tiled viewports, make sure Tile is highlighted in the status bar. If Tile is not highlighted, double-click it.

If you need more viewports than the standard configurations provide, you can configure additional viewports inside the current viewport without replacing the entire display.

Also, you can join adjacent viewports if their common boundary is the same size. When you join viewports, the view is based on the first viewport you select.

Working in Tiled Viewports

Use tiled viewports to see how changes in detail affect the drawing overall. When you work with multiple tiled viewports, you can connect elements in one viewport to elements in another. For example, in a very large drawing, you can draw a line from a detail in one corner to a detail in a distant corner by displaying each section and then drawing the line from one viewport to the other.



Drawing a line between viewports

As you work, you can display different views of the drawing, such as a plan or a front or side elevation. The viewports can show the progress of your work from different views. Errors you might miss in one view may be apparent in other views. You can also minimize repetitive work when working on large drawings. For example, you might copy notes and dimensions of common features such as stairs and bathrooms from one view to another view.

You make a tiled viewport current by clicking within it. When a viewport is current, the arrow cursor changes to crosshairs, and the borders are highlighted. You can enter points and select objects in the current viewport. You can also switch between viewports in the middle of a command.

For example, to draw a line using two viewports, you must start the line in the current viewport, make another viewport current by clicking within it, and then specify the endpoint of the line in the second viewport.

Reusing Viewport Configurations

You can set up and save tiled viewport configurations for reuse so that you don't have to set up the viewports and views every time you need them.

Saving Viewport Configurations

When you name and save a specific viewport configuration with the current drawing, the saved information includes the number of viewports, their positions on the screen, and the settings for each viewport. These settings are the same as the settings saved with individual views (see "Saving Views" on page 214).

To save a viewport configuration

Menu	Command
View ➤ Tiled Viewports ➤ Save	VPORTS Save

The configuration name you specify can be up to 31 characters in length.

Restoring Viewport Configurations

You can restore a named viewport configuration any time you need to redisplay the views it contains.

To restore a viewport configuration

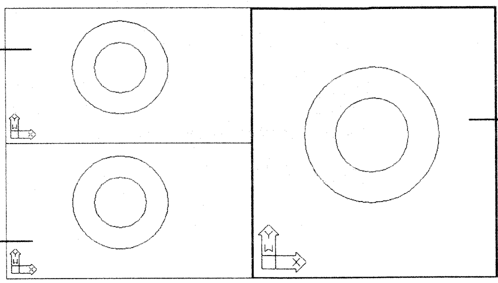
Menu	Command
View ➤ Tiled Viewports ➤ Restore	VPORTS Restore

By entering **?** and pressing ENTER twice you can see a list of configurations. Enter the name of the viewport configuration you want to use.

If you no longer need a configuration, you can delete it. At the Command prompt, enter **vports** and then **delete**.

Listing Viewport Configurations

You can display information about the current and saved configurations. A list in the text window describes the location of each viewport's lower-left and upper-right corners. These corners are described by values in which 0,0 represents the lower-left corner of the graphics area and 1,1 represents the upper-right corner.



Three-viewport configuration

```
Current configuration:
id# 2
   corners:   0.5000,0.0000   1.0000,1.0000   current viewport
id# 3
   corners:   0.0000,0.5000   0.5000,1.0000
id# 4
   corners:   0.0000,0.0000   0.5000,0.5000

Configuration PLANS:                          named configuration with three viewports
   0.5000,0.5000   1.0000,1.0000
   0.5000,0.0000   1.0000,0.5000
   0.0000,0.0000   0.5000,1.0000

Configuration 3D_VIEWS:                       named configuration with four viewports
   0.5000,0.0000   1.0000,0.5000
   0.0000,0.5000   0.5000,1.0000
   0.0000,0.0000   0.5000,0.5000
   0.5000,0.5000   1.0000,1.0000
```

Sample list of viewport configurations

To list the viewport configurations

1 At the Command prompt, enter **vports**.
2 To list all the viewport configurations in the drawing, enter **?** and press ENTER twice.

ONLINE HELP See "Working With Tiled Viewports."

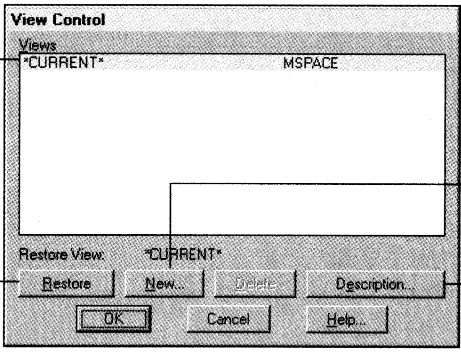
Using Named Views

In the course of using ZOOM and PAN to view your drawing, you may want to save a particular view. A *view* of a drawing includes the zoom value and view position settings, as well as any grid, snap, and UCS icon settings.

You can name and save a view you want to reuse. When you no longer need the view, you can delete it.

Saving Views

When you save a view, the viewing position and scale are saved. When you begin a new drawing, you usually use a single viewport that fills the entire graphics area. If you are working with multiple viewports, the view in the current viewport is saved. If you are working in paper space, the paper space view is saved. If you want to save only part of the current view, you can specify a window and save the view within the window.



The name can be up to 31 characters long and contain letters, digits, and the special characters dollar sign ($), hyphen (–), and underscore (_).

To save a view

Standard Toolbar	Menu	Command
	View ➤ Named Views	DDVIEW

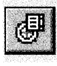
Related VIEW saves views from the command line.

Restoring Named Views

When you need to reuse a named view, you can restore it. If you are working with multiple viewports, the view is restored to the current viewport. You can use named views in a paper space layout or specify them when you plot (see "Using Tiled Viewports" on page 208 and "Using Named Views in Paper Space" on page 344).

Click within the viewport that contains the view you want to replace, and then choose Named Views from the View menu. Select the name of the view from the list, and click Restore.

To restore a named view

Standard Toolbar	Menu	Command
	View ➤ Named Views	DDVIEW

When you no longer need a view, you can delete it.

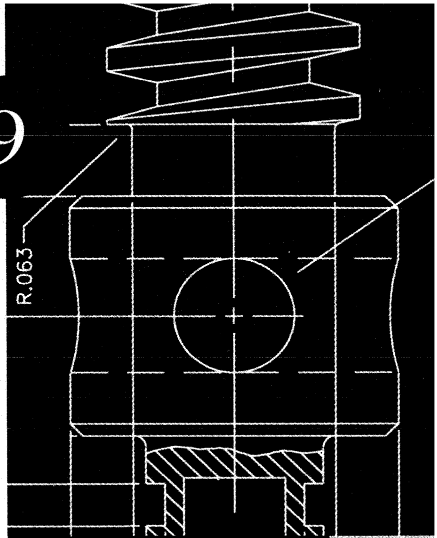
ONLINE HELP See "Saving and Restoring Views by Name."

Turning Visual Elements On and Off

The complexity of your drawing affects how fast AutoCAD LT refreshes the screen or processes commands. You can increase program performance by turning off the display of text, fill, or selection highlighting. You can also turn off blips, temporary markers that indicate where you have clicked your cursor in the graphics area.

Increasing program performance by turning off visual elements

To turn off the display of...	Choose...
Solid-filled objects such as polygons and splines	Tools ➤ Drawing Aids; turn off Solid Fill
Text	Tools ➤ Drawing Aids; turn on Quick Text
Selection highlighting	Tools ➤ Drawing Aids; turn off Highlight
Blips (off by default)	Tools ➤ Drawing Aids; turn off Blips
UCS icon	View ➤ Display ➤ UCS Icon ➤ Off



9

Adding Text to Drawings

Text conveys important information in your drawing. You use text for title blocks, to label parts of the drawing, to give specifications, or to make annotations.

AutoCAD LT provides various ways to create text. For short, simple entries, use line text. For longer entries with internal formatting, use multiline, or paragraph, text. Although all entered text uses the current text style, which establishes the default font and format settings, you can use several methods to customize the text appearance.

In this chapter

- Creating and changing collections of formats known as styles
- Creating and editing single-line and multiline text
- Overriding text styles by formatting individual characters
- Changing existing text
- Substituting fonts
- Checking the spelling of text

Creating and Modifying Text Styles

All text in an AutoCAD LT drawing has a *style* associated with it. When you enter text, AutoCAD LT uses the current text style, which sets the font, size, angle, orientation, and other text characteristics. Text style controls the attributes listed in the following table.

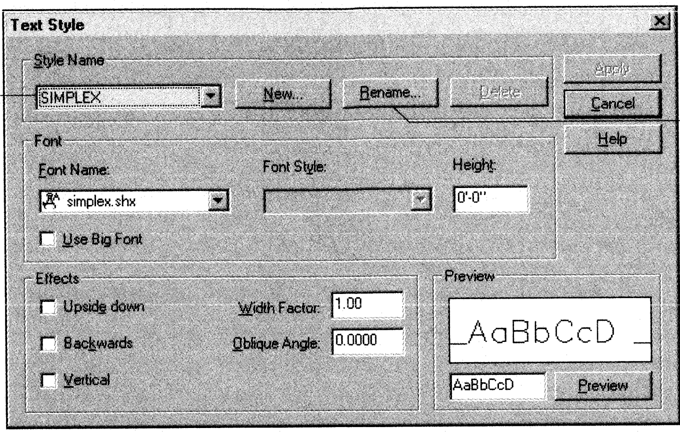
Style settings

Setting	Default	Description
Style name	STANDARD	Name with up to 31 characters
Font file	*txt.shx*	File associated with a font (character style)
Big Font file	none	Special shape definition file used for a non-ASCII character set, such as kanji
Height	0	Character height
Width factor	1	Expansion or compression of the characters
Obliquing angle	0	Slant of the characters
Backwards	No	Backwards text
Upside-down	No	Upside-down text
Vertical	No	Vertical or horizontal text

You can use or modify the default style or create and load new styles. Once you've created a style, you can modify its attributes, change its name, or delete it when you no longer need it.

Except for the default STANDARD style, you must create any text style that you want to use. New text inherits height, width factor, obliquing angle, and backwards, upside-down, and vertical alignment properties from the current text style. When you create or modify a text style, you use the Text Style dialog box.

To create text using a different style, you can change the current style



Renaming an existing text style updates any text using the old name to the new text style name

You can modify an existing style in the Text Style dialog box by changing the settings. The sample text in the Preview area is updated to show the changes you make to the style.

You can also update existing text of that style type to reflect the changes. If you change an existing style's font or orientation, all text using that style is regenerated using the new font or orientation. Changing text height does not change existing text but does change subsequently created text objects. Changes to alignment, width, and rotation have no effect on multiline text objects. See "Creating Multiline Text" on page 226.

For more information about modifying text style characteristics, see "Assigning Fonts" below, "Setting Text Height" on page 221, "Setting Obliquing Angle" on page 221, and "Setting Horizontal or Vertical Orientation" on page 222.

To change text style options

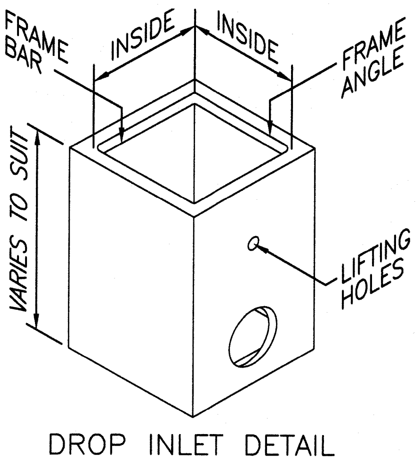
Menu	Command
Format ➤ Text Style	DDSTYLE

ONLINE HELP See "Working with Text Styles."

Assigning Fonts

Fonts define the shapes of the text characters that make up each character set.

A single font can be used by more than one style. If your company has a standard font type, you can modify other style settings to create a set of styles that use a standard font in different ways. The following illustration shows the same font used by different styles that have different obliquing settings to define the slant of the text.



Same font used with different obliquing angles

You can assign a font to a text style by selecting either a TrueType typeface name and its font style (bold or italic, for instance) or an AutoCAD LT SHX font.

ONLINE HELP See "Assigning Text Fonts."

Using TrueType Fonts

TrueType fonts always appear filled in your drawing. However, when you plot, the TEXTFILL system variable controls whether the fonts are filled. By default TEXTFILL is set to 1, to plot the filled-in fonts.

ONLINE HELP See "Using TrueType Fonts."

Using Unicode and Big Fonts

AutoCAD LT supports the Unicode character-encoding standard. A Unicode font can contain 65,535 characters, with shapes for many languages. Unicode fonts contain many more characters than are defined in your system; therefore, to use a character not directly available from the keyboard, you can enter the escape sequence **\U+nnnn**, where *nnnn* represents the Unicode hexadecimal value for the character. All AutoCAD LT SHX shape fonts are Unicode fonts.

ONLINE HELP See "Using Unicode and Big Fonts."

Setting Text Height

Text height determines the size in drawing units of the letters in the font you are using. The value usually represents the size of the uppercase letters, with the exception of TrueType fonts.

The text height you set should be determined by the scale factor for your drawing, or the ratio of plotted units to drawing units. If you specify text height as part of a text style you can create different text styles for different scale factors. For more information on determining the scale factor of your drawing, see "Determining the Scale Factor" on page 70.

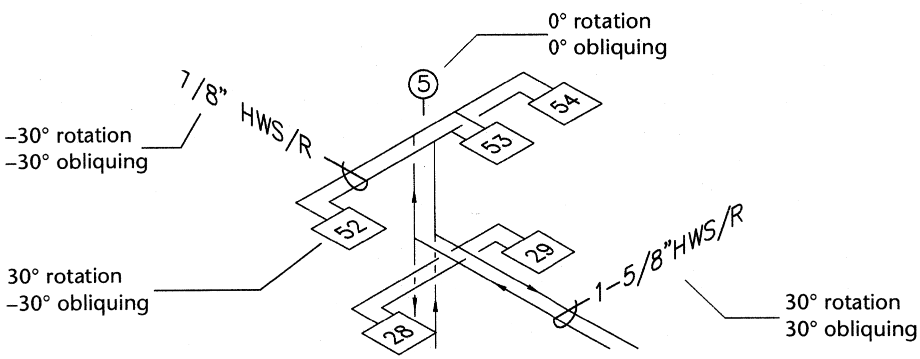
If you specify a fixed height as part of a text style, AutoCAD LT bypasses the Height prompt when you create line text. When the height is set to 0 in the text style, AutoCAD LT prompts for the height each time you create line text. Set the value to 0 if you want to change the height as you create text.

NOTE For display only, the height of text in the Multiline Text Editor may be adjusted by AutoCAD LT to fit within the width of the dialog box.

ONLINE HELP See "Setting Text Height."

Setting Obliquing Angle

The obliquing angle determines the forward or backward slant of the text. The angle represents the offset from 90 degrees.



Text created using various obliquing and rotation angles

Entering a value between **–85** and **85** makes the text oblique. A positive obliquing angle slants text to the right. A negative obliquing angle slants text to the left. An angle of zero degrees restores oblique text to its original setting.

ONLINE HELP See "Setting Obliquing Angle."

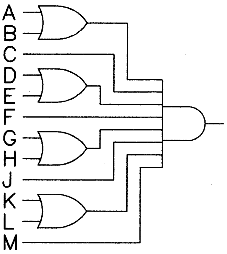
Setting Horizontal or Vertical Orientation

AutoCAD LT orients line text to be vertical or horizontal. Text can have a vertical orientation only if the associated font supports dual orientation. You can create more than one line of vertical text. Each successive text line is drawn to the right of the preceding line. The normal rotation angle for vertical text is 270 degrees.

Vertical orientation is not supported for TrueType fonts.

ONLINE HELP See "Setting Horizontal or Vertical Orientation."

Using Line Text



vertical text

The text you add to your drawings conveys a variety of information. It may be a complex specification, title block information, a label, or even part of the drawing. For short entries that do not require multiple fonts or lines, create line text using DTEXT or TEXT. Line text is more convenient for labels. For longer entries, use MTEXT. See "Using Multiline Text" on page 225.

Creating Line Text

When you create one or more lines of single-line text, each line ends when you press ENTER. Each text line is an object that you can relocate, reformat, or otherwise modify.

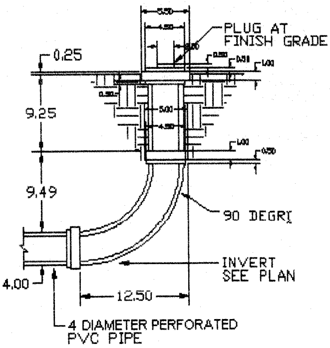
If you select another point during this command, the cursor box moves to that point, and you can continue typing. This action creates a separate text object. Press ENTER on a blank line to end text creation.

To create line text

Menu	Command
Draw ➤ Single Line Text	DTEXT

1 From the Draw menu, choose Single Line Text.
2 Specify the insertion point for the first character.
3 Specify the text height if you are prompted for it, enter a rotation angle, and then begin typing.

Related TEXT creates text on the command line but does not display it in the graphics area until you complete the entry. You must enter the command again before you start a second line.



Each line of text is a separate object, created in succession by selecting new points during the DTEXT command.

Formatting Line Text

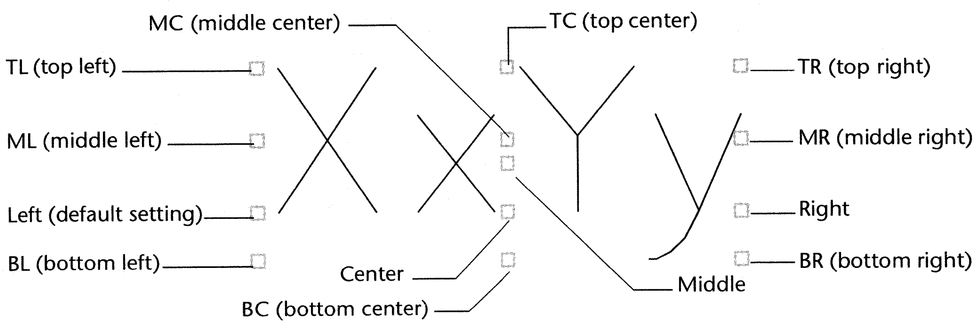
If you want to assign a text style and set alignment for line text, you specify the options on the command line before you enter the text. Use Justify to specify what part of the text character aligns with the insertion point. Use Style to set the default format characteristics. You *cannot* apply different formats to individual words and characters within the same text object.

Assigning a Style to Line Text

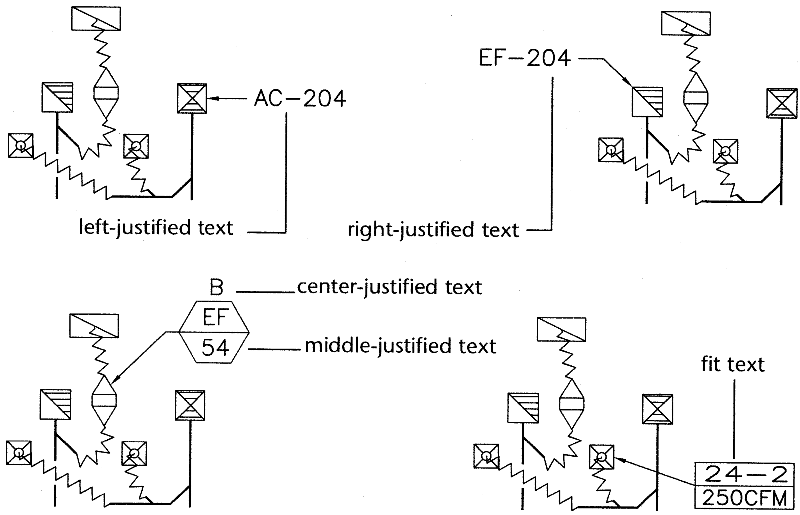
When you create text, you can assign an existing style by entering its name at the Style prompt. The styles used for line text are the same as those used for multiline text.

Aligning Line Text

As you create text, you can align it horizontally. That is, you can justify it with one of the alignment options shown in the following illustration. Left alignment is the default. To left-align text, do not enter an option after the Justify prompt.



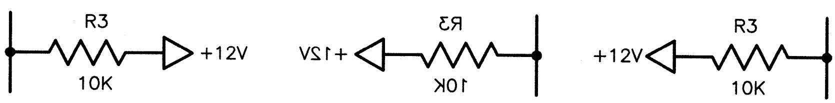
You also can fit line text between points that you specify. This option stretches or squeezes the text to fill the designated space. All of the alignment options are useful for creating text that must align with a specific insertion point or within geometric constructions.



Examples of text aligned using justification options

Changing Line Text

Like any other object, a line text object can be moved, rotated, erased, and copied. You also can mirror, or make a reverse copy of, text. If you do not want the text to be reversed when you mirror it, you can set the MIRRTEXT system variable to 0. For more information about mirroring, see "Mirroring Objects" on page 165.



object

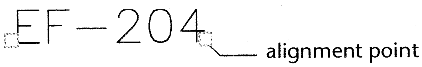
object mirrored

object mirrored
MIRRTEXT = 0

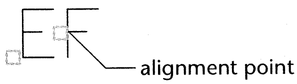
The following table describes two different ways you can change line text, depending on whether you need to update only the content, or the formatting also.

Commands for changing line text	
To change...	**Choose...**
Content only	Modify ➤ Object ➤ Text (DDEDIT)
Content, style, location, orientation, or justification	Modify ➤ Properties (DDMODIFY)

Text objects also have grips for stretching, scaling, and rotating. A line text object has grips at the lower-left corner of the baseline and at the alignment point.



grips for right-justified line text



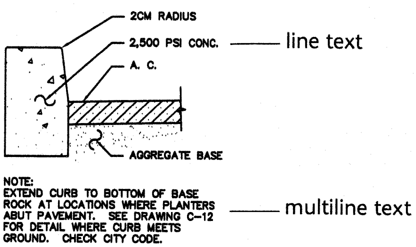
grips for middle-justified line text

The effect of a command depends on which grip you choose. For more information, see "Editing with Grips" on page 151.

ONLINE HELP To create line text, see "Creating Single-Line Text." To format or change line text, see "Changing Text."

Using Multiline Text

For long, complex entries, create multiline, or paragraph, text using MTEXT. Multiline text fits a specified width but can extend vertically to an indefinite length. In contrast to single-line text, you can format individual words or characters within the multiline text. For example, you can apply underlining, font, color, and text height changes to individual words or phrases within a paragraph.

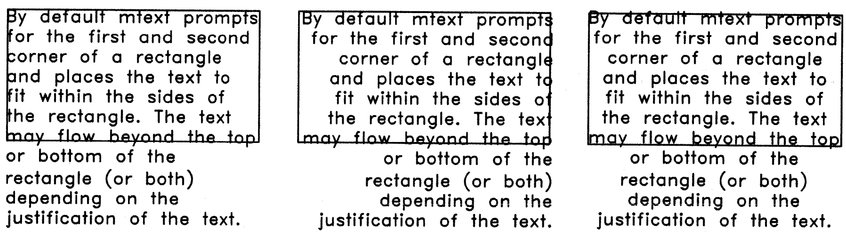


Multiline text consists of any number of text lines or paragraphs that fit within a width you specify. Regardless of the number of lines, each set of paragraphs created in a single editing session forms a single object, which you can move, rotate, erase, copy, mirror, stretch, or scale. See chapter 7, "Editing Your Drawing."

Creating Multiline Text

You can create text in the Multiline Text Editor dialog box, on the command line, or with an alternative text editor, which you can specify with the MTEXTED system variable. The Multiline Text Editor dialog box provides a quick way to set properties that affect the entire object and formats that affect only selected text.

Before creating the text, you define the paragraph's width by specifying the two opposite corners of a text boundary. When text entry is complete, AutoCAD LT inserts the text entered in the dialog box within this width limit. You can apply the text height, justification, rotation angle, and style to the text object, and apply character formatting to selected characters. Justification determines where the text is inserted with respect to the text boundaries. See "Formatting Multiline Text" on page 228.



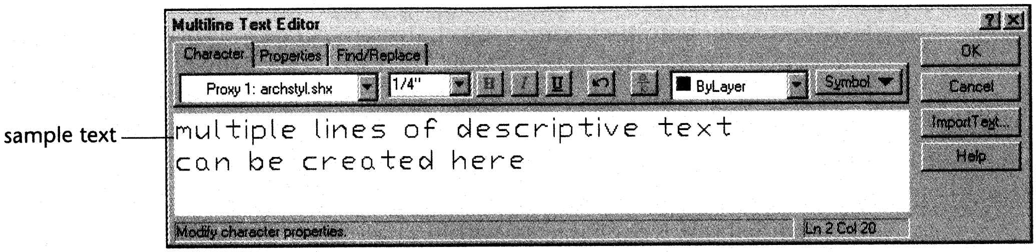
left-justified right-justified center-justified

The length of the multiline text object you create with MTEXT depends on the amount of text, *not* the length you specify when selecting the opposite corner of the boundary box on screen.

To create multiline, or paragraph text

1 From the Draw menu choose Paragraph Text.
2 Specify the corners of the text boundary.

  Notice the arrows inside the boundary box, which indicate the direction entered text flows based on the current justification setting.
3 In the Multiline Text Editor dialog box, enter the text, allowing it to wrap to the next line.



For information about changing style and format, see "Formatting Multiline Text" on page 228.

Words that extend beyond the bounding rectangle of the paragraph text object you specified wrap to the next line in the Multiline Text Editor.

You can use the standard Windows control keys described in the table to enter text.

Control keys used to edit text

Key	Description
CTRL+C	Copies selection to the Clipboard
CTRL+V	Pastes Clipboard contents over selection
CTRL+X	Cuts selection to the Clipboard
CTRL+SPACEBAR	Inserts a nonbreaking space
ENTER	Ends the current paragraph and starts a new line

To create multiline text

Draw Toolbar	Menu	Command
A	Draw ➤ Paragraph Text	MTEXT

ONLINE HELP See "Creating Multiline Text."

Formatting Multiline Text

New text automatically assumes the characteristics of the current text style. The STANDARD text style is the default. You can override the default text style by applying formatting to individual characters and applying properties to the text object. You can also specify formatting or insert special characters using the methods described in the following sections.

Formatting options such as underlining, stacked text, or fonts can be applied to individual words or characters within a paragraph. Properties such as style, justification, width, and rotation affect the entire text object. You can change both formats and properties in the Multiline Text Editor dialog box.

Formatting Individual Words or Characters

As you create multiline text, you can apply formats to selected text. The format changes affect only the characters you select. The current text style is not changed.

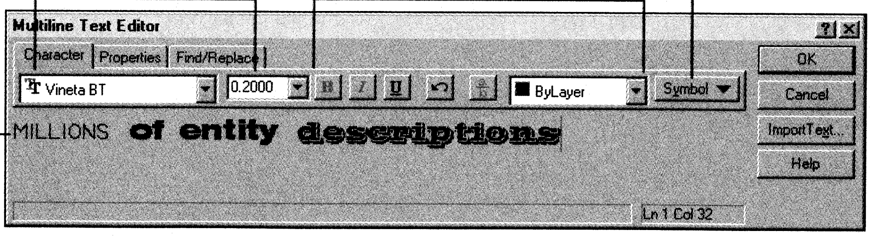
Text height sets the height of capitalized text. This height establishes the default value for all the text in the text boundary. You can override it by specifying a different height for selected text. The change of text height displayed in the Multiline Text Editor is proportional to the change in height of the actual characters as they are displayed.

Choose the font and text heights.

Apply character formats, stack numbers, change text color, or undo changes.

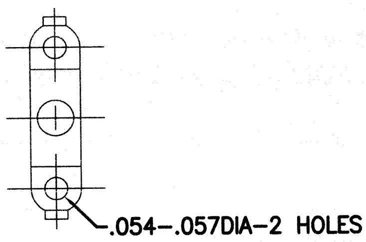
Click Symbol to insert special characters.

sample text

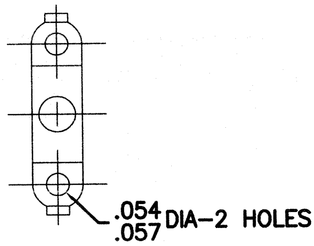


Creating Stacked Text

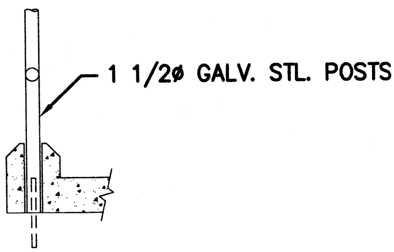
Stacked text is text or fractions aligned vertically.



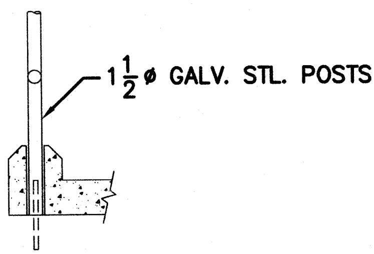
plain text



stacked text



plain fraction



stacked fraction

Examples of stacked text

AutoCAD LT uses the special characters slash (/) and caret (^) to indicate where selected text should be stacked. By default, stacked characters are vertically aligned at the midpoint of the text that precedes them. Enter **054^057** to stack text as shown in the first example in the illustration. Enter **1/2** to stack fractions as shown in the second example. Highlight the text that you want to stack and click the Stack icon on the Character tab.

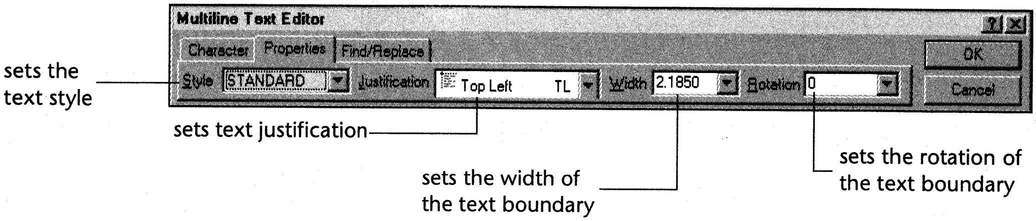
Text stacked in the text editor is always aligned with the text baseline.

ONLINE HELP To top-align or middle-align stacked text, see "Formatting Multiline Text in a Text Editor."

Formatting Multiline Text Objects

As you create multiline text, you can set the properties that control style, text justification, and the size and rotation of the text boundary. These settings affect all text within the text boundary, not specific words or characters.

You can set these properties on the command line before the Multiline Text Editor dialog box opens. If you already are in the Multiline Text Editor dialog box, choose the Properties tab and change the settings using the Change Properties toolbar.



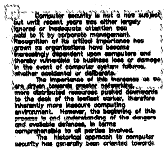
The Style option sets the default fonts and formatting characteristics for new text. As you create text, you can select which style you want to use from a list of existing styles.

When you change the style of a multiline text object that has character formatting applied to any portion of the text, the style is applied to the entire object, and some formatting of characters might not be retained. For instance, changing from a TrueType font to a style using an SHX font or another TrueType font causes the text to use the new font for the entire object, and any character formatting is lost.

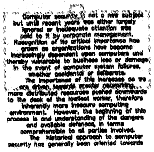
Effects of style or font change on character formatting

Formatting	Retained?
Bold	No
Color	Yes
Font	No
Height	No
Italic	No
Stacking	Yes
Underlining	Yes

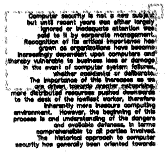
The justification option controls both text alignment and text flow based on a specified justification point. Text is justified with respect to the left and right text boundaries. Text flows from the middle, the top, or the bottom of the paragraph with respect to the top and bottom text boundaries. The top and bottom boundaries are based on the top and bottom lines of the multiline text object. AutoCAD LT offers nine justification settings.



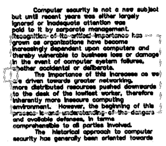
top left:
left-aligned



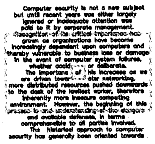
top center:
centered



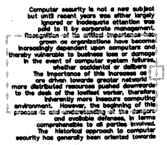
top right:
right-aligned



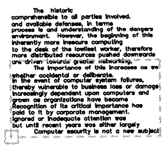
middle left:
left-aligned



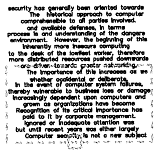
middle center:
centered



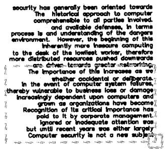
middle right:
right-aligned



bottom left:



bottom center:



bottom right:

You can adjust line width by entering the line width (*not* the character width) in drawing units in the Width box. Width can also be adjusted using grips. For more information on using grips, see "Editing with Grips" on page 151.

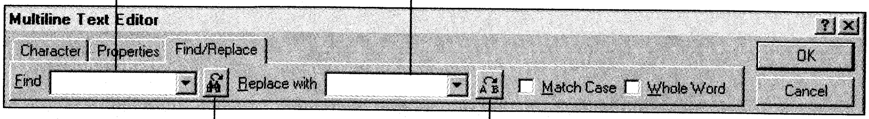
Rotation controls the angle of rotation of the text boundary.

Finding and Replacing Multiline Text

To quickly search for and replace text on a word-by-word basis, use the find and replace feature in the Multiline Text Editor dialog box. Replacement is based on text content only; character formatting and text properties are unchanged.

In the Find box, enter the text to change.

In the Replace with box, enter the word to replace the found text.

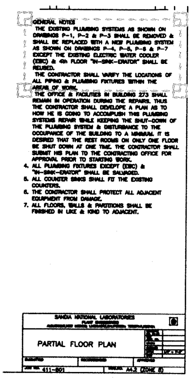


Click the Find tool to locate the specified text.

Click the Replace tool to replace the text.

When the text object has been completely searched, a message at the bottom of the dialog box indicates that the search is resuming at the beginning of the text object.

Changing Multiline Text



grips

As with line text, multiline text objects can be moved, rotated, erased, and copied. You also can mirror, or make a reverse copy of, text. If you do not want the text to be reversed when you mirror it, you can set the MIRRTEXT system variable to 0. For more information on mirroring objects, see "Mirroring Objects" on page 165.

There are two different ways you can change multiline text, depending on whether you need to update only the content and formatting, or change the location and such properties as layer and color also.

- DDEDIT changes content, formatting, and text object properties such as style and justification.
- DDMODIFY changes the origin, or insertion point, of the text object as well as color and layer properties.

Changing Multiline Text Content and Formatting

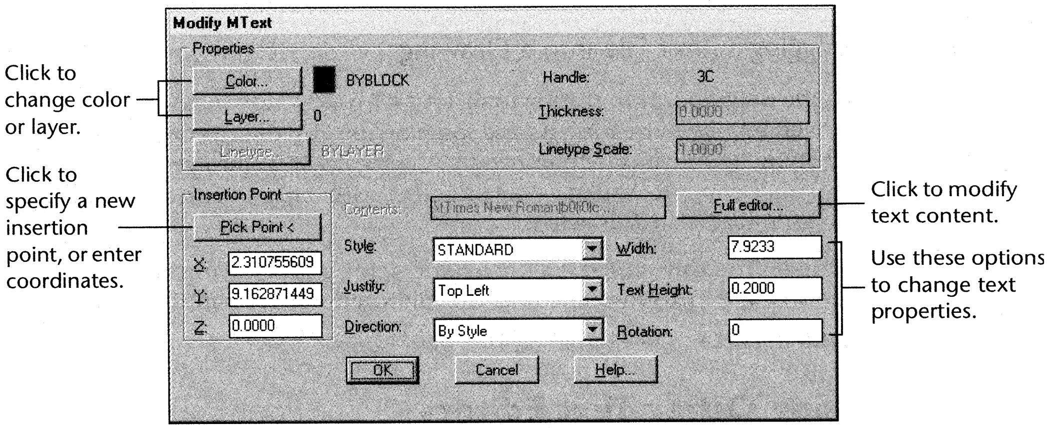
Use DDEDIT when you want to change either content or formatting of multiline text. The changes you make affect only the text you select, not the text style.

To change multiline text content and formatting

Modify II Toolbar	Menu	Command
	Modify ➤ Object ➤ Text	DDEDIT

Modifying Multiline Text Properties and Content

Use DDMODIFY when you want to modify multiline text properties. You can move text, change its layer and color, as well as edit content and change properties such as style, justification, and text boundary size.



To change multiline text location, color, and layer

Object Properties Toolbar	Menu	Command
	Modify ➤ Properties	DDMODIFY

ONLINE HELP See "Changing Multiline Text."

Using External Text Files

You can insert plain text files created in other word processors into your AutoCAD LT drawing. You can either import the text file or drag a file icon from Windows Explorer.

Importing Text Files

You can save time by importing TXT or RTF files from other sources. For example, you can create a text file of standard notes that you include in drawings. Instead of entering this information each time you use it, you can import the file. The imported text becomes an AutoCAD LT text object, which you can edit as if you created it in AutoCAD LT. Imported text retains its original formatting properties. Imported text files are limited to a maximum of 16 KB.

NOTE If the Clipboard contains text, you can choose Paste from the Edit menu to paste the Clipboard contents into AutoCAD LT.

Dragging a Text File into a Drawing

Use the drag-and-drop feature to insert TXT or RTF text files into a drawing. Inserted text uses the formats and fonts defined by the current AutoCAD LT text style. Its width is determined by line breaks and carriage returns in the original document. You can drag only files with a file extension of *.txt* or *.rtf* into an AutoCAD LT drawing.

Related You can start DTEXT before dragging a text file into AutoCAD LT. When you use this method, drag the file when you see the Text prompt.

ONLINE HELP See "Importing Text from External Files."

Using Other Text Editors

Use the MTEXTED system variable to specify the text editor you want to use to create and modify paragraph text. If MTEXTED is set to Internal or is not set, it uses the Multiline Text Editor dialog box. For example, you can specify the Notepad text editor.

ONLINE HELP See "Using an Alternate Text Editor."

Substituting Fonts

You can designate fonts to be substituted for other fonts or as defaults when AutoCAD LT cannot find a font specified in a drawing.

The fonts used for the text in your drawing are determined by the text style and, for multiline text, by individual font formats applied to sections of text. You might want to ensure that your drawing uses only certain fonts, or perhaps you might want to convert the fonts you used to other fonts. You can use any text editor to create font mapping tables for both of these purposes.

You can use these font mapping tables to enforce corporate font standards, or to facilitate off-line printing. For example, if you share drawings with consultants, you may want to use a font mapping table to specify what font AutoCAD LT substitutes when it encounters a text object created with another font. Similarly, to edit the drawing using quicker-drawing SHX fonts, and then switch to more complex fonts for the final plot, you can set up a font mapping table that converts each SHX font to an equivalent.

The font mapping table is a plain ASCII text (FMP) file containing one font mapping per line. Each line contains the base name of the font file (with no directory name or path) followed by a semicolon (;) and the name of the substitute font file. The substitute file name includes a file extension such as *.ttf*.

For example, you could use the following entry in a font map table to specify that the *times.ttf* TrueType font file be substituted for the *romanc.shx* font file:

romanc; times.ttf

AutoCAD LT comes with a default font mapping table in the file *aclt.fmp* in the program folder. You can edit this file using any ASCII text editor. You also can specify a different font mapping table file in the Preferences dialog box, or you can specify the font mapping file using the FONTMAP system variable.

To specify a font mapping table

Menu	Command
Tools ➤ Preferences	PREFERENCES

Related The FONTMAP system variable specifies the name of the current font mapping table.

With AutoCAD LT, you can specify an alternate font file used to display text when a previously saved drawing is opened and the original font file is no longer available on your system. For example, Arial is a TrueType font that is installed with Windows 95 or NT and therefore is a suitable alternate font setting. Use the Files tab in the Preferences dialog box to change the default alternate font file. By default, the alternate font file is set to *simplex.shx*.

ONLINE HELP See "Setting Interface Preferences."

To set an alternate font

Menu	Command
Tools ➤ Preferences	PREFERENCES

Related The FONTALT system variable specifies the name of the current alternate font.

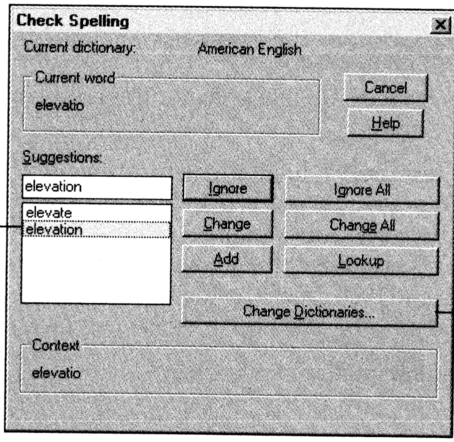
The following table shows the font substitution rules used by AutoCAD LT when a font file cannot be located at the time a drawing is opened.

Font substitution rules

File type	Mapping order			
TTF	Uses FONTMAP value	Uses font defined in text style	Windows substitutes a similar font	
SHX	Uses FONTMAP value	Uses font defined in text style	Uses FONTALT	Prompts for new font
PS	Uses FONTMAP value	Uses FONTALT	Prompts for new font	

Checking Spelling

SPELL checks the spelling in your drawing, including spelling in dimension text. You use one of several main dictionaries, available in different languages. The main dictionaries use a standard word list, which you can customize.



If AutoCAD LT finds a questionable word, it displays the Check Spelling dialog box.

Select the suggested word from the list, then click Change to apply.

Click to change dictionaries, and create and edit custom dictionaries.

To check spelling

Standard Toolbar	Menu	Command
	Tools ➤ Spelling	SPELL

Switching Dictionaries

During a spelling check, AutoCAD LT matches the words in the drawing to the words in the current main dictionary. Any words you add are stored in the custom dictionary that is current at the time of the spelling check. For example, you can add proper names so that AutoCAD LT no longer identifies them as misspelled words.

Creating and Editing Custom Dictionaries

To check spelling in another language, you can change to a different main dictionary. You also can create any number of *custom dictionaries* and switch to them as needed.

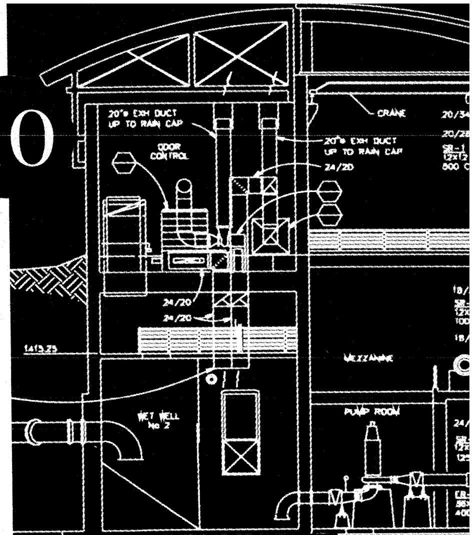
A custom dictionary is a list of spelling exceptions that you have identified. The files that contain them can be identified by the *.cus* file extension. You can use any ASCII text editor to add and delete words or to combine dictionaries. If a word is preceded by a tilde (~), AutoCAD LT always flags the word as incorrect.

You also can create, edit, and set alternate custom dictionaries from inside AutoCAD LT.

ONLINE HELP To check spelling, switch dictionaries, and create and edit custom dictionaries, see "Checking Spelling."

10

Dimensioning and Tolerancing



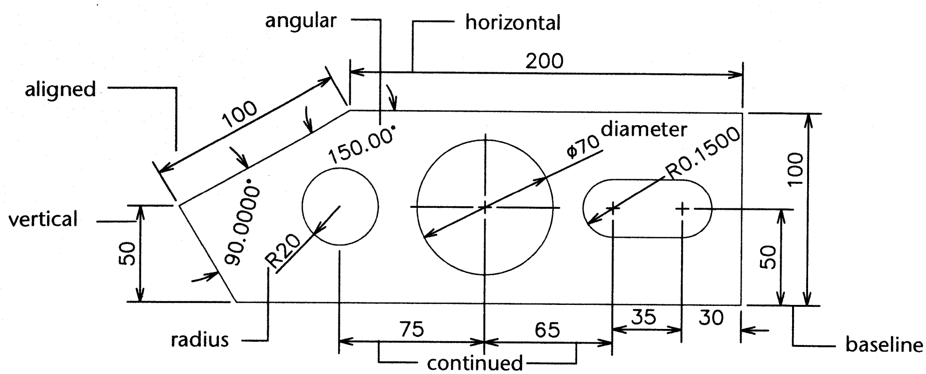
Dimensions add measurements to a drawing. Tolerances specify by how much a dimension can vary. With AutoCAD LT, you can easily manage dimensioning with dimension styles, style families, and overrides.

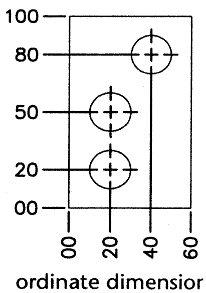
In this chapter

- Understanding dimensioning concepts
- Creating dimension styles
- Setting dimension scale
- Creating linear, radial, angular, and ordinate dimensions
- Creating leader lines with annotation
- Modifying dimensions and styles
- Creating and modifying arrowheads
- Adding geometric tolerancing

Dimensioning Concepts

Dimensions show the geometric measurements of objects, the distances or angles between objects, or the *X* and *Y* coordinates of a feature. AutoCAD LT provides three basic types of dimensioning: linear, radial, and angular. A simple example of each is shown here.



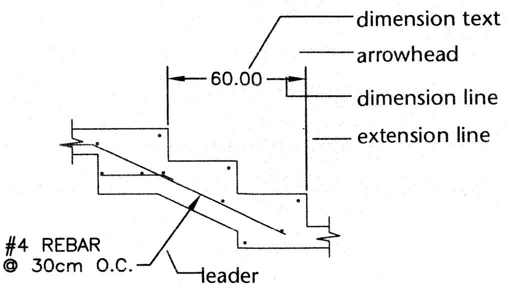


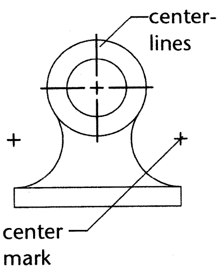
Linear dimensions include horizontal, vertical, aligned, rotated, ordinate, baseline, and continued dimensions. You can dimension lines, double lines, arcs, circles, and polyline segments by selecting the objects using object snaps, or you can draw dimensions that stand alone.

AutoCAD LT draws dimensions on the current layer. Every dimension has a dimension style associated with it, whether the default or one you define. The style controls characteristics such as color, text style, and linetype scale. Style families allow for subtle modifications to a base style for different types of dimensions. Overrides allow for style modifications to a specific dimension.

Parts of a Dimension

Dimensions have several distinct elements: dimension lines, extension lines, dimension text, arrowheads, and leaders.





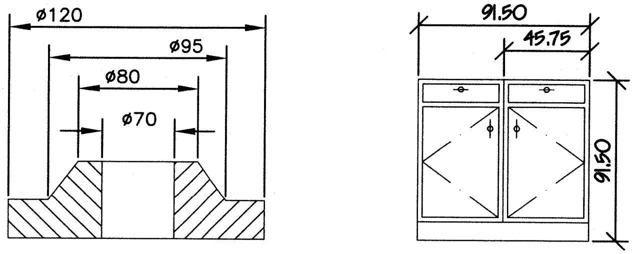
- ***Dimension line*** Indicates the direction and extent of a dimension. For angular dimensioning, the dimension line is an arc.
- ***Extension lines*** Extend from the feature being dimensioned to the dimension line. Also called projection lines or witness lines.
- ***Dimension text*** A text string that usually indicates the actual measurement. The text may also include prefixes, suffixes, and tolerances.
- ***Arrowheads*** Can be added to each end of the dimension line. Also called symbols of termination.
- ***Leader*** A solid line leading from an annotation to the referenced feature.
- ***Center mark*** A small cross that marks the center of a circle or arc.
- ***Centerlines*** Broken lines that mark the center of a circle or arc.

Styles and Style Families

A dimension *style* is a named group of dimension settings that determines the appearance of the dimension. *Style families* are groups of styles based on a *parent style* with variations for different types of dimension. When you create a new style, it automatically becomes the parent style of a style family. Members of dimension style families, or *child* dimension styles, are based on a parent style and apply to the type of dimension you specify. For example, you might create a parent style for all dimensions in your architectural drawings and then create a variation, or child dimension style, specifically for any radial dimensions you create.

Dimension Text

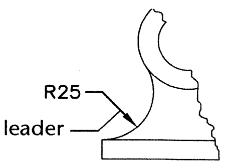
Dimension text is any kind of text that is associated with dimensions, including measurements, tolerances (both lateral and geometric), prefixes, suffixes, and textual notes in single-line or paragraph form. You can use the default measurements computed by AutoCAD LT as the text, supply your own text, or suppress the text entirely. You can also add information, such as special manufacturing procedures or assembly instructions.



Different dimension text styles

AutoCAD LT provides two types of dimension text: line text and paragraph or multiline text. Single-line dimension text uses the text style you set in the DDIM Annotation dialog box. Paragraphs of text use the current text style with any modifications you make in your text editor. For example, in the Multiline Text Editor dialog box, accessed with MTEXT or through the Text option of some dimensioning commands, you can underline text.

Leader Lines



The default *leader line* is a straight line with an arrowhead that refers to a feature in a drawing. Usually, a leader's function is to connect an annotation with the feature. Annotation in this case means paragraph text, blocks, or feature control frames. Such leader lines are different from the simple leader lines AutoCAD LT creates automatically for radial, diameter, and linear dimensions whose text won't fit between extension lines. For a description of these simple leader lines, see "Fitting Text within Extension Lines" on page 268.

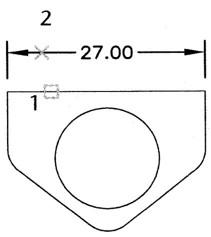
Leader lines are associated with the annotation, so when the annotation is edited, the leader is updated accordingly. You can copy an annotation used elsewhere in a drawing and append it to a leader, or you can create a new annotation using the LEADER command. You can also create a leader with no annotation appended. See "Creating a Leader" on page 252.

Associative Dimensions

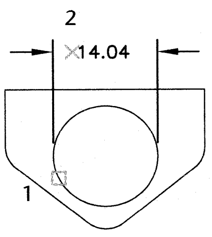
Associative dimensions are dimensions in which all the lines, arrowheads, arcs, and text are drawn as a single dimension object. As a result, you can update a dimension automatically as you modify the object with which it's associated. You can create a nonassociative dimension if you need greater flexibility in controlling the appearance of dimensions. In general, however, associative dimensions are easier to maintain because they can be modified as a single object.

Creating Dimensions

You can create dimensions by selecting an object to dimension (1) and specifying the dimension line location (2).



result of selecting a line to dimension



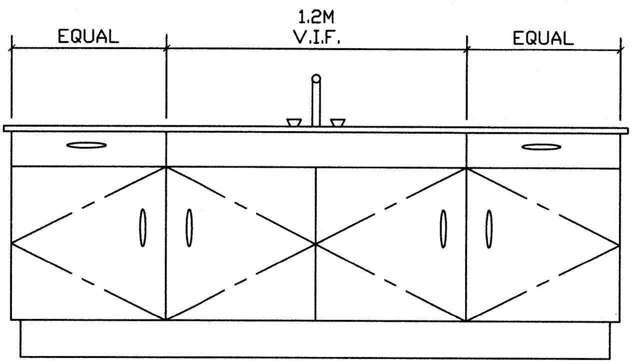
result of selecting a circle to dimension

You can also create dimensions by specifying the extension line origins. For lines, polyline segments, and arcs, the default extension line origins are the endpoints. For circles, the endpoints of the diameter are used at the specified angle. You could also use the Quadrant object snap for example, to snap to adjacent quadrants of a circle.

Setting Dimension Text Options on the Command Line

As you create dimensions, you can modify the dimension text and its angle relative to the dimension line by entering *any* of the following options at the Dimension line location prompt:

- To edit the text, enter **t** (Text).
- To edit text with the Multiline Text Editor, enter **m** (Mtext).
  In the Multiline Text Editor dialog box, add text in front of or after the angle brackets (<>), which denote the dimension text, or replace the dimension text by selecting the angle brackets and entering new dimension text. Then choose OK.



Examples of single-line and multiline dimension text

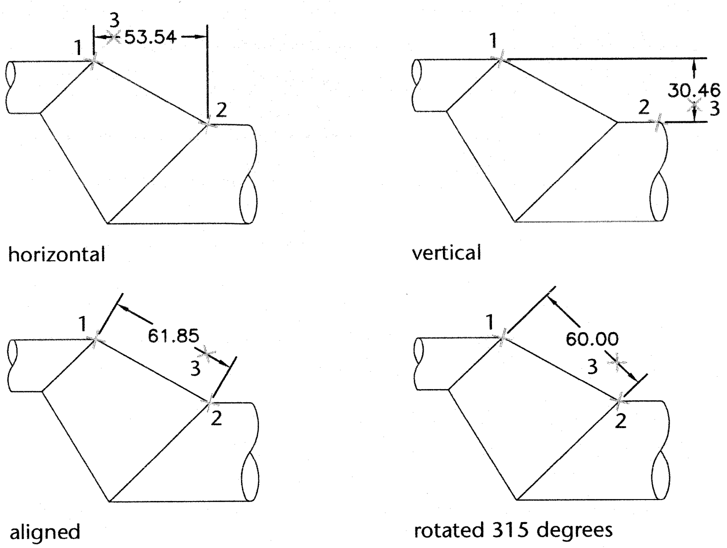
When you edit dimension text, the original text is deleted. To restore the original dimension text, enter <>.

- To rotate the text, enter **a** (Angle). Then enter the text angle.

The dimensions you create use the current dimension style. The default style is STANDARD. If you have created other dimension styles, you can change the style before you start creating the dimension.

Creating Linear Dimensions

Linear dimensions can be horizontal, vertical, aligned, or rotated. Aligned dimensions have the dimension line parallel to the line along which the extension line origins lie.



Examples of linear dimensions

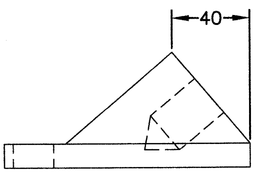
In all four of these illustrations, the extension line origins (1 and 2) are designated explicitly. The resulting dimension line (3) location is also shown.

Baseline (parallel) and continued (chain) dimensions are series of consecutive dimensions that can be based on a linear dimension.

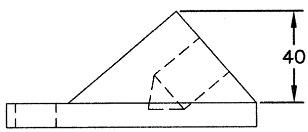
As you create linear dimensions, you can modify the text, the angle of the text, or the angle of the dimension line. You can also place text in any location using the User Defined option in the DDIM Format dialog box.

Creating Horizontal and Vertical Dimensions

AutoCAD LT automatically applies a horizontal or vertical dimension, depending on the extension line origins you specify or the point where you select an object. However, you can specify horizontal or vertical when you create the dimension. For example, in the following illustration, a horizontal dimension is drawn by default unless you specify vertical.



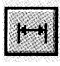
horizontal dimension
created by default



vertical dimension specified

You create horizontal or vertical dimensions by choosing Linear from the Dimension menu. If you want to select the object to dimension, press ENTER, or you can specify the origins of the extension—lines using object snaps, for example.

To create a linear dimension

Dimension Toolbar	Menu	Command
	Dimension ➤ Linear	DIMLINEAR

You can enter any of the following options at the Dimension line location prompt to override the current linear dimension settings:

- To rotate the extension lines, enter **r** (Rotated). Then enter the dimension line angle.

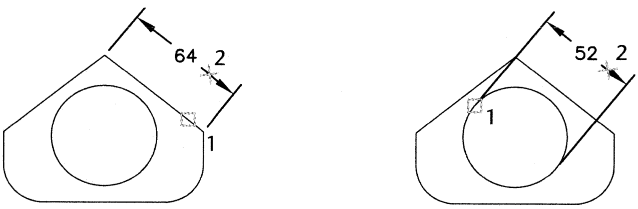
Examples of rotated dimensions

- To override the default dimension orientation, enter **h** (Horizontal) or **v** (Vertical).

Specify the dimension line location to complete the dimension.

Creating Aligned Dimensions

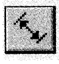
In aligned dimensions, the dimension line is parallel to the extension line origins. The figure shows an example of aligned dimensioning. The object is selected (1), and the location of the aligned dimension is specified (2). The extension lines are drawn automatically.



Examples of aligned dimensions

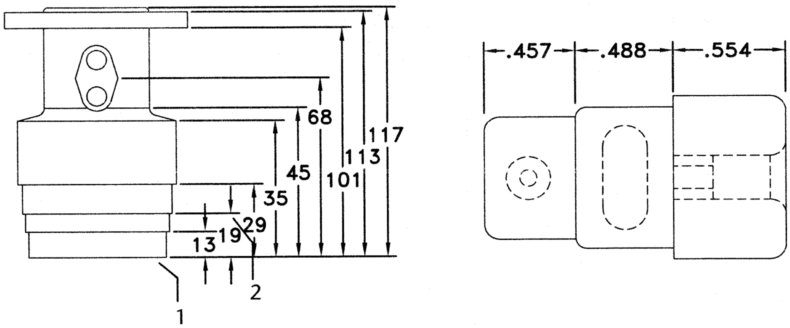
The circle uses the endpoints of the diameter for the origins of extension lines. The point where you select the circle defines one end of the diameter.

To create an aligned dimension

Dimension Toolbar	Menu	Command
	Dimension ➤ Aligned	DIMALIGNED

Creating Baseline Dimensions

Baseline dimensions are multiple dimensions measured from the same baseline. You must create a linear, ordinate, or angular dimension before you create baseline or continued dimensions. You create baseline dimensions from the most recently created dimension in the current session.

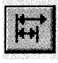


Baseline dimensions

The illustration shows baseline dimensions created from the initial vertical dimension (13 units). This must be created from bottom to top. If the top extension line is created first, the baseline dimension starts from that point.

The additional linear dimensions were created by using the base dimension's first extension line (1) as the origin of the second dimension's first extension line. Using the Endpoint object snap, the end of the section (2) was then selected as the second extension line origin. AutoCAD LT automatically placed the second dimension to the right of the first.

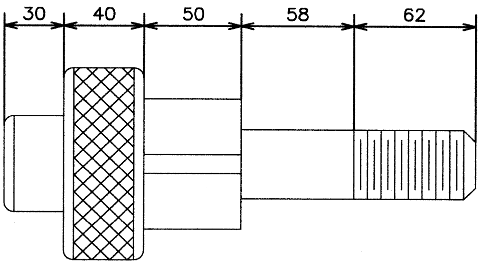
To create a baseline dimension

Dimension Toolbar	Menu	Command
	Dimension ➤ Baseline	DIMBASELINE

Related You can control the spacing of baseline dimension lines by using the Spacing option in the DDIM Geometry dialog box, or by setting the DIMDLI system variable.

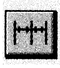
Creating Continued Dimensions

Continued dimensions are multiple dimensions placed end to end. Creating continued dimensions is similar to creating baseline dimensions. The difference is that AutoCAD LT uses the origin of each dimension's *second* extension line as the origin for the next dimension's *first* extension line. The dimensions are aligned horizontally or vertically.

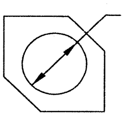


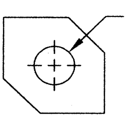
Continued dimensions

To create continued dimensions

Dimension Toolbar	Menu	Command
	Dimension ➤ Continue	DIMCONTINUE

Creating Radial Dimensions



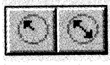


Radial dimensions measure the radii and diameters of arcs and circles with optional centerlines or a center mark.

Center marks and centerlines apply only to diameter and radius dimensions. They are drawn only if the dimension line is placed outside the circle or arc.

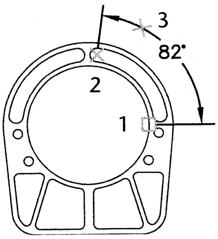
The size of the centerline refers to the length of the centerline segments extending from the center mark.

To create a diameter or radial dimension

Dimension Toolbar	Menu	Command
	Dimension ➤ Radius	DIMRADIUS
	Dimension ➤ Diameter	DIMDIAMETER

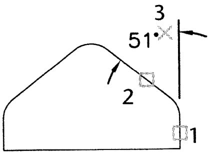
Related The DIMCEN system variable controls whether center marks or lines are drawn and the size of what is drawn. (A negative value means lines are drawn at the size of the absolute value.)

Creating Angular Dimensions



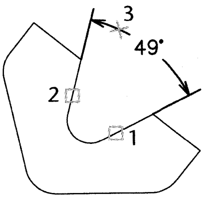
Angular dimensions measure the angle formed by two lines or three points. For example, you can use them to measure the angle between two radii of a circle. The dimension line forms an arc.

To dimension an angle on an arc, circle, or line, you select the object and specify the angle endpoints. You can also dimension an angle by specifying the angle vertex and endpoints. As you create the dimension, you can modify the text content and angle.



The example at the left shows how to dimension the angle by selecting the circle and specifying the angle's endpoints. You select the circle at the first endpoint of the angle (1). Then you specify the second endpoint of the angle (2). Click to specify the location of the dimension line arc (3).

To dimension the angle between two lines, select a line (1), a second line (2), and the location of the dimension line arc (3).



If you use two straight, nonparallel lines to specify an angle, the dimension line arc spans the angle between the two lines. If the arc doesn't meet one or both of the lines being dimensioned, AutoCAD LT draws one or two extension lines to intersect the dimension line arc. The arc is always less than 180 degrees.

If you use an arc or circle or select three points to specify an angle, AutoCAD LT draws the dimension line arc between the extension lines. The extension lines are drawn from the angle endpoints to intersect the dimension line arc.

AutoCAD LT uses the location of the dimension line arc to determine the angle specified by the angle vertex and extension lines.

To create an angular dimension

Dimension Toolbar	Menu	Command
	Dimension ➤ Angular	DIMANGULAR

Related Use the DIMADEC system variable to control the number of places of precision displayed for angular dimensions.

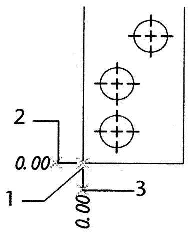
Creating Ordinate Dimensions

Ordinate, or datum, dimensions measure the perpendicular distance from an origin point, called the datum, to a dimensioned feature, such as a hole in a part. These dimensions prevent escalating errors by maintaining accurate offsets of the features from the datum.

Ordinate dimensions consist of an *X* or *Y* ordinate with a leader line. *X*-datum ordinate dimensions measure the distance of a feature from the datum along the *X* axis. *Y*-datum ordinate dimensions measure the same distance along the *Y* axis. If you specify a point, AutoCAD LT automatically determines whether it is an *X*- or *Y*-datum ordinate dimension. This is called an *automatic* ordinate dimension. If the difference between the feature location and the leader endpoint is greater in the *Y* ordinate, the dimension measures the *X* ordinate. Otherwise, it measures the *Y* ordinate. AutoCAD LT uses the origin of the current UCS to determine the measured ordinates. The absolute value of the ordinate is used.

The text is aligned with the ordinate leader line regardless of the text orientation defined by the current dimension style. You can accept the default text or supply your own.

The following procedure shows how to measure the coordinates of the circles' centers in relation to the datum (1) in the figure. In this case, the datum coordinate is (0,0).



To determine the *Y* ordinate of a datum

1. From the Dimension menu, choose Ordinate.
2. At the Select Feature prompt, enter **int** (Intersection object snap).
3. Click your pointing device when AutoCAD LT locates the intersection, then turn on Ortho to ensure straight leaders, and enter **y** to specify the *Y* coordinate value (2).
4. Click to locate the leader endpoint.

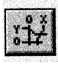
To determine the *X* value of a datum

1. Restart DIMORDINATE by pressing ENTER.
2. Repeat steps 2 and 3, entering **x** to specify the *X* value (3).

You can now use ordinate dimensions to show the relative *X* and *Y* coordinates of the circles' centers. Use the Center object snap to specify the centers.

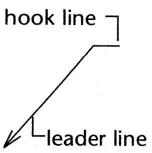
To create ordinate dimensions

1. From the Dimension menu, choose Ordinate.
2. At the Select Feature prompt, enter **cen** (Center object snap), and then select a circle.
3. Specify *X* or *Y* on the command line, and then specify a location for the ordinate leader line.

Dimension Toolbar	Menu	Command
	Dimension ➤ Ordinate	DIMORDINATE

Creating Leaders with Annotation

A leader is a line that connects some annotation to a feature in a drawing. Leaders and their annotation are associative, which means if you modify the annotation, the leader is updated accordingly. Don't confuse this leader with the leader line AutoCAD LT automatically generates as part of a dimension line. See "Fitting Text Using the Fit Options" on page 269.



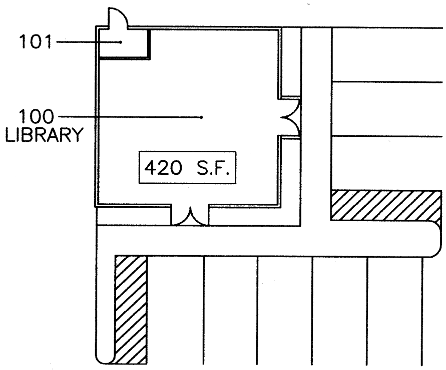
Creating a Leader

You can create a leader from any point or feature in a drawing and control its appearance as you draw it. Leaders can be straight line segments or smooth spline curves. Leader color is controlled by the current dimension line color. Leader scale is controlled by the overall dimension scale set in the current dimension style. The type and size of arrowhead, if one is present, is controlled by the first arrowhead defined in the current style.

A small line known as a hook line usually connects the annotation to the leader line. Hook lines appear with multiline text and feature control frames if the last leader line segment is at an angle greater than 15 degrees from horizontal. The hook line is the length of a single arrowhead. If the leader has no annotation, it has no hook line. You can use the object snaps to specify the start point of a leader accurately.

Adding the Leader Annotation

Leader annotations can be multiline text, a feature control frame, or a block reference. You can create new annotation, or you can append a copy of existing annotation.

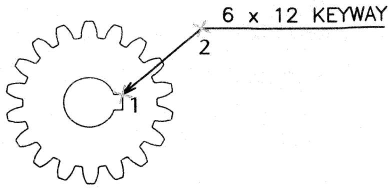


Examples of leader annotation

Text annotation can be entered either on the command line, one line at a time, or in the Multiline Text Editor dialog box. If you enter text on the command line, it is created as *no word-wrap* text (its width is set to zero).

Text or feature control frame annotation is placed at a specified offset from the final endpoint of the leader. You specify this offset in the DDIM Annotation dialog box under Text Gap.

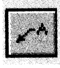
Use the Vertical Justification options in the DDIM Format dialog box to align the annotation vertically with the leader. In the following illustration, the vertical justification is Above.



The simple leader in the example was created by specifying the From (1) and To (2) points of the leader, pressing ENTER, and then entering the leader text.

In addition to simple leader lines with appended text lines, you can also create leaders using splines and use the Multiline Text Editor to create paragraphs of appended text.

To create a leader with annotation

Dimension Toolbar	**Menu**	**Command**
	Dimension ➤ Leader	LEADER

ONLINE HELP See "Creating Text with Leaders." To edit leaders and annotation, see "Changing Text with Leaders."

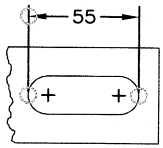
Modifying Dimensions

You can modify dimensions with the AutoCAD LT editing commands and grip editing commands. Grip editing is the quickest and easiest method. You can also modify dimensions by applying dimension style overrides (see "Using Style Overrides" on page 261).

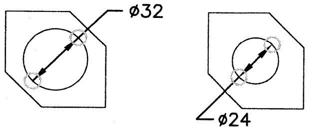
Using Definition Points to Update Dimensions

When you edit dimensioned objects with the editing commands, you must include the relevant dimension *definition points* in the selection set in order for the dimensions to be updated correctly. Definition points are created at the point of attachment to the dimensioned object.

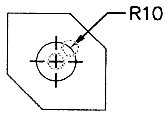
The definition points for each type of dimension are circled in the following illustrations. The middle point of the dimension text is also a definition point for all dimension types.



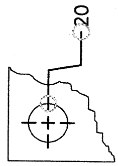
linear—extension line origins and intersection of first extension line and dimension line



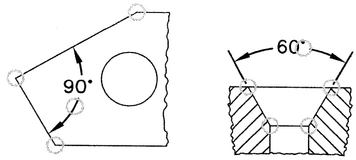
diameter—selection point and opposite point



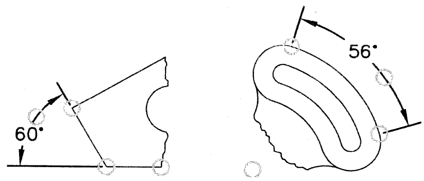
radius—selection point and center



ordinate—UCS origin, feature location, and leader endpoint



three-point angular—angle vertex, extension line origins, and dimension line arc



two-line angular—extension line origins and dimension line arc

Definition points for different dimension types

If no angle vertex is shown, AutoCAD LT places definition points at the ends of the lines that form the angle. In the two-line angular example, because an arc is being dimensioned, AutoCAD LT places a definition point at the center of the arc.

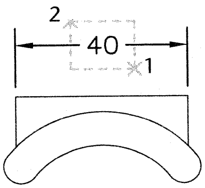
Definition points are drawn on a special layer named DEFPOINTS that is not plotted. If you want to plot definition points, rename the DEFPOINTS layer. AutoCAD LT creates a new DEFPOINTS layer for any subsequent dimensions in your drawing.

Some editing commands change the definition points of the dimension. For example, the definition points of a stretched or rotated dimension change because the object itself has changed. If the dimension is copied or arrayed, the definition points remain the same, because the dimensioned object has not been changed.

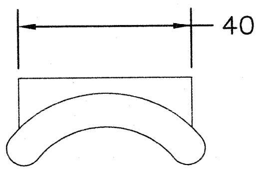
Stretching Dimensions

To stretch a dimension, you must include the appropriate definition point in the selection set, which you can most easily do by turning on grips and selecting the object so the grips are highlighted. For example, the center point of dimension text is a definition point, so you can move the dimension text anywhere as long as you select this point, as shown with the crossing selection here.

If you move the text so that it no longer requires the dimension line to be split, the dimension line rejoins.

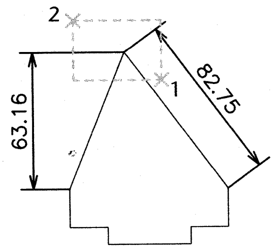


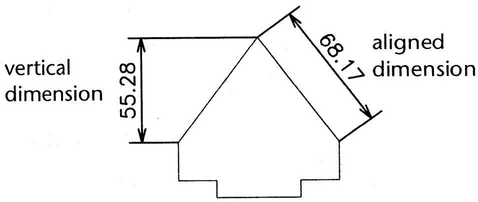
text definition point selected to edit text



dimension line rejoined when no longer split by text

The following illustration shows how to stretch the top vertex of the triangle. You must include definition points for the aligned and vertical dimensions in the crossing selection (1, 2) so that the dimensions are stretched as well.

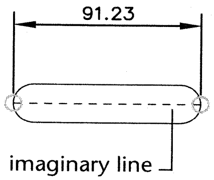




The dimension type (aligned, horizontal, or vertical) is retained after stretching. Notice that the aligned dimension realigns and is recalculated. The vertical dimension, however, still measures the vertical distance from the triangle base to the top vertex.

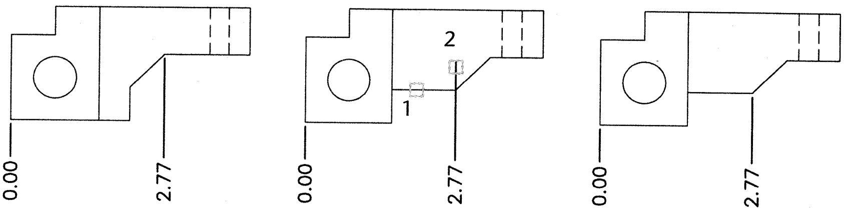
ONLINE HELP See "Stretching Dimensions."

Trimming and Extending Dimensions



As you edit your drawing, you may need to separately adjust some dimensions. You can trim and extend all forms of linear and ordinate dimensions. To trim or extend a linear dimension, AutoCAD LT draws an imaginary line (nonprinting) between the two extension line definition points.

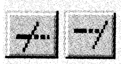
AutoCAD LT trims or extends the imaginary line and then adjusts the dimension accordingly. To trim or extend an ordinate dimension, AutoCAD LT moves the feature location point (2) to the boundary edge perpendicular to the measured ordinate (1), so the ordinate value remains unchanged.



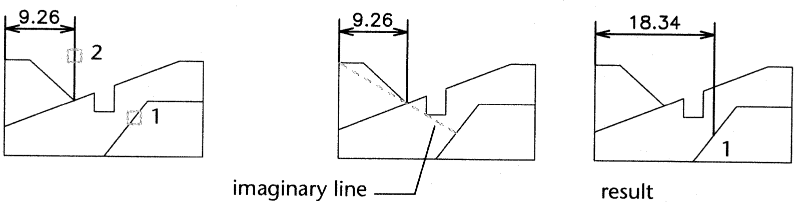
original line and dimension

result of trim

To extend or trim a dimension

Modify Toolbar	Menu	Command
	Modify ➤ Trim Modify ➤ Extend	TRIM or EXTEND

If the edited object is not parallel to the dimension definition points, as shown in the following illustration, the imaginary line technique used by AutoCAD LT probably will not produce the result you want. The following figure illustrates extending a linear dimension by stretching. The dimension is stretched to where the imaginary line meets the selected feature (1).



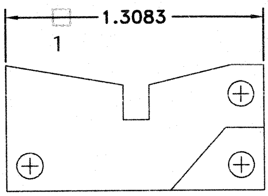
result

In these situations, stretch the dimension rather than trimming or extending it. You cannot trim a dimension and split it in two by cutting a portion out of the middle.

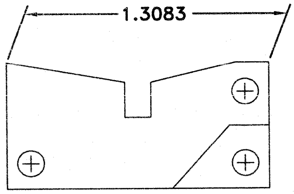
ONLINE HELP See "Trimming and Extending Dimensions."

Making Extension Lines Oblique

AutoCAD LT creates extension lines perpendicular to the dimension line. However, if the extension lines conflict with other objects in a drawing, you can change their angle. New dimensions are *not* affected when you make an existing dimension oblique.

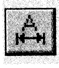


existing dimension selected



result after entering an oblique angle of 70

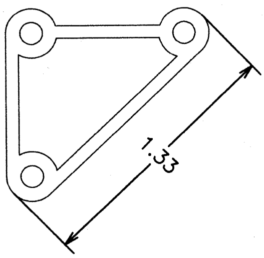
To make extension lines oblique

Dimension Toolbar	Menu	Command
	Dimension ➤ Oblique	DIMEDIT

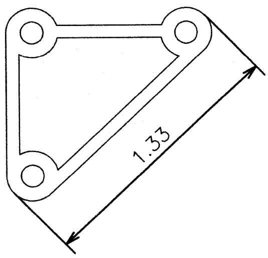
ONLINE HELP See "Creating Dimensions with Oblique Extensions."

Moving and Rotating Dimension Text

Once you've created a dimension, you can rotate the existing text or replace it with new text. You can move the text to a new location or back to its home position, which is the position defined by the current style. In the following illustration, the home position is above and centered horizontally on the dimension line.



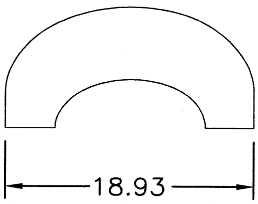
dimension text rotated



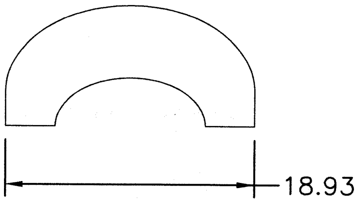
dimension text moved back to home position

When you rotate or replace dimension text, you specify the change first, for example, a rotation angle. You can then select any number of existing dimensions to apply this change to before pressing ENTER to end selection in the usual way. When you move dimension text, you can select a single dimension to apply the move to.

You can move dimension text to the left, right, or center position on the dimension line or to any position inside or outside the extension lines. A quick and simple way to do this is by using grips. If you move text up or down, the current vertical justification of the text relative to the dimension line is not changed, so the dimension and extension lines are modified accordingly. The following illustration shows the result of moving text to the right. The text remains centered vertically on the dimension line.

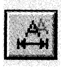


text centered vertically on the dimension line



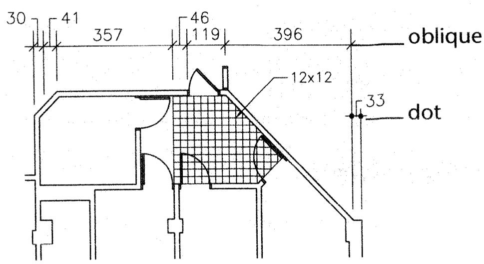
result of moving text to the right and outside the extension lines

To move and rotate dimension text

Dimension Toolbar	Menu	Command
	Dimension ➤ Align Text	DIMTEDIT

Creating Dimension Styles

A named *dimension style* is a group of settings that determine the appearance of the dimension. Using named dimension styles, you can establish and enforce drafting standards for drawings.

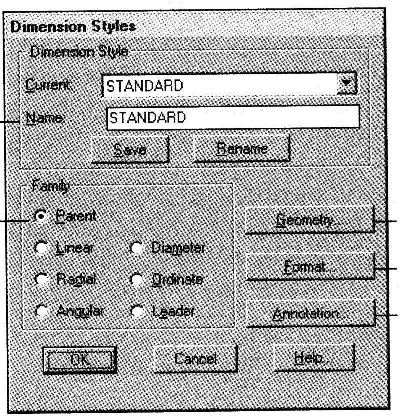


Dimension arrowhead styles

The dimensions you create use the current dimension style. The default style is STANDARD. If you have created other dimension styles, you can change the style before you start creating the dimension.

To set up a *parent* dimension style, you begin by naming and saving a style. The new style is based on the current style and will include all subsequent changes to the layout of the dimension structure (DDIM Geometry dialog box), the positioning of text (DDIM Format dialog box), and the appearance of annotation (DDIM Annotation dialog box). Annotation in this case means primary and alternate units, tolerances, and text.

You create a parent dimension style using the Dimension Styles dialog box.



Enter a style name and click Save.

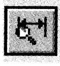
Notice that Parent is selected by default.

Sets dimension structure.

Sets format of dimension text.

Sets appearance of dimension text.

To create a dimension style

Dimension Toolbar	Menu	Command
	Dimension ➤ Style	DDIM

Creating Style Families

Style *families* are groups of styles based on a parent style with variations for different types of dimension. With dimension style families, you can design a dimension style and then specify variations for each dimension type, without having to set up and use an entirely new style.

Dimension style families are related to one another by name. The *child* dimension styles—dimension styles based on a parent style—are automatically named with a numeric suffix that indicates the dimension type, as shown in the following example.

ISO-25	ISO-25 parent dimension style
ISO-25$0	Linear child of ISO-25 (both rotated and aligned types)
ISO-25$2	Angular child of ISO-25 (both 2-line and 3-point types)
ISO-25$3	Diameter child of ISO-25
ISO-25$4	Radial child of ISO-25
ISO-25$6	Ordinate child of ISO-25
ISO-25$7	Leader child of ISO-25 (also used for tolerance objects)

As an example, you may want to force a dimension line for radius and diameter dimensions only. Instead of creating a different style for each dimension type with only this option different, you create the parent style with Force Line Inside (or DIMTOFL) off, then create radius and diameter family members with Force Line Inside on. When dimensioning, you select the dimension base, or parent, style and AutoCAD LT automatically applies the appropriate family member style according to the type of dimension you're creating. In this case, AutoCAD LT forces a dimension line only when you create radial or diameter dimensions.

You can set up family member styles for linear, radial, angular, diameter, and ordinate dimensions and for leaders. Changes to a parent style after it has been created and saved are not applied to family members.

Families provide a set of permanent variations on the parent style. When you create a dimension, AutoCAD LT chooses the appropriate family member of the current style for the dimension you're creating. If there are no family styles defined, the parent style is applied.

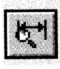
Using Style Families

Style families are used for different reasons than overrides. For example, you could use a style family member for ordinate dimensions to format units appropriately. However, the need to suppress extension lines usually arises in individual cases only and is thus a candidate for the use of overrides.

You can use overrides for local changes to a dimension and for changes to individual variable settings in the dimension style. You can apply overrides to a specific dimension.

NOTE Once you apply an override to a style, the override applies to *all* dimensions in the drawing created when that style was current. For more information on dimension styles, see "Working with Dimension Styles" on page 279.

To create a family member style

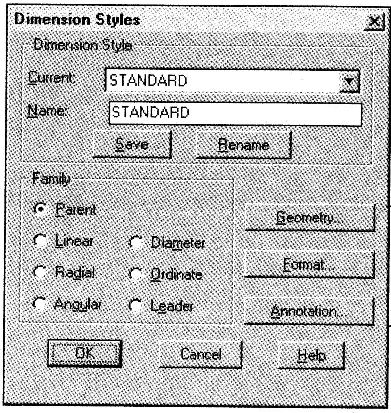
Dimension Toolbar	Menu	Command
	Dimension ➤ Style	DDIM

Using Style Overrides

A *style override* is a change made to an aspect of a dimension style. You can apply overrides to a specific dimension and also to the settings of individual variables in the dimension style. Once you apply an override to a style, the override applies to all dimensions created when that style was current. It's equivalent to changing the status of a dimension variable without changing the current dimension style. For example, if you change the color of extension lines in the DDIM Geometry dialog box, no change occurs in the current dimension style. However, the new value for the changed color is stored in the DIMCLRE system variable. The next dimension you create will have extension lines in the changed color.

Some dimension characteristics are common to a drawing or to a style of dimensioning and are therefore suited to be permanent style settings. Others generally apply on an individual basis and can more effectively be applied as overrides. For example, a drawing usually uses a single type of arrowhead, so it makes sense to define the arrowhead type as part of the style. Suppression of extension lines, however, usually applies in individual cases only and is more suited to a style override.

There are several ways to set up style overrides. You can change the options in the DDIM dialog boxes or the settings of individual dimensioning system variables on the command line. In this case, the style is based on the current style and displayed in the form +STYLENAME, indicating that overrides exist.



You can override the current style by making changes to the current style settings using Geometry, Format, and Annotation options.

The overrides apply to the dimension you are creating and all subsequent dimensions created with that style until you make a new style current.

You can also use DDMODIFY or DIMOVERRIDE on the command line to set up overrides for an existing dimension style. You can use the Apply option of DIMSTYLE to apply overrides to a dimension you select.

You can also set up overrides while you are creating a dimension by entering any dimension variable name at any dimension creation prompt. In this example, the dimension line color is changed in this way. The change will apply to subsequent dimensions until you make another style current.

Command: *Enter* **dimlinear**
First extension line origin or ENTER to select: *Enter* **dimclrd**
Current value < 4 (cyan) > New value: *Enter* **5**
First extension line origin or ENTER to select: *Specify the first extension line origin or select an object to dimension*

ONLINE HELP See "Creating Dimension Styles." To set up style overrides, see "Using Dimension Style Overrides."

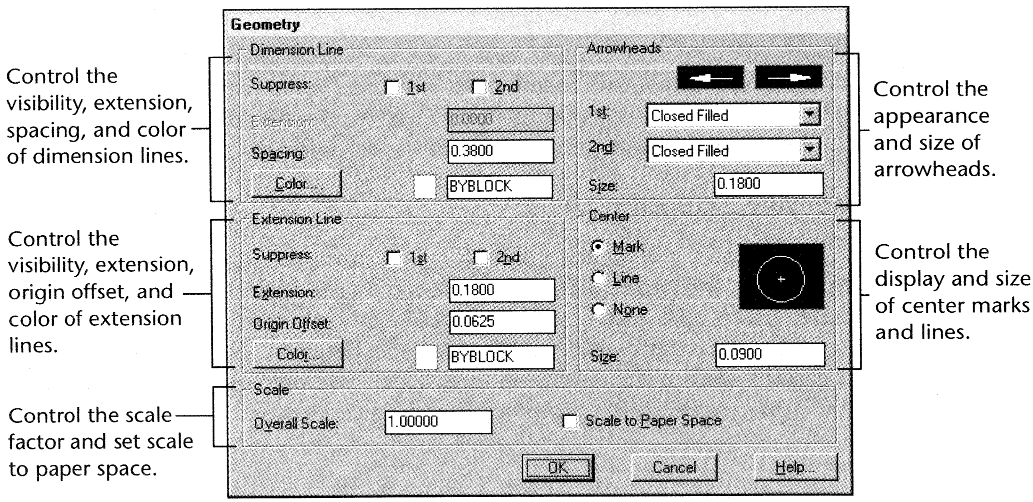
Modifying Dimension Styles

Dimension styles control the layout of the dimension structure and the appearance and positioning of arrowheads and annotation. Annotation includes text as well as measurement units, tolerance values, and symbols. Use the Dimension Styles dialog box to access and modify all of these settings.

Controlling Dimension Geometry

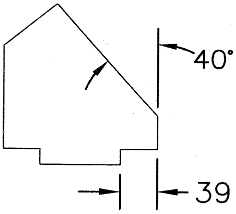
Dimension geometry refers to the extension lines, dimension lines, arrowheads, and center marks or lines. You control the appearance of these elements of the dimension in the DDIM Geometry dialog box.

You can also set the scale of the dimension sizes, distances, and offsets here. This scale setting does not affect tolerances, measured lengths, coordinates, or angles.



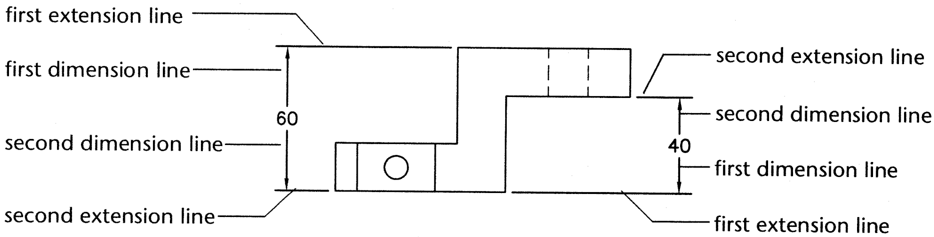
The following section describes the modifications you can make to dimension lines.

Modifying Dimension Lines



You can control several aspects of a dimension line. If you are using oblique stroke arrowheads, you can set the distance by which the dimension line extends beyond the extension lines. For baseline dimensions, you can control the spacing between successive dimension lines. You can also give the dimension line a specific color.

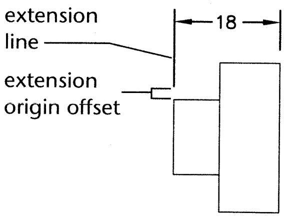
If text divides a dimension line in two, use the Suppress option to control the visibility of each part of the dimension line. The order of dimension lines is determined by the order of extension lines. The first extension line is on the side where you specify the first extension line origin. If the dimension line is split, the first dimension line is on this side too.

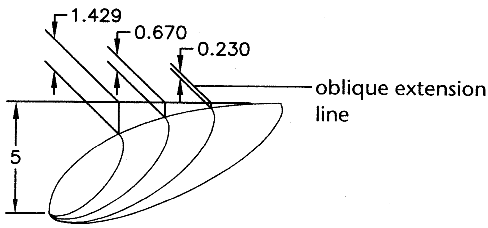


If you create a linear dimension by selecting a line object instead of specifying two extension origins, AutoCAD LT assigns the first extension line to the end of the line object that is nearest to the selection point.

Modifying Extension Lines

You can control several aspects of the extension lines. If extension lines are unnecessary or there is no space for them, you can suppress one or both. Use the Extension option to specify how far beyond the dimension line the extension line extends. Control the distance between the extension line origin and the start of the extension line, which is known as the *extension origin offset*. You can also give extension lines a specific color.





Extension lines are normally perpendicular to the dimension line. However, when space is limited you can make linear dimensions oblique. See "Making Extension Lines Oblique" on page 257.

ONLINE HELP See "Controlling Dimension Geometry."

Setting Dimension Scale

Dimension scale is defined by the Overall Scale value (the DIMSCALE system variable) you set in the DDIM Geometry dialog box. It affects the size of the dimension geometry relative to the drawing, *not* the value of the dimension measurements, and it should be set according to the intended plot scaling. Dimension scale affects sizes, such as text height and arrowhead size, and offsets, such as the extension line origin offset. You should set these sizes and offsets to values that represent their actual plotted size. AutoCAD LT does not apply the overall scale factor to tolerances or measured lengths, coordinates, or angles. To scale these features, see the procedure, "To control dimension annotation style" on page 274.

You can also set a scale relative to paper space. This scale adjusts dimensions in model space to the scale of the paper space view. If you're working directly in paper space (TILEMODE is set to 0), AutoCAD LT uses the default scale factor of 1.0. If you choose to scale in paper space only, AutoCAD LT computes this scale automatically, and the Overall Scale option is not available.

Calculating Dimension Scale

The dimension scale is independent of the drawing scale, although they should be identical. For more information about the drawing scale, see "Determining the Scale Factor" on page 70. Changing the drawing scale might cause dimension text and arrowheads to appear at inappropriate sizes. For example, in a large facilities management drawing, arrowheads in the default size are invisible. Set text height and arrowhead size to the actual sizes you want in the plotted drawing. These values are then multiplied by the overall dimension scale.

You can set scalar dimension variables, such as dimension text and arrowhead sizes, to the actual size that you want in your plotted output. The following table shows the relationship of the dimension scale factor and the plot scale.

Relation between dimension scaling, plot scale, and output text height

Drawing text height (DIMTXT)	Dimension scale factor (DIMSCALE)	Plot scale (plotted units: drawing units)	Plotted text height
0.25	1	1:1	0.25
0.25	1	1:2	0.125
0.25	2	1:2	0.25
0.25	1	2:1	0.5

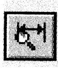
The relationship can be summarized as follows:

$$\text{PlottingHeight} = \text{DIMTXT} \times \frac{\text{PlotUnits}}{\text{DrawingUnits}} \times \text{DIMSCALE}$$

This relationship applies to other scalar dimension variables, such as arrowhead size.

To set the dimension scale factor

1. From the Dimension menu, choose Style.
2. In the Dimension Styles dialog box, choose Geometry.
3. In the Geometry dialog box, enter a value for Overall Scale, and check Scale to Paper Space if you want to adjust dimensions in model space to the scale of the paper space view.

Dimension Toolbar	Menu	Command
	Dimension ➤ Style	DDIM

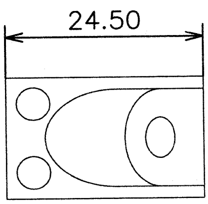
You can draw dimensions in both paper space and model space, but it is recommended that you draw dimensions in model space. If you dimension your drawing in paper space, the paper space dimension does not change when you use editing commands or change the magnification of the display in the model space viewport. However, if you want to dimension in paper space, you need to set the dimension scaling system variable called DIMLFAC.

Using the DIMLFAC dimensioning system variable, you can automatically adjust the dimension length scale factor to the zoom scale factor of a model space viewport. This method does not work for ordinate dimensions.

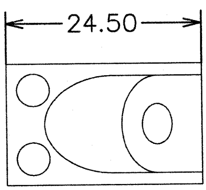
ONLINE HELP See "Setting the Scale for Dimensions."

Controlling Dimension Format

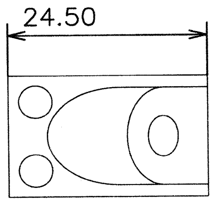
Dimension format refers to placement of dimension text, arrowheads, and leader lines relative to the dimension and extension lines. For example, you can control how text is placed both horizontally and vertically relative to the dimension line and the extension lines. AutoCAD LT provides several justification settings that facilitate compliance with international standards, or you can choose your own location for the text.



text centered above the dimension line



text centered horizontally and vertically



text left-aligned above the dimension line

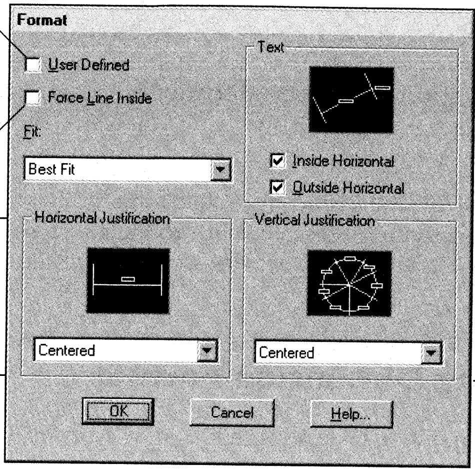
All these settings can be determined by the dimension style. Many of the settings are interdependent. Image tiles in the Format dialog box are updated dynamically to illustrate how text appears as you change the settings.

If you want to change, move, or rotate existing dimension text or return it to its default position according to the current style, use DIMEDIT or DIMTEDIT. See "Modifying Dimensions" on page 253.

To access the controls for dimension format

1 From the Dimension menu, choose Style.

2 In the Dimension Styles dialog box, choose Format.



Fitting Text within Extension Lines

Many factors, such as space between extension lines and arrowhead size, influence how dimension text and arrowheads fit within the extension lines. User-positioned text, including text positioned with grip editing, may also be subject to these factors. In general, AutoCAD LT automatically applies the best fit given the available space. If possible, both text and arrowheads are accommodated between the extension lines, no matter what fit option you choose.

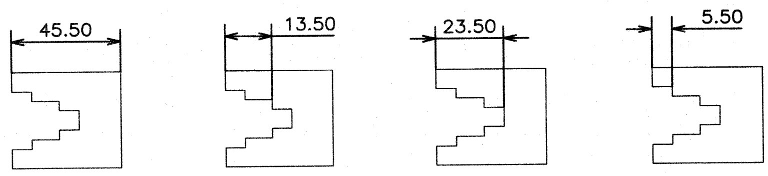
When creating new dimensions, you can place text by entering coordinates or by using the pointing device. This is known as user-defined text placement. Alternatively, you can allow AutoCAD LT to compute the text position. The options for automatic fitting of text and arrowheads are listed under Fit in the Format dialog box:

- ***Text and Arrows*** Specifies that text and arrowheads be kept together. In this case, if there is not room for both between the extension lines, they are both placed outside.
- ***Text Only*** Specifies that if there is room for only text *or* arrowheads, then text only is placed between the extension lines.
- ***Arrows Only*** Specifies that if there is room for only text *or* arrowheads, then arrowheads only are placed between the extension lines.
- ***Best Fit*** Specifies that the size of the text and arrowheads change in order to fit in the space available.

- ***Leader*** Specifies that if there is not room for text between the extension lines, AutoCAD LT creates a leader line automatically. This is useful in cases where text outside the extension lines would clash with other geometry, for example, in continued dimensions.
- ***No Leader*** Specifies that no leader lines are created automatically.

Fitting Text Using the Fit Options

These examples show how AutoCAD LT applies fit options for arrowheads and text.



places text and arrowheads inside

allows only arrowheads inside

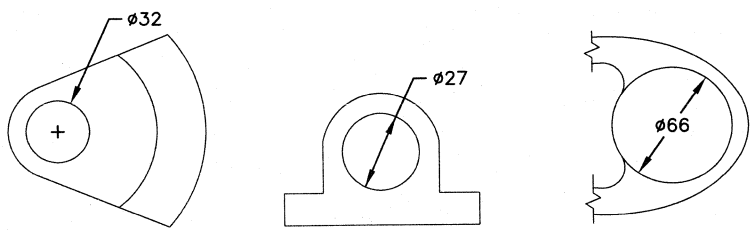
allows only text inside

places text and arrowheads outside

Forcing a Line Inside Extension Lines

Even if the arrowheads are outside the extension lines, you can have a line drawn between the extension lines. This is called *forcing an internal line.*

You can draw several different styles of diameter dimensions depending on the horizontal settings for dimension text and whether you select Force Line Inside.

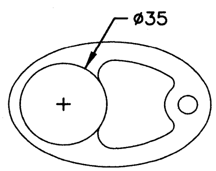


default horizontal placement—text outside circle, center mark, no forced interior line

user-defined horizontal placement—dimension line and arrows, no center mark, forced interior line

Text and Arrows inside with Inside Horizontal option selected

The rightmost previous illustration shows a combination of fit and orientation settings. The Text and Arrows and the Inside Horizontal options are both selected. The resulting text is inside the extension lines and forced to remain horizontal instead of aligning with the dimension line. For a description of orientation settings, see "Controlling Orientation of Text" on page 271.

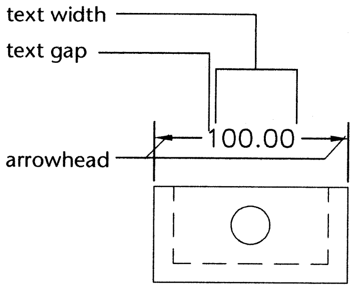


diameter dimension

In the diameter dimension at the left, AutoCAD LT places the text outside the circle and draws a leader line from the text to the circumference of the circle. Because Center Mark is selected, a center mark is drawn inside the circle. In this case, the text is placed horizontally with no forced interior lines for the dimension line.

Fitting Text and Arrowheads by Changing Their Size

You can fit text and arrowheads by changing their size yourself. The figure illustrates the features of the dimension line and text that affect the fit of text and arrowheads.

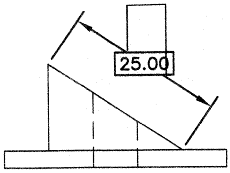


You need room for two arrowheads, two pieces of the dimension line, two text gaps, and the text. Each arrowhead and each piece of dimension line must be at least as large as the text gap. The Oblique and None arrowhead types have zero width and are therefore useful when space is limited. When you place the text above the dimension line, the text gap is not required. A basic formula for determining the minimum width of text centered on the dimension line and within the extension lines is as follows:

minimum width = 2 (overall scale) (text gap + arrowhead size) + effective text width

The effective text width is the size of the intersection of the dimension line with a box surrounding the dimension text, sometimes called a bounding box. The box is the size of the dimension text plus the text gap around that text. This box is identical to the box drawn around what is called a basic dimension. You set the text gap with the Gap option in the Annotation dialog box.

effective
text width

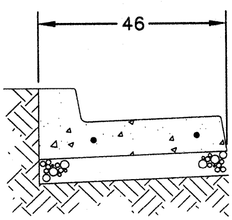


For simple vertical dimensions with no text rotation, the effective text width is twice the text gap plus the text height. The figure illustrates the effective text width when text has been rotated. In this case, basic dimensions have been used to outline the box surrounding the dimension text. Basic dimensions show theoretically exact measurements.

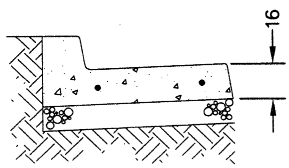
The simplest way to get the text to fit within extension lines is to reduce the size of the text gap or the arrowhead size. You can reduce the overall dimension scale. See "Setting Dimension Scale" on page 265. You can rotate the dimension text so that its orientation is close to perpendicular to the dimension line. Also, you can reduce the length of the text itself by adjusting the appropriate unit settings or by using a text style that specifies a text width factor of less than one. See "Controlling the Appearance of Text" on page 278.

Controlling Orientation of Text

Whether text is inside or outside the extension lines, you can choose whether it is aligned with the dimension line or forced to remain horizontal. The following examples show two combinations of these options.



text inside extension lines
oriented horizontally



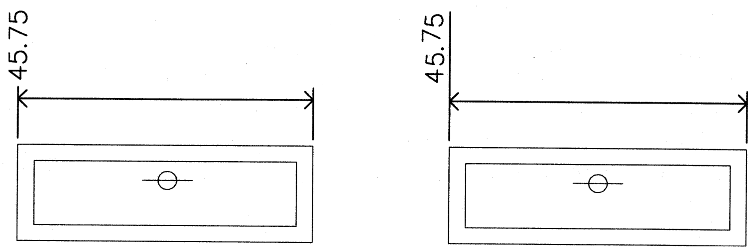
text outside extension lines
aligned with dimension line

The AutoCAD LT default is horizontal dimension text, even for vertical dimensions.

Positioning Text Horizontally

The position of text in relation to the extension lines is referred to as *horizontal justification*. To place text yourself when you create a dimension, use the User Defined option. Use the Horizontal Justification options to automatically place text at the center of the dimension line, at either extension line, or over either extension line. The Vertical Justification settings further determine how the text appears in relation to the dimension line.

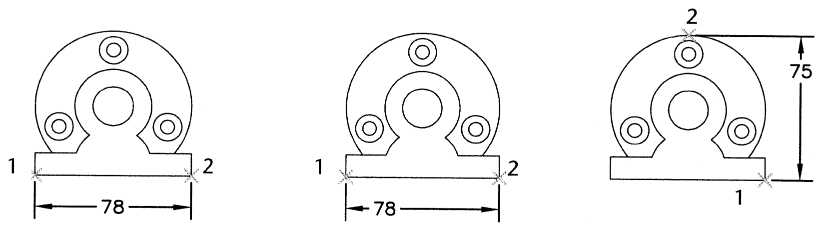
When text is *at* either extension line, it can be either vertical or horizontal. When it is *over* an extension line, it is always aligned with the extension line. For example, the following illustration shows the difference between text placed above the first dimension line when the Centered and Above Vertical Justification options are selected.



horizontal justification—
vertical justification—Centered

horizontal justification—
vertical justification—Above

First and second extension lines are defined by the order you specified the extension line origins. The exception is angular dimensions: their second extension line is counterclockwise from the first. In the next illustrations, the first extension line origin (1) and the second (2) are shown.



text horizontal and centered along dimension line

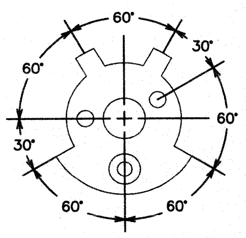
text horizontal at first extension line

text horizontal at second extension line

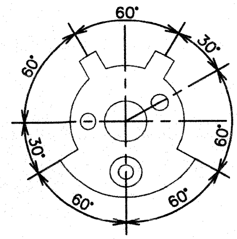
With the User Defined option, you can place the dimension text anywhere along the dimension line, inside or outside the extension lines, as you create the dimension. This option provides more flexibility and is especially useful when space is limited. However, the Horizontal Justification options provide greater accuracy and consistency between dimensions.

Justifying Text Vertically

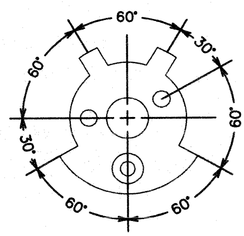
The position of text relative to the dimension line is referred to as *vertical justification*. Text can be placed above, below, or centered within the dimension line. In the ANSI standards, centered text usually splits the dimension line. In the ISO standards, it is usually above or outside the dimension line.



ANSI standard text centered within the dimension line

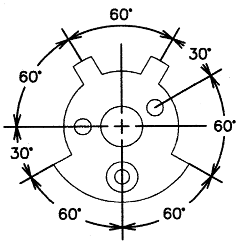


ISO standard text above the dimension line

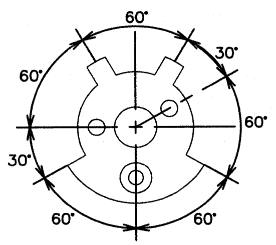


text aligned with and centered within the dimension line

Other settings, such as Inside Horizontal and Outside Horizontal, affect the vertical justification of text. Inside Horizontal affects dimension text within the extension lines; Outside Horizontal affects dimension text outside of the extension lines. For example, if Inside Horizontal is selected, text centered on the dimension line is horizontal, as shown in the ANSI standard illustration, even if the dimension is rotated. If Inside Horizontal is selected, text above and outside the dimension line appears as shown in the following illustrations.



text above dimension line



text outside dimension line

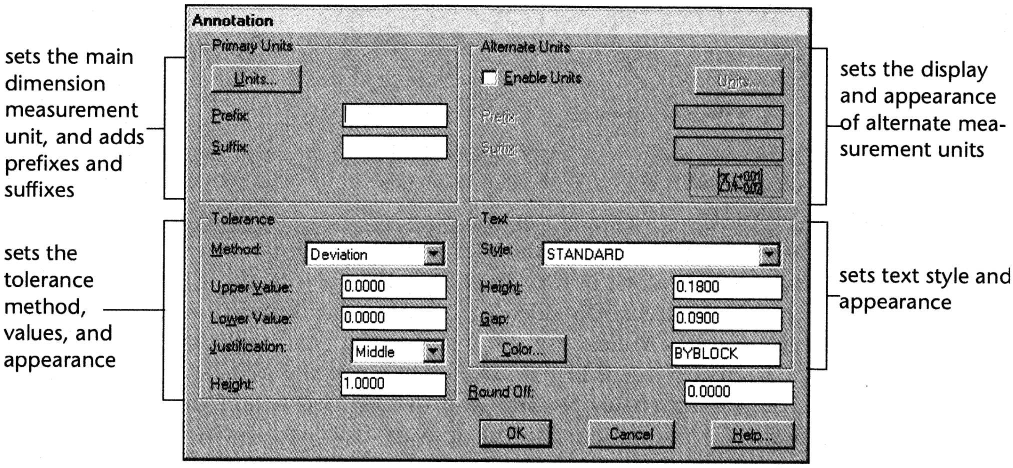
The vertical justification of text when Inside Horizontal is selected

ONLINE HELP See "Controlling Dimension Format."

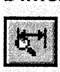
Controlling Dimension Annotation

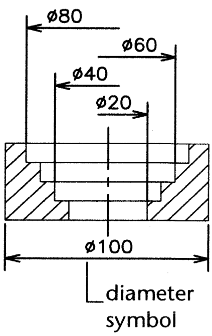
Dimension annotation style establishes defaults for prefixes and suffixes and the display of AutoCAD LT–generated measurements. For example, you could add a diameter symbol as a prefix to a measurement or add the abbreviation for a unit, such as *mm*, as a suffix. Text in this context refers to all dimension text, prefixes and suffixes, primary and alternate units, and lateral tolerances. Geometric tolerances are controlled independently (see "Adding Geometric Tolerances" on page 282).

Dimension annotation is treated as a single string of text, which you create and format using your text editor. You control dimension annotation style using the Annotation dialog box.



To control dimension annotation style

Dimension Toolbar	Menu	Command
	Dimension ➤ Style	DDIM



Adding Primary Units

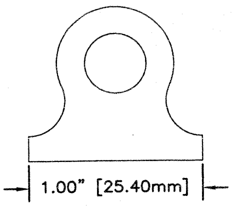
Primary units settings control the main dimension values. For example, you can enter the diameter symbol as a prefix, as shown in the illustration. Any prefix you specify replaces the prefixes normally used for diameter and radial dimensions (∅ and R, respectively).

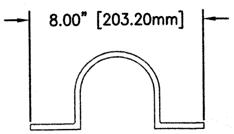
You can set a global measurement scale for linear dimensions or a length-scaling value for dimensions created in paper space (see "Setting Dimension Scale" on page 265).

Adding Alternate Units

You can create dimensions in two systems of measurement simultaneously. A common use of this feature is to add feet and inches dimensions to drawings created in metric units. The alternate units appear in square brackets [ ] in the dimension text. Alternate units cannot be applied to angular dimensions.

If alternate units dimensioning is turned on when you edit a linear dimension, the measurement is multiplied by the alternate scale value specified in the Alternate Units dialog box. This value represents the number of alternate units per current unit of measurement. The default value is 25.4, which is the number of millimeters per inch. The dimension would look like the figure shown at the left if you create a horizontal dimension of 1 inch using architectural units for the primary units and decimal units for the alternate units, with *mm* as the alternate units suffix. The number of decimal places is specified by the alternate units precision value.





Adding Custom Text

In addition to the prefixes and suffixes specified for primary and alternate units, you can supply your own text as you create a dimension. Because the prefix, suffix, and user-supplied text form a single text string, you can represent tolerance stacks and apply changes to font, text size, and other characteristics using the text editor.

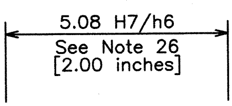
To add custom text above and below the dimension line, use the formatting character \X. Text that precedes this symbol is aligned with and above the dimension line. Text that follows the \X formatting character is aligned with and below the dimension line. The space between the dimension line and the text is determined by the value you enter for Gap in the Annotation dialog box. To add further lines of text above or below the dimension line, use the formatting character \P.

NOTE Formatting characters are case sensitive and must be entered in uppercase.

In the following example, the primary dimension measurement is 5.08, and the alternate dimension measurement is 2.00. The primary units have the suffix *H7/h6*, and the alternate units have the suffix *inches*. At the text prompt, while creating the dimension, you enter the following format string:

<>\XSee Note 26\P[]

The result is displayed as follows:



ONLINE HELP See "Supplying User Text to Dimensions."

Suppressing Zeros in Primary and Alternate Units

If you suppress leading zeros in decimal dimensions, 0.500 becomes .500. If you suppress trailing zeros, 0.500 becomes 0.5. If you suppress *both* leading and trailing zeros, 0.5000 becomes .5 and 0.0000 becomes 0.

If you dimension your drawing using feet and inches, you can suppress the display of zero feet and zero inches. If you suppress feet, any dimension less than a foot shows just the inches portion. For example, 0'-6" becomes 6". If you suppress inches, 1'-0" becomes 1'.

If feet are included with a fractional inch, the number of inches is indicated as zero, no matter which option you select. Thus, a dimension such as 4'-3/4" is edited to 4'-0 3/4".

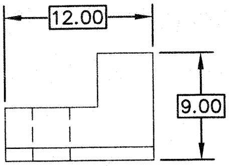
The table shows the effect of selecting each option and provides examples of the architectural units style.

Zero suppression for feet and inches

Selected option	Effect	Examples			
0 Feet and 0 Inches	Suppresses zero feet and zero inches	1/2"	6"	1'	1'-0 3/4"
No Options	Includes zero feet and zero inches	0'-0 1/2"	0'-6"	1'-0"	1'-0 3/4"
0 Inches	Suppresses zero inches (includes zero feet)	0'-0 1/2"	0'-6"	1'	1'-0 3/4"
0 Feet	Suppresses zero feet (includes zero inches)	1/2"	6"	1'-0"	1'-0 3/4"

Creating Lateral Tolerances

A lateral tolerance specifies the amount by which a dimension can vary. By specifying tolerances in manufacturing, you can control the degree of accuracy needed for a feature or some aspect of a part in your drawing. A feature might be a point, line, axis, or surface.



basic dimension

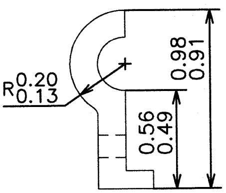
You can apply tolerances directly to a dimension by appending the tolerances to the dimension text. These dimension tolerances show the largest and smallest permissible size of the dimension. You can also apply geometric tolerances, which indicate deviations of form, profile, orientation, location, and runout. See "Adding Geometric Tolerances" on page 282.

Lateral tolerances can be specified from theoretically exact measurements. These are called basic dimensions and have a box drawn around them.

If the dimension value can vary in both directions, the plus and minus values you supply are appended to the dimension value as deviation tolerances. If the deviation tolerance values are equal, AutoCAD LT displays them with a ± symbol and they are known as symmetrical. Otherwise, the plus value goes above the minus value.

deviation tolerance

symmetrical deviation tolerance



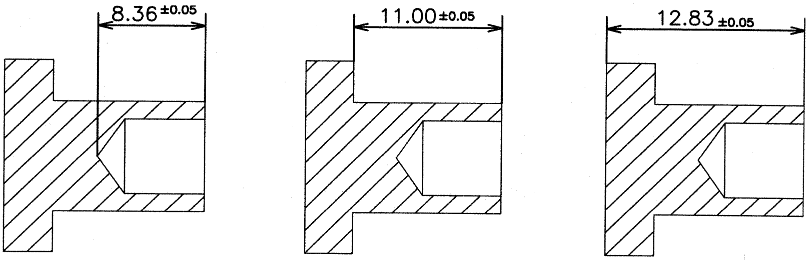
If the tolerances are applied as *limits*, AutoCAD LT uses the plus and minus values you supply to *calculate* a maximum and minimum value. These values replace the dimension value. If you specify limits, the upper limit is displayed above the lower.

To specify limits

1 From the Dimension menu, choose Style.
2 In the Dimension Styles dialog box, choose Annotation.
3 In the Annotation dialog box under Tolerance, select Limits from the Method list.
4 Enter the upper tolerance deviation.
5 Enter the lower tolerance deviation.

Formatting Lateral Tolerances

You can control the vertical placement of tolerance values relative to the main dimension text. Tolerances can align with the top, middle, or bottom of the dimension text.



top middle bottom

You can also control zero suppression, as you can with the primary and alternate units. Suppressing zeros in lateral tolerances has the same effect as suppressing them in the primary and alternate units. If you suppress leading zeros, 0.5 becomes .5, and if you suppress trailing zeros, 0.5000 becomes 0.5.

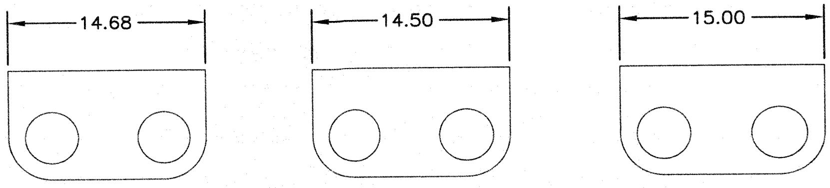
Controlling the Appearance of Text

The appearance of dimension text is governed by the text style selected in the Annotation dialog box. You can choose a text style while creating a dimension style and specify a text color and a height independent of the current text style's height setting. You can also specify the gap between basic dimension text and the box that surrounds it. For information about creating text styles, see "Creating and Modifying Text Styles" on page 218.

The text styles that you can apply to dimension annotation are the same styles you create for all text in your drawing.

Rounding Off Dimensions

You can round off all dimension values. For example, if you specify a round-off value of 0.25, all distances are rounded to the nearest 0.25 unit. The number of digits displayed after the decimal point depends on the precision you set for primary and alternate units and lateral tolerance values.



round-off value set to default (0)

round-off value set to .5

round-off value set to 1

Examples of rounding off the dimension value are set to 2 decimal places

To set dimension value round-off

1 From the Dimension menu, choose Style.
2 In the Dimension Styles dialog box, choose Annotation.
3 In the Annotation dialog box under Round Off, enter the round-off value.

ONLINE HELP See "Controlling Dimension Annotation."

Working with Dimension Styles

You can apply any dimension style to an existing dimension. You can examine styles by comparing the current style settings with the style of a selected dimension or by listing the settings of a specified style's dimension variables. You can also list all the dimension styles in the current drawing.

Restoring and Applying Dimension Styles

When you create a dimension, the current dimension style is associated with that dimension. The dimension retains this style unless you apply a new one or set up overrides. You can restore an existing style by naming it or by selecting a dimension whose style you want to restore. A restored dimension style becomes the current style and is applied to all newly created dimensions.

To restore a dimension style

1 At the Command prompt, enter **dimstyle**.
2 Enter **r** (Restore).
3 Enter the name of the dimension style to restore, or press ENTER to select a dimension whose style you want to restore.

You can also apply the current style, including any overrides, to selected dimensions.

To apply a dimension style to existing dimensions

1 At the Command prompt, enter **dimstyle**.
2 Enter **a** (Apply).
3 Select the dimensions to apply the current style to.

Comparing Dimension Styles

You can compare the current dimension style with another style and see a list of the differences in settings or variables. When you use DIMSTYLE, AutoCAD LT lists any running overrides that apply to the current style.

In this example, you compare the STANDARD dimension style with the current style.

To compare dimension styles

1 At the Command prompt, enter **dimstyle**.
2 Enter **r** (Restore).
3 Enter **~standard**. AutoCAD LT displays a listing of the compared dimension style variables.

Listing Dimension Styles and Variables

You can list all dimensioning system variables and their current settings, or you can list just the variables affected by a dimension style. You can also list all the dimension styles in the current drawing.

To list all dimension variables for the current style

1 At the Command prompt, enter **dimstyle**.
2 Enter **st** (Status).

   The dimensioning system variables are displayed in a scrolling list with their current settings and a brief description of each variable.

ONLINE HELP See "Listing and Comparing Dimension Styles and Variables."

Using Externally Referenced Dimension Styles

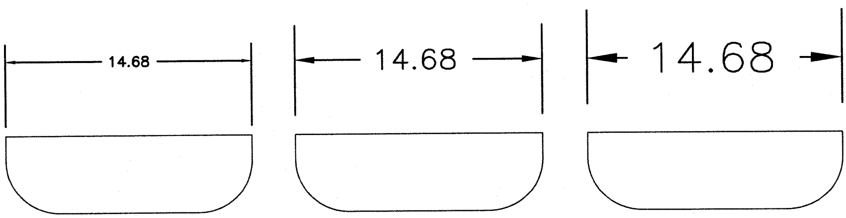
AutoCAD LT displays externally referenced dimension style names using the same syntax as for other externally dependent symbols. The name of the xref is combined with the dimension style name, and the two names are separated by the vertical bar (|) character. For example, if the drawing *baseplat.dwg* has a dimension style called FRACTIONAL-1 and you attach *baseplat.dwg* as an xref to a new drawing, then the dimension style appears in the new drawing as BASEPLAT|FRACTIONAL-1.

Externally referenced dimension styles can be examined, but they cannot be modified or made current. To use an externally referenced dimension style as a template for creating a new style, select the style from the Current list in the Dimension Styles dialog box and enter a new name in the Name box. All settings from the selected dimension style are used for the new style.

Creating and Modifying Arrowheads

The arrowheads AutoCAD LT uses are blocks, or named groups of objects that are saved and reused. To define your own arrowhead, provide the name of an existing block created with WBLOCK or BMAKE. For information on how to create a block, see "Defining Blocks" on page 290.

Arrowhead sizing relies on the overall dimension scale factor. When you create a dimension, AutoCAD LT inserts your block where the arrowheads would normally go. The object's X and Y scale factors are set to *arrowhead size × overall scale.*

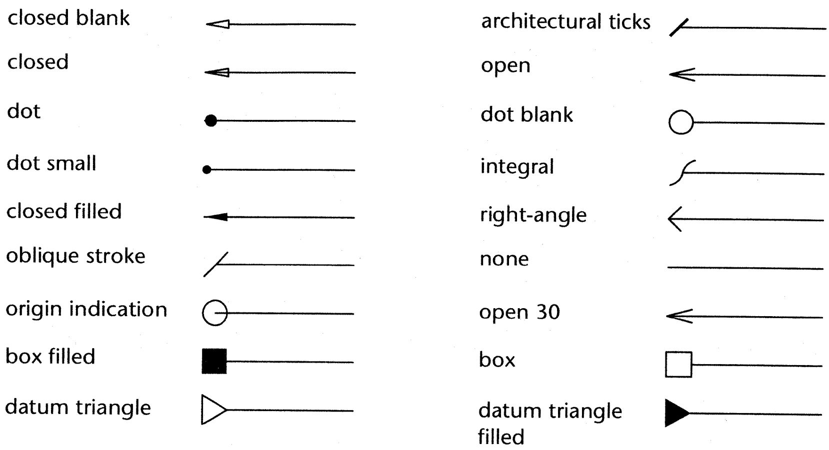


overall scale = 1
arrowhead size = 0.10

overall scale = 2
arrowhead size = 0.10

overall scale = 3
arrowhead size = 0.10

Drafting standards for arrowheads, (termination symbols) differ. AutoCAD LT provides several standard types of arrowheads. You can apply a different type to each end of the dimension line and control the size of each. The first arrowhead is displayed in the *left* image tile and the second in the *right*. The order of arrowheads is determined by the order of extension lines. The first extension line is where you specify the first extension line origin when you are creating the dimension. For angular dimensioning, the second extension line is counterclockwise from the first. Leader lines use the first arrowhead only.



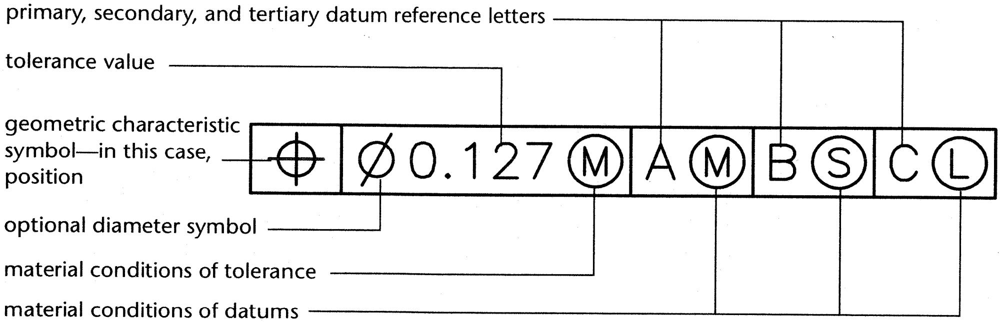
Standard arrowhead types supplied by AutoCAD LT

The first arrowhead type automatically applies to the second arrowhead unless you select a different type for the second arrowhead. You can choose not to use any arrowhead or one arrowhead only. You can also create your own arrowheads.

ONLINE HELP See "Creating and Modifying Custom Arrowheads."

Adding Geometric Tolerances

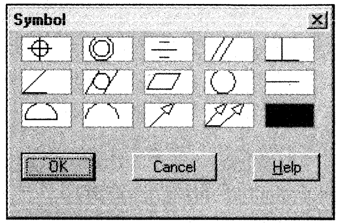
Geometric tolerancing shows deviations of form, profile, orientation, location, and runout of a feature. You add geometric tolerances in feature control frames. These frames contain all the tolerance information for a single dimension.



A feature control frame and its components

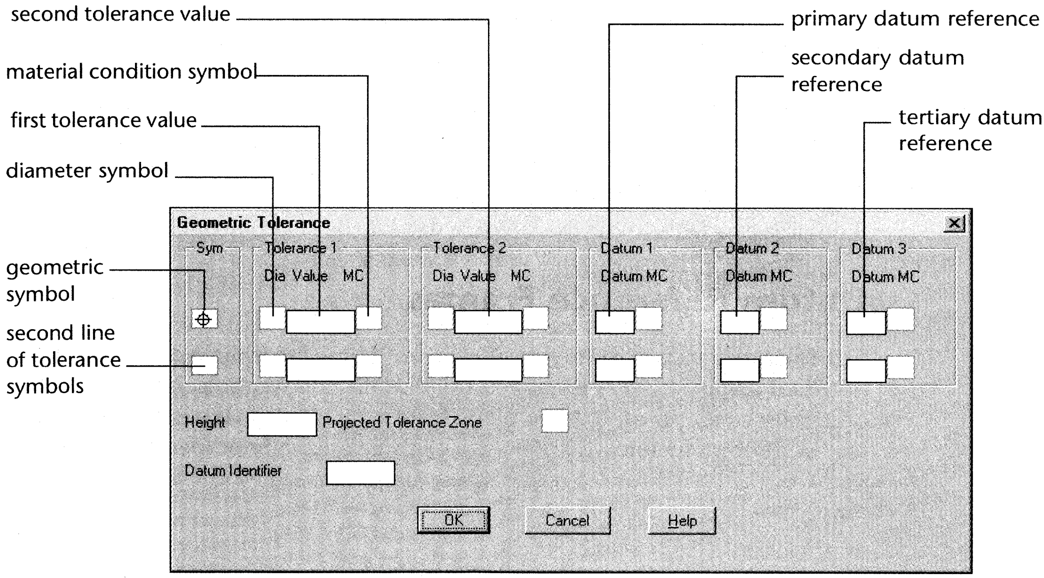
A feature control frame consists of at least two compartments. The first contains a symbol that represents the geometric characteristic to which a tolerance is being applied, for example, form, position, or runout. Form tolerances control straightness, flatness, circularity, cylindricity, and profiles of line and surface. In the illustration, the characteristic is position.

The second compartment contains the tolerance value. Where applicable, the tolerance value is preceded by a diameter symbol and followed by a material condition symbol.



Symbol dialog box showing the available tolerance symbols

Tolerances are set in the Geometric Tolerance dialog box.



To create a geometric tolerance

Dimension Toolbar	Menu	Command
	Dimension ➤ Tolerance	TOLERANCE

ONLINE HELP See "Adding Geometric Tolerances."

To copy, move, erase, stretch, scale, and rotate feature control frames, you can snap to them using object snaps. You can use DDEDIT to edit the tolerance values and symbols or edit them with grips. See "Editing with Grips" on page 151.

To edit a geometric tolerance

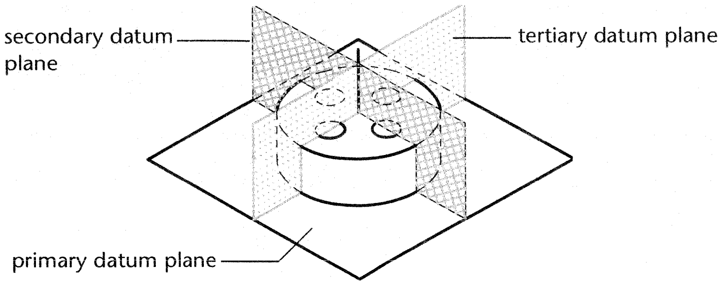
Modify II Toolbar	Menu	Command
	Modify ➤ Object ➤ Text	DDEDIT

Material Conditions

Material conditions apply to features that can vary in size. At maximum material condition (M, also known as MMC), a feature contains the maximum amount of material stated in the limits. At MMC, a hole has minimum diameter, whereas a shaft has maximum diameter. At least material condition (L, also known as LMC), a feature contains the minimum amount of material stated in the limits. At LMC, a hole has maximum diameter, whereas a shaft has minimum diameter. Regardless of Feature Size (S, also known as RFS) means a feature can be any size within the stated limits.

Datum Reference Frames

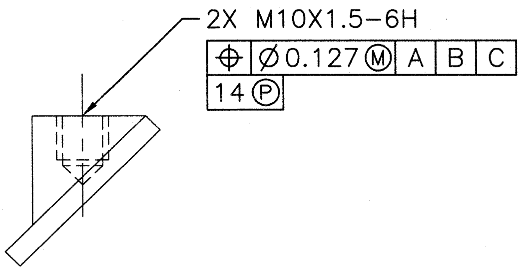
The tolerance values in the feature control frame are followed by up to three optional *datum reference letters* and their modifying symbols. A datum is a theoretically exact point, axis, or plane from which you make measurements and verify dimensions. Usually, two or three mutually perpendicular planes perform this task best. Together, these are jointly called the *datum reference frame.*



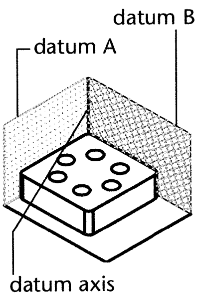
A datum reference frame

Projected Tolerance Zones

Projected tolerances are specified in addition to positional tolerances to make the tolerance more specific. Projected tolerances control, for example, the perpendicularity tolerance zone of an embedded part.



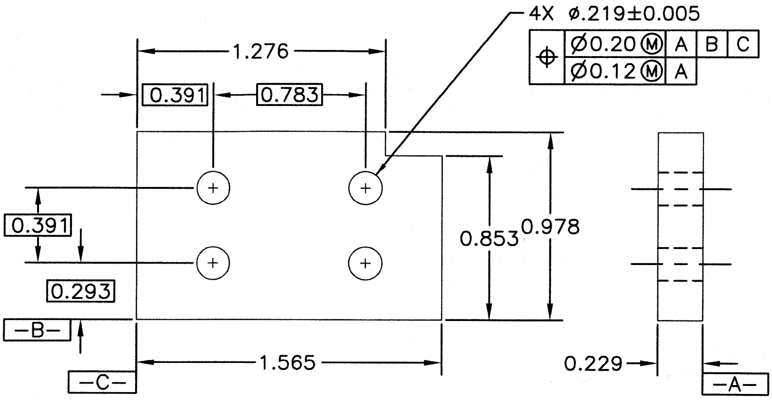
The symbol for projected tolerance (Ⓟ) is preceded by a height value, which specifies the minimum projected tolerance zone. The projected tolerance zone height and symbol appear in a frame below the feature control frame.



Composite Tolerances

A composite tolerance specifies two tolerances for the same geometric characteristic of a feature or features that have different datum requirements. One tolerance relates to a pattern of features and the other tolerance to each feature within the pattern. The individual feature tolerance is more restrictive than the pattern tolerance.

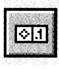
In the illustration, where datums A and B intersect is the *datum axis*, the point from which the position of the pattern is calculated. A composite tolerance could specify both the diameter of the pattern of holes and the diameter of each individual hole.



Example of composite tolerances in a two-line feature control frame

When you add composite tolerances to a drawing, you create the first line of a feature control frame and then choose the same geometric characteristic symbol for the second line of the feature control frame. AutoCAD LT extends the geometric symbol compartment over both lines. You can then create a second line of tolerance symbols.

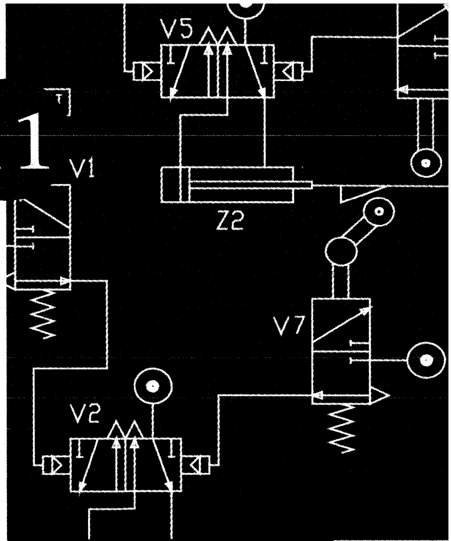
To add composite tolerances

Dimension Toolbar	Menu	Command
	Dimension ➤ Tolerance	TOLERANCE

ONLINE HELP See "Composite Tolerances."

11

Using Blocks, Attributes, and External References



In this chapter

- Defining, inserting, and exploding blocks
- Using blocks with layers, colors, and linetypes
- Creating and editing attributes and extracting attribute information
- Managing external references
- Managing drawings and blocks

AutoCAD LT provides several features to help you manage objects in your drawings. Using blocks, you can organize and manipulate many objects as one component. With attributes, you can associate items of information with the blocks in your drawings. Using external references, or xrefs, you can overlay or attach entire drawings to your current drawing. When you open your current drawing, any changes that have been made in the referenced drawing appear in the current drawing.

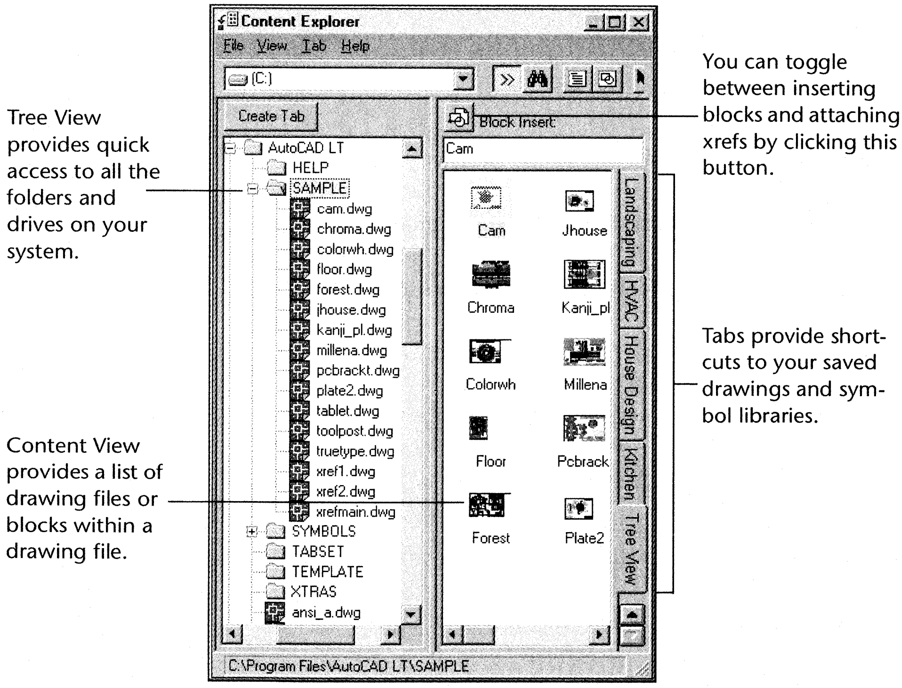
Using the Content Explorer feature, you can now easily locate, insert using drag and drop, and organize drawings and blocks across your system and network drives.

Using Symbols and External References

Most engineering and architectural drawings contain repetitive *symbols*. A symbol is a group of geometric objects combined to represent a physical object. In AutoCAD LT, you create symbols by combining several objects into a single object called a *block*. By including a description when you create a block, you can search for and identify a block based on the text description you supplied when you created it. Reusing blocks makes it easier and faster to get your work done.

You can also use drawing information from other drawings by creating an external reference, or *xref*. All or part of a drawing can be attached to your current drawing. A single master drawing can be referenced by many drawings, which are updated automatically whenever you update the master.

To work efficiently, you need to manage the blocks and drawings you reference. Content Explorer provides tools to make it easier and faster to locate, identify, and insert blocks and attach externally referenced drawings.



Using Content Explorer to insert blocks and attach xrefs is described in "Inserting Blocks" on page 295 and "Attaching Xrefs" on page 308. Detailed information about using Content Explorer to perform searches and to manage drawing files and blocks is described in "Managing Blocks and Xrefs with Content Explorer" on page 320.

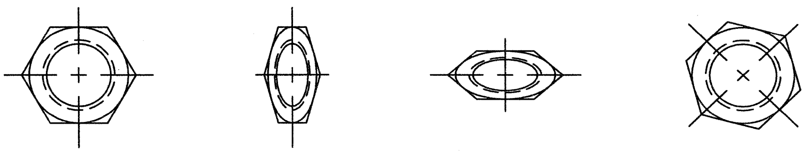
Working with Blocks

A block is a collection of objects you can associate together to form a single object, or *block definition*. In addition to a name for the block definition, AutoCAD LT includes the option to provide a text description, which you can use to search for and locate blocks you created. You can insert, scale, and rotate a block in a drawing. You can explode a block into its component objects, modify them, and redefine the block. If you modify the definition of a block, all references of the block definition in the drawing are updated automatically.

Blocks streamline the drawing process. For example, you can use blocks to:

- Build a standard library of frequently used symbols, components, or standard parts; you can insert the same block numerous times instead of recreating the drawing elements each time.
- Revise drawings efficiently by inserting, relocating, and copying blocks rather than individual geometric objects.
- Save disk space by storing all references to the same block as one block definition in the drawing database.

When you insert a block in your drawing, you are creating a *block instance*. Each time you insert a block instance, you assign a scale factor and rotation angle to the inserted block. You can also scale a block instance using different values in any (*X*, *Y*, *Z*) direction.



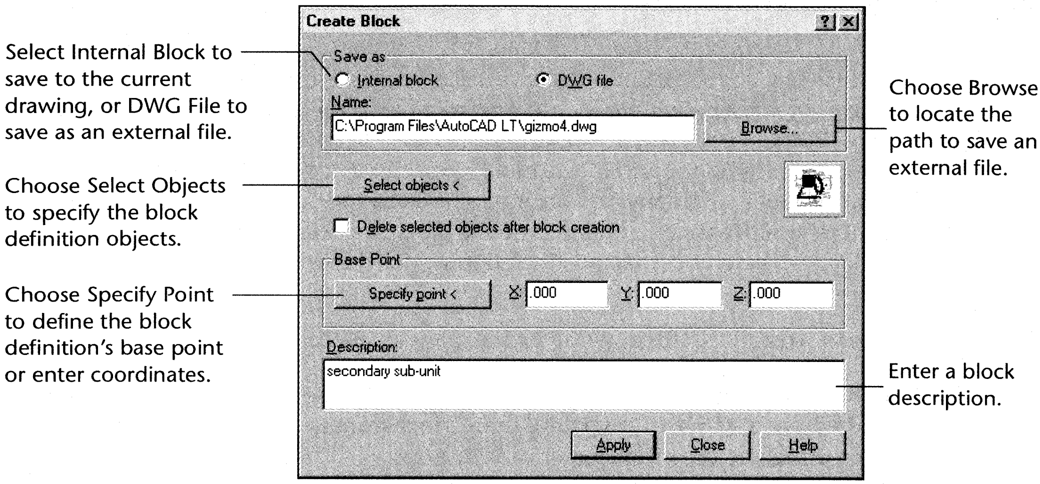
default values | *X* scale = .5, *Y* scale = 1 | *X* scale = 1, *Y* scale = .5 | rotation angle = 45

Blocks make it possible for you to organize your drawing tasks in a systematic way, so that you can set up, redesign, and sort the objects in your drawings and the information associated with them.

Defining Blocks

In AutoCAD LT, you can define a block as a block definition within the current drawing (*internal block*), or as a separate drawing (DWG) file.

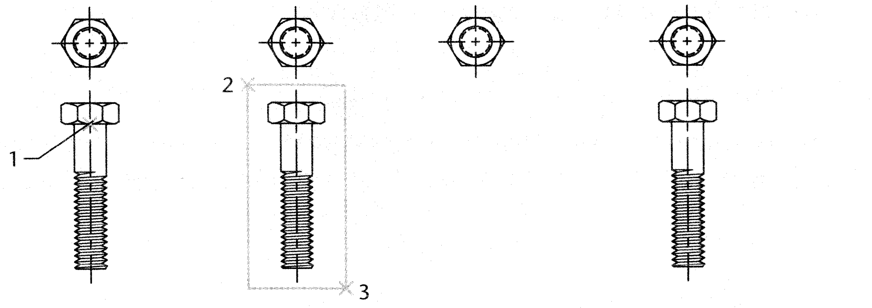
Use the Create Block dialog box to create a block definition.



To create a block definition

Draw Toolbar	Menu	Command
	Draw ➤ Block ➤ Make	BMAKE

The following illustration shows how to create a block definition. You specify a block's insertion base point (1), and then select all the objects to be included in the block definition (2, 3). If the option to remove selected objects from the drawing after block creation is checked, the selected objects are deleted from the drawing.



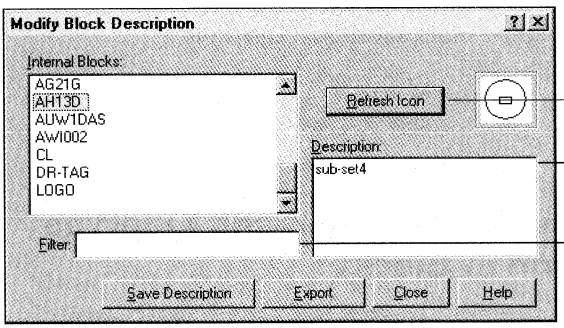
Block definition sequence if the option to remove selected objects from the drawing after block creation is checked

ONLINE HELP See "Creating Blocks."

Modifying a Block Description

After you have created a block definition, you can modify the block description associated with it. You can also rename or delete the block definition.

To modify a block definition's description, or to rename or delete a block definition, use the Modify Block Description dialog box.



To modify a block description

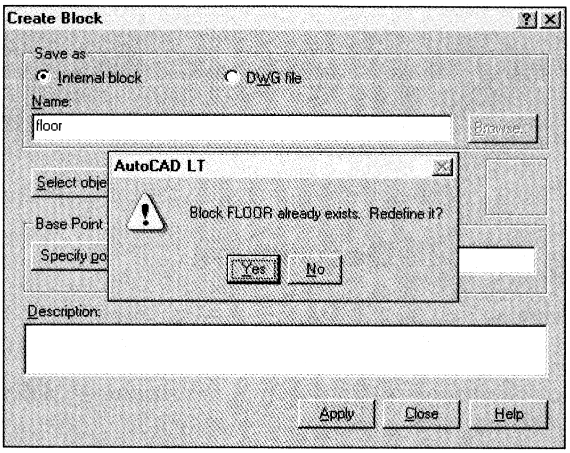
Menu	Command
Modify ➤ Object ➤ Block Description	BMOD

ONLINE HELP See "Modifying Blocks."

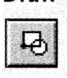
Redefining a Named Block

If you need to update the block definition that you created, you can redefine it using the Create Block Definition dialog box.

When you redefine a block, all the references to that block in the drawing are immediately updated to reflect the new definition. If you use the previous name for a redefined block, a warning dialog box is displayed. Choose Yes to overwrite the previous definition.

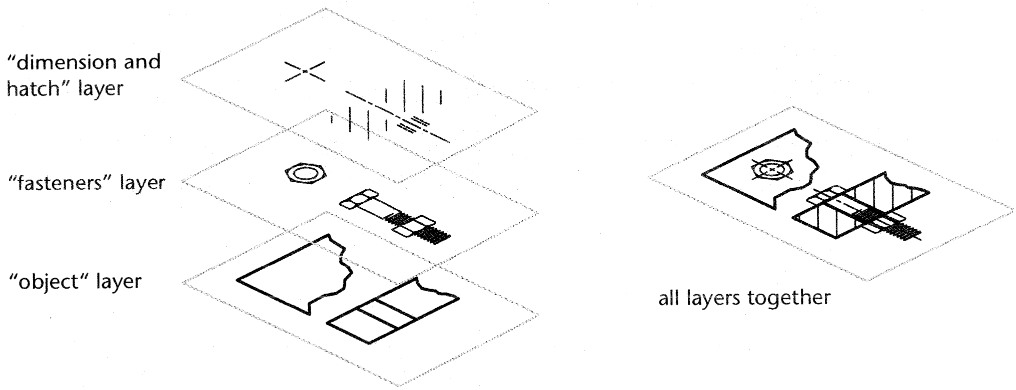


To redefine a named block saved as a DWG file

Draw Toolbar	Menu	Command
	Draw ➤ Block ➤ Make	BMAKE

Controlling Layer, Color, and Linetype Information in Blocks

Blocks can be defined from objects that were originally drawn on different layers with different colors and linetypes. You can preserve the layer, color, and linetype information of objects in a block. When a block that consists of objects that have the layer, color, and linetype properties explicitly assigned is inserted, each object within the block is drawn on its original layer with its original color and linetype.



Objects within a block can be inserted on the layers on which they were originally drawn.

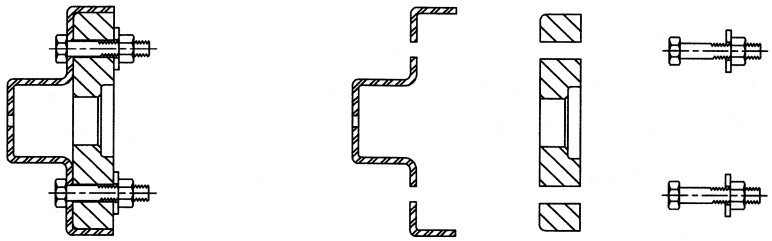
You can also define blocks so that they assume color and linetype properties of the layer on which they are inserted. When a block that consists of objects drawn on layer 0 and assigned the color and linetype BYLAYER is inserted on the current layer, it assumes the color and linetype properties of the current layer. The properties of the layer on which the block is inserted override any color or linetype explicitly assigned to that block.

You can also define blocks that inherit color and linetype properties that you assign explicitly. A block that consists of objects that have color or linetype specified with BYBLOCK is drawn with the color and linetype that are current when the block is inserted. If the color and linetype are not explicitly assigned, the block assumes the color and linetype of the layer.

ONLINE HELP See "Controlling Color and Linetype Properties in Blocks."

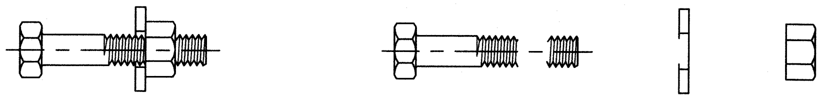
Nesting Blocks

A block reference can contain other (nested) blocks. For example, you can insert a drawing of a mechanical assembly that contains a housing, a bracket, and fasteners, with each fastener composed of a bolt, washer, and nut. The only restriction on nested blocks is that you cannot insert or create blocks that reference themselves.



assembly block

blocks that are components of the assembly block



fastener block

blocks that are components of the fastener block

Nested blocks

Sometimes a nested block contains objects that are on layer 0 or that have color and linetype specified with BYBLOCK. Such objects are called floating objects, and their properties are determined by the block that contains them in the nested structure.

Although block nesting can be useful, floating layers, colors, and linetypes can make nesting complicated if they aren't used correctly. To minimize confusion, follow these guidelines:

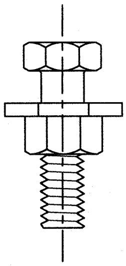
- If all instances of a particular block need the same layer, color, and linetype properties, assign properties explicitly to all objects in the block (including any nested blocks).
- If you want to control the color and linetype of each instance of a particular block by using the color and linetype of the layer on which you insert it, draw each of the block's objects (including any nested blocks) on layer 0 with color and linetype set to BYLAYER.
- If you want to control the color and linetype of each instance of a particular block using the current explicit color and linetype, draw each of its objects (including any nested blocks) with color and linetype set to BYBLOCK. Before creating a block, you can change the layer, color, and linetype of its constituent objects using DDCHPROP.

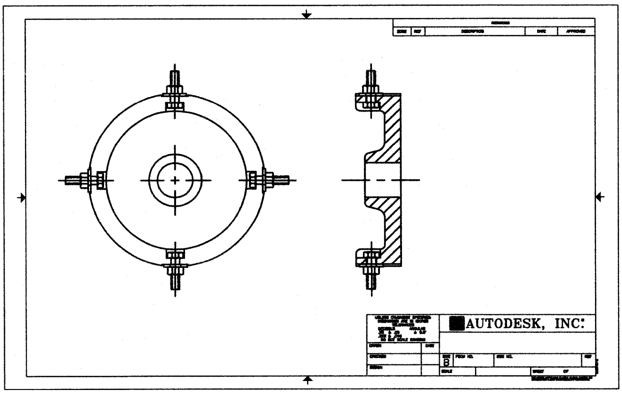
Inserting Blocks

You can insert blocks or entire drawings into the current drawing in three ways:

- Use Content Explorer. Depending on the insertion method, you can specify the insertion point, scale, or rotation settings.
- Use DDINSERT to insert internal blocks or external drawing files. Specify the insertion point, scale, and rotation settings within the Insert dialog box. Use this method if you want to explode the inserted block into its component objects.
- Use INSERT to insert a block from the Command prompt.

When you insert an entire drawing into another drawing instead of attaching it as an external reference, AutoCAD LT treats the inserted drawing like any other block reference. Subsequent insertions reference the block definition (which contains the geometric description of the block) with different insertion point, scale, and rotation settings, as shown in the next illustration.





fastener — block instances of fastener, shown at a reduced scale and rotated

A block and its instances in a drawing

If you change the original drawing after inserting it, the changes have no effect on the inserted block. If you expect the original drawing to change and you want the changes to be reflected in the current drawing, you may want to attach it as an external reference instead of inserting it as a block. See "Using External References" on page 306.

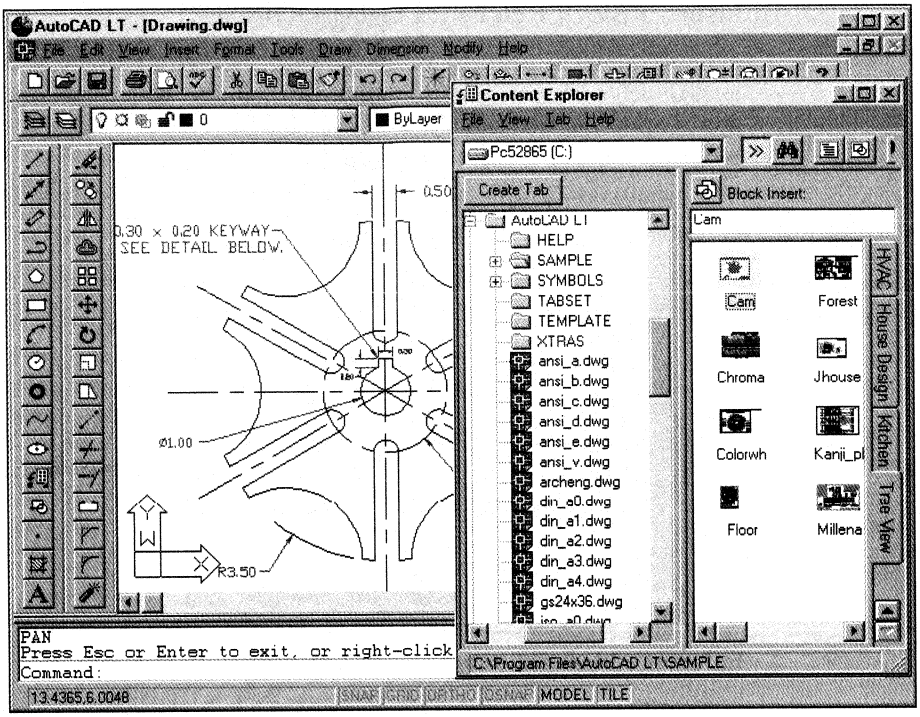
By default, AutoCAD LT uses the coordinate 0,0,0 as the insertion base point for inserted drawings. You can change the base point by opening the original drawing and using BASE to specify a different insertion base point. AutoCAD LT uses the new base point the next time you insert the drawing.

If the drawing you insert contains paper space objects, those objects will *not* be included in the current drawing's block definition. To use the paper space objects in another drawing, open the original drawing and use BLOCK to define the paper space objects as a block. This nested block definition is then included in the drawing database when you insert that drawing into another drawing. You can insert the drawing into either paper space or model space.

Inserting a Block Using Content Explorer

To choose to insert a block, make sure you toggle the Insert Block/Attach Xref button by clicking it. When you have selected the block you want to insert, use either of two methods:

- ***Drag and drop*** Select a block from the list displayed on the current tab, then hold down the pick button on your pointing device and drag the selected block where you want it to be inserted in your drawing. The insertion point of the block reference is specified by the cursor position when you release the pick button on your pointing device. In addition, you can control the insertion point using any snap settings that are active when you drag the block into the drawing. The block is inserted with the default scale (1), and rotation (0) settings. Use this method if you don't want to specify the insertion point using coordinates, change the scale, or rotate the block reference.
- ***Double-click*** When you double-click a block listed in Content Explorer, AutoCAD LT displays prompts on the command line that specify the block's insertion point, scale, and rotation settings. You can also specify the insertion point of the block reference by using your pointing device.



Inserting a block by dragging it into a drawing from Content Explorer

NOTE Multiple blocks can be selected but cannot be inserted into the current drawing.

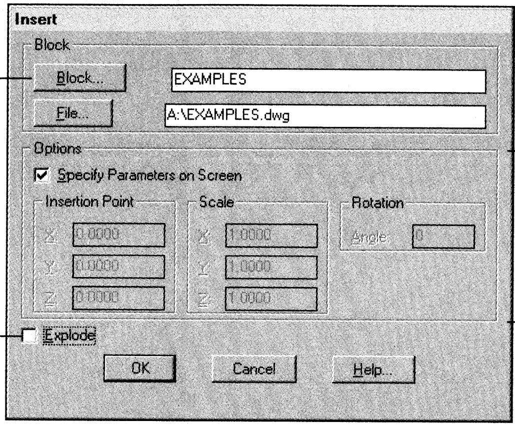
If you want to rescale or rotate the block reference or explode it into object components, use DDINSERT.

To insert a block reference using Content Explorer

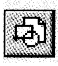
Draw Toolbar	Menu	Command
	Insert ➤ Content Explorer	CONTENT

Insert a Block Reference Using DDINSERT

When you insert a block or drawing using DDINSERT, you specify the insertion point, scale, and rotation angle in the Insert dialog box or on the command line. You can also specify whether you want to explode the block reference into its component objects.



To insert a block reference using DDINSERT

Draw Toolbar	Menu	Command
 Flyout	Insert ➤ Block	DDINSERT

Related If you want the inserted block to reflect the changes you made to the original drawing, you can redefine the block by reinserting the original drawing. This can be done with INSERT by using <blockname>= syntax at the "Block Name" prompt, or with DDINSERT by selecting the original *.dwg* file.

Redefining a Block with DDINSERT

If you have modified the original drawing file for a block, you can redefine the block in the current drawing with DDINSERT. Choose File to specify the file for the block in the Insert dialog box.

Redefinition affects previous as well as future insertions of a block reference. Constant attributes are lost and replaced by any new constant attributes. Variable attributes remain unchanged, even if the new block definition has no attributes. For more information, see "Working with Attributes" on page 300.

ONLINE HELP See "Inserting Blocks."

Inserting Blocks with Long File Names

Because a block name cannot include spaces, when you insert or externally reference a drawing that uses a long file name, specify the block name using quotation marks, as follows:

blockname="filename"

You must use the double quotation marks (""). For example, enter the following to assign *summer home.dwg* to the xref (external reference) *house:*

Xref to attach: **house="summer home.dwg"**

You *cannot* select the file from a file list in a dialog box, because the name of the block definition is taken from the file name and will not include the necessary quotation marks.

If you want to insert a drawing but you don't want the file name to be the same as the block name, or if the file name is greater han 31 characters or contains spaces, you can insert it at the Block Name prompt by entering ***blockname=filename***.

Exploding a Block

Use EXPLODE to disassemble a block into its component objects. By exploding a block instance, you can modify the block or add to or delete the objects that define it.

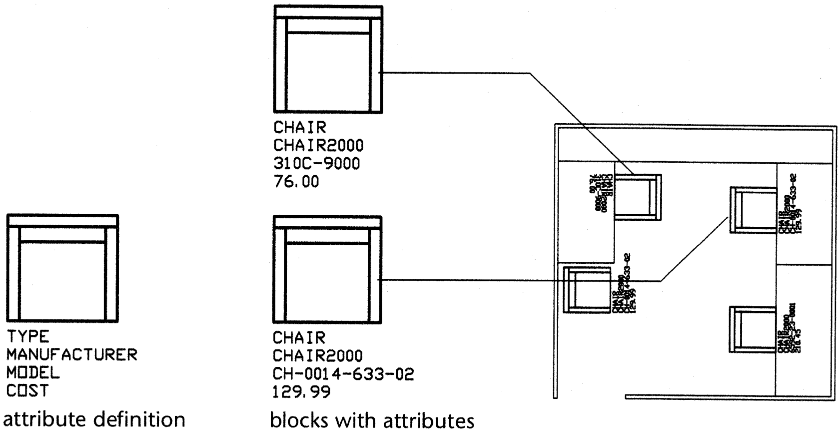
To explode a block, choose Explode from the Modify menu, and then select the block to explode. The block instance is broken into its component objects; however, the block definition still exists in the drawing's block symbol table.

To explode a block into its component objects

Modify Toolbar	Menu	Command
	Modify ➤ Explode	EXPLODE

Working with Attributes

An *attribute* provides an interactive label or tag for you to attach text to a block. You can insert a block with a variable attribute or a constant attribute. Whenever you insert a block that has a variable attribute, AutoCAD LT prompts you to enter the data to be stored with the block. Examples of data are part numbers, prices, comments, and owners' names.



Blocks with attributes

You can extract attribute information from a drawing and use that information in a spreadsheet or database to produce items such as a parts list or bill of materials (BOM). You can associate more than one attribute with a block, provided that each attribute has a different tag. AutoCAD LT prompts you for the value of each attribute when you insert the block.

You can also define constant attributes; because they have the same value in every occurrence of the block, AutoCAD LT does not prompt for a value when you insert the block.

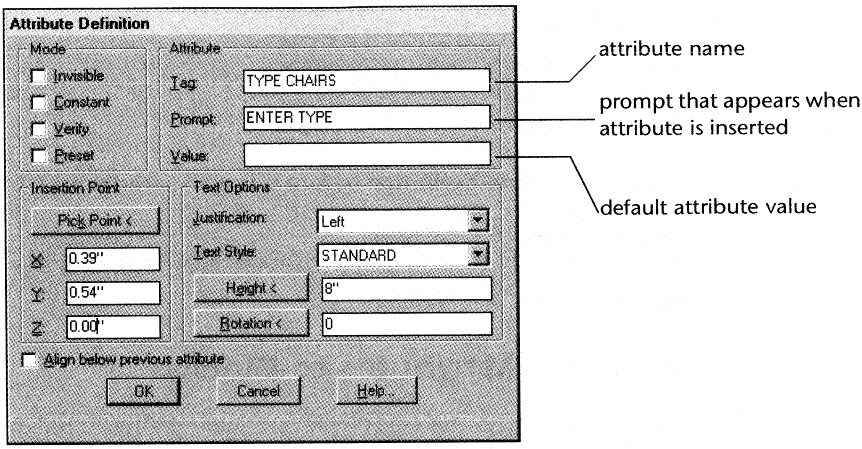
Attributes can be invisible, which means the attribute is not displayed or plotted. However, information on the attribute is stored in the drawing file and written to an extract file by DDATTEXT and ATTEXT.

ONLINE HELP See "Attaching Data to Blocks."

Creating Attributes

To create an attribute, you must first use DDATTDEF to create an attribute definition, which describes the characteristics of the attribute. The characteristics include the tag, the prompt, value information, text formatting, location, and any optional modes. You can then attach one or more attributes to a block when you define or redefine that block.

Open the Attribute Definition dialog box by entering **ddattdef** at the Command prompt. To define an attribute, you specify the insertion point, set the attribute modes and text options, and enter a tag and a prompt.



After creating the attribute definition, you can select it as an object in a block definition. If the attribute definition is incorporated into a block, whenever you insert the block, AutoCAD LT prompts you with the text string you specified for the attribute. Each subsequent instance of the block can have a different value specified for the attribute.

If you want to use several attributes together, define them separately as described above and then include them in the same block. There are additional options for multiple attributes, including order of prompts and alignment.

To define an attribute

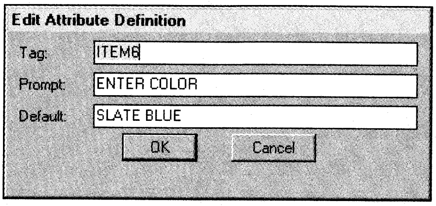
Menu	Command
Draw ➤ Block ➤ Define Attributes	DDATTDEF

Related DDSELECT controls the order of prompts for multiple attributes. ATTDEF creates the attribute definition on the command line. The AFLAGS system variable also sets the DDATTDEF mode. ATTDISP globally controls the visibility of attributes.

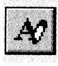
ONLINE HELP See "Defining Block Attributes."

Editing Attribute Definitions

You can use DDEDIT to edit an attribute definition before it is associated with a block.



To edit an attribute definition before it is associated with a block

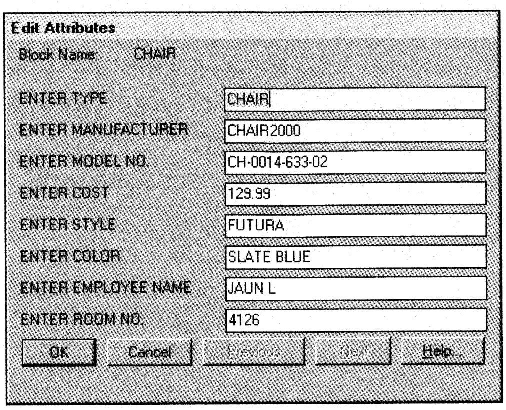
Modify II Toolbar	Menu	Command
	Modify ➤ Object ➤ Text	DDEDIT

Attaching Attributes to Blocks

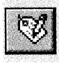
You can attach attributes to a block when you define or redefine that block. When AutoCAD LT prompts you to select the objects to include in the block definition, include the attributes you want in the selection set. The order in which you select the attributes determines the order in which you are prompted for attribute information when you insert the block (DDSELECT). See "Defining Blocks" on page 290.

Editing Attributes Attached to Blocks

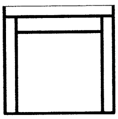
To edit attributes that are already attached to a block and inserted in a drawing, you use the Edit Attributes dialog box.



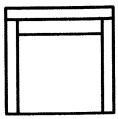
To edit an attribute already attached to a block

Modify II Toolbar	Menu	Command
	Modify ➤ Object ➤ Attribute ➤ Single	DDATTE

Extracting Attribute Information



You can extract attribute information from a drawing and create a separate text file for use with database or spreadsheet software. Extracting attribute information does not affect the drawing. You must create a template file so AutoCAD LT can generate a query report containing the extracted attribute information. The template file contains all of the information associated with attribute tags such as type, manufacturer, model, or cost, as shown in the table. After you create a template file, AutoCAD LT uses that file to determine what attribute information to extract from the drawing.



Template file information

Attribute tag	(C)haracter or (N)umeric data	Maximum field length	Decimal places
Type	C	040	000
Manufacturer	C	006	000
Model	C	015	000
Cost	N	005	003

Each field in the template file extracts information from the drawing. Each line in the template file specifies one field to be written in the extract file, including the name of the field, its character width, and its numerical precision. In the extract file, each record of the extract file includes all the specified fields in the order given by the template file.

The following template file displays the 15 possible fields.

BL: LEVEL	Nwww000	(*Block nesting level*)
BL: NAME	Cwww000	(*Block name*)
BL: X	Nwwwddd	(*X coordinate value of block insertion point*)
BL: Y	Nwwwddd	(*Y coordinate value*)
BL: Z	Nwwwddd	(*Z coordinate value*)
BL: NUMBER	Nwww000	(*Block counter; same for* MINSERT)
BL: HANDLE	Cwww000	(*Block handle; same for* MINSERT)
BL: LAYER	Cwww000	(*Block insertion layer name*)
BL: ORIENT	Nwwwddd	(*Block rotation angle*)
BL: XSCALE	Nwwwddd	(*X scale factor*)
BL: YSCALE	Nwwwddd	(*Y scale factor*)
BL: ZSCALE	Nwwwddd	(*Z scale factor*)
BL: XEXTRUDE	Nwwwddd	(*X component of block's extrusion direction*)
BL: YEXTRUDE	Nwwwddd	(*Y component*)
BL: ZEXTRUDE	Nwwwddd	(*Z component*)
numeric	Nwwwddd	(*Numeric attribute tag*)
character	Cwww000	(*Character attribute tag*)

The template file can include any or all of the BL:*xxx* field names listed above. The template file must include at least one attribute tag field. The attribute tag fields determine which attributes, hence which blocks, are included in the extract file. If a block contains some, but not all, of the specified attributes, the values for the absent ones are filled in with blanks (if characters) or zeros (if numeric). Block references that do *not* contain any of the specified attributes are excluded from the extract file. Each field can appear no more than once in the template file.

The comment fields should not be included in the template file.

To create a template file

1 Click Start on the taskbar, and then choose Programs.

2 From the Accessories program group, choose Notepad.

You can use any text editor or word processor that can save a text file in ASCII format.

3 Enter template information in NotePad and save with a *.txt* extension.

4 To extract data about a specific tag, insert the tag name in place of the "numeric" or "character" fields.

WARNING Do not use tab characters when constructing the template file with a word processor. To align the columns, use ordinary spaces: use SPACEBAR, not TAB.

An attribute is extracted only if its tag name matches the field name specified in the template file. DDATTEXT extracts the attribute information using one of the following formats:

- Comma-delimited file (CDF)
- Space-delimited file (SDF)
- Drawing interchange file (DXF)

The CDF format produces a file containing one record for each block reference in the drawing. A comma separates the fields of each record, and single quotation marks enclose the character fields. Some database applications can read this format directly.

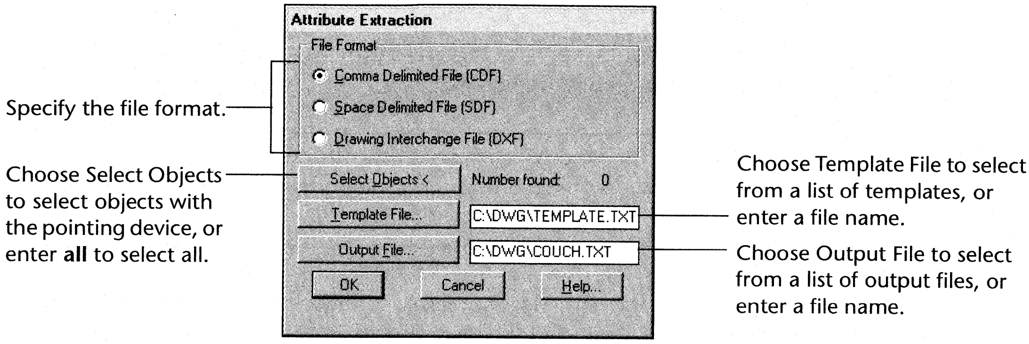
The SDF format also produces a file containing one record for each block reference in a drawing. The fields of each record have a fixed width and employ neither field separators nor character-string delimiters.

The DXF format produces a subset of the AutoCAD LT drawing interchange file format containing only block references, attributes, and end-of-sequence objects. DXF format extraction requires no template. The file extension *.dxx* distinguishes the output file from normal DXF files.

The following is a sample template file.

BL: NAME	C008000	*(Block name, 8 characters)*
BL: X	N007001	*(X coordinate value, format nnnnn.d)*
BL: Y	N007001	*(Y coordinate value, format nnnnn.d)*
SUPPLIER	C016000	*(Manufacturer's name, 16 characters)*
MODEL	C009002	*(Model number, 9 characters)*
PRICE	N009002	*(Unit price, format nnnnn.dd)*

Using the previous template file information, you can extract attribute information with DDATTEXT.



To extract attribute information

Command
DDATTEXT

ONLINE HELP See "Extracting Block Attribute Data."

Using External References

An external reference, or xref, links another drawing to the current drawing. When you insert a drawing as a block, the block definition and all of the associated geometry is stored in the current drawing database. It is *not* updated if the original drawing changes. When you insert a drawing as an xref, however, the current drawing is updated when the referenced drawing changes. A drawing that contains xrefs, therefore, always reflects the most current editing of each externally referenced file.

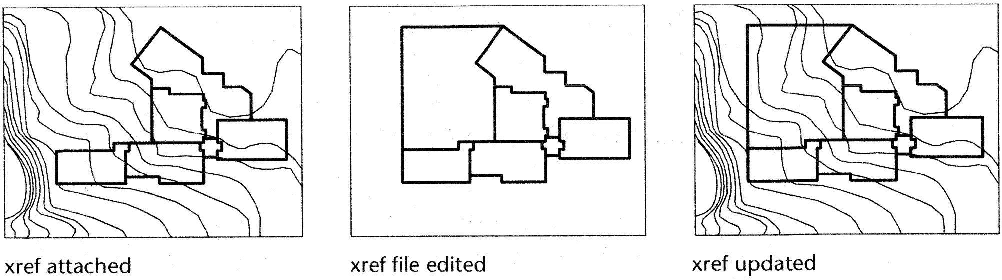
Like a block reference, an xref is displayed in the current drawing as a single object. However, an xref does not significantly increase the file size of the current drawing and cannot be exploded. As with block references, you can nest xrefs that are attached to your drawing.

By attaching xrefs, you can

- Assemble a master drawing from component drawings that may undergo changes as a project develops.
- Coordinate your work with the work of others by overlaying other images on your drawing to keep up with the changes being made by other users.
- Ensure that the most recent version of the referenced drawing is displayed.

Updating Xrefs

Whenever you open or plot your drawing, AutoCAD LT reloads each xref to reflect the latest state of the referenced drawing. After you make changes to an externally referenced drawing and save the file, other users can access your changes immediately by reloading the xref.



Updating of an xref

Demand Loading and Maximizing Xref Performance

Through a combination of demand loading and saving drawings with indexes, you can increase the performance of drawings that contain external references. Demand loading works in conjunction with the XLOADCTL and INDEXCTL system variables. When you turn on demand loading, if indexes have been saved in the referenced drawings, AutoCAD LT loads into memory only the data from the reference drawing that is necessary to regenerate the current drawing. In other words, referenced material is read in "on demand."

To realize the maximum benefits of demand loading, you first need to save the referenced drawings with a layer index. The performance benefits of demand loading are most noticeable when you freeze many layers of the xref and the externally referenced drawing is saved with a layer index. If demand loading is on, only the objects on thawed layers are read into the current drawing as required.

When demand loading is on, AutoCAD LT places a lock on all reference drawings so that it can read in any geometry it needs to on demand. Other users can open those reference drawings, but they cannot save changes to them.

When you turn off demand loading, AutoCAD LT reads in the entire reference drawing, regardless of layer visibility.

Using Demand Loading with Copy

When you want other users to be able to modify an xref that is being demand loaded into another drawing, select the Demand Loading with Copy option. If you turn on Demand Loading with Copy, AutoCAD LT makes a temporary copy of the externally referenced file and demand loads the temporary file. You can then demand load the xref while allowing the original reference drawing to be available for modification. XLOADPATH is the system variable that controls the location of the temporary copy of the externally referenced file.

Using Demand Loading with AutoCAD Release 14 Drawings

In addition to the performance increase afforded by freezing layers in xrefs, AutoCAD Release 14 creates clipped boundaries of xrefs, to display only a specific section of the xref file in the master drawing. AutoCAD LT can take advantage of this additional feature and display clipped xrefs in drawings that were saved in AutoCAD Release 14. Clipped xrefs are referenced by a spatial index saved in the externally referenced drawing. Adjust the system variable INDEXCTL to display clipped xrefs in AutoCAD Release 14 drawings.

To turn on demand loading for xrefs

Menu	Command
Tools ➤ Preferences	PREFERENCES

Attaching Xrefs

Using xrefs is an efficient way to manage drawings because the xref geometry is not saved in the host drawing. Instead, it is referenced and loaded when the host drawing is opened, which reduces the file size of the host drawing. When used with demand loading, using xrefs instead of blocks can greatly increase drawing performance.

Attached xrefs are designed to help you build drawings using other drawings. By attaching drawings as xrefs, as opposed to inserting drawing files as blocks, you can display changes to the externally referenced drawing in the host drawing each time it is opened; the host drawing always reflects the latest revisions made to the referenced files.

Xrefs can be nested within other xrefs. You can attach as many copies of an xref as you want, and each can have a different position, scale, and rotation. You can also control the dependent layers and linetype properties that are defined in that xref.

The name of an xref should be kept as short as possible, due to the symbol table length name restriction of 31 characters. When an xref is attached, all of its dependent symbols (layers, linetypes, dimension styles, blocks, and text styles) are added to the current drawing's symbol table prefixed with the xref's name and the pipe (|) character. For example, if an xref has the name THISISALONGFILE that contains a layer named THISISALONGLAYER, the attachment operation will be disallowed, because the new xref-dependent layer name of THISISALONGFILE|THISISALONGLAYER exceeds 31 characters.

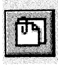
Attaching links one drawing (the reference file, or xref) to the current drawing. Only the definition of the xref is attached to the drawing. AutoCAD LT must read the reference drawing to determine what to display in the current drawing. If the reference file is missing or corrupt, its data is not displayed in the current drawing. All objects and xref-dependent symbol table information is loaded from the referenced file each time a drawing that contains the referenced file is opened. If the VISRETAIN system variable is on, AutoCAD LT stores locally any xref-dependent layer information that has been updated in the current drawing.

ONLINE HELP See "Resolving Name Conflicts in Referenced Drawings."

There are two ways you can attach an xref:

- Insert a drawing file using the External Reference dialog box. If you want to reference a drawing as an overlay, use this method.
- Insert a drawing file into your current drawing using Content Explorer.

To attach an xref using the External Reference dialog box

Draw Toolbar	Menu	Command
 Insert Flyout	Insert ➤ External Reference	XATTACH

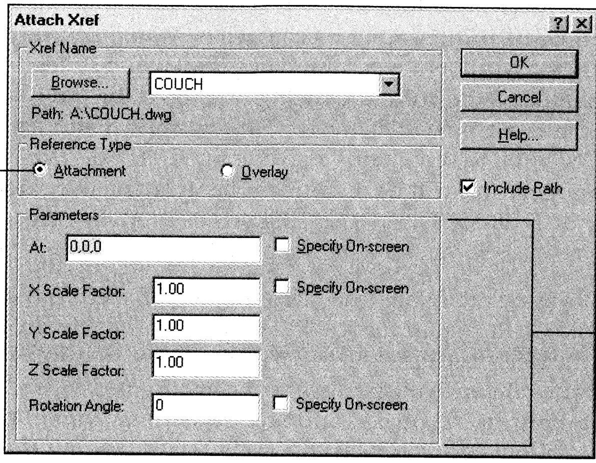
1 From the Insert menu, choose External Reference.

2 In the External Reference dialog box, choose Attach.

The first time you attach an xref, the Select File dialog box is displayed. After you select a file to attach, the Attach Xref dialog box is displayed.

3 Click Browse and select a file.

Select Attachment to attach an xref.

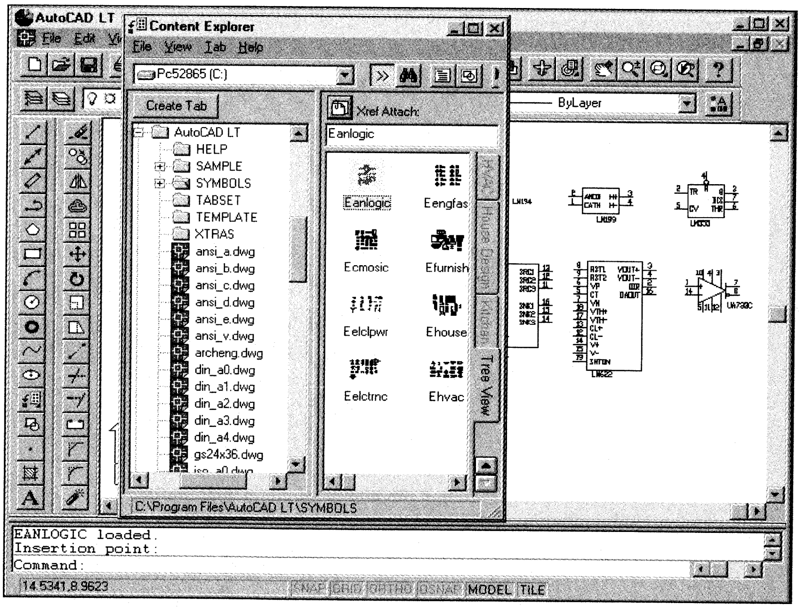


Specify the xref parameters, or check the Specify On-screen options to use your pointing device.

To attach an xref using Content Explorer

Draw Toolbar	Menu	Command
	Insert ➤ Content Explorer	CONTENT

Use Content Explorer to attach xrefs to your current drawing. Make sure the button at the top of Content View is toggled to Attach Xref.



Attaching an xref by dragging from Content Explorer

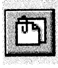
Attach an xref to the current drawing in Content Explorer by using one of the following methods:

- *Drag and drop* Select a drawing from the list displayed on the current tab, and then hold down the pick button on your pointing device and drag the selected drawing where you want it to be attached to your drawing. The insertion point of the block reference is specified by the cursor position when you release the pick button on your pointing device. In addition, you can control the insertion point using any snap settings that are active when you drag the xref into the drawing. The xref is inserted with the default scale (1), and rotation (0) settings. Use this method if you don't want to specify the insertion point using coordinates, change the scale, or rotate the xref.
- *Double-click* When you double-click a drawing listed in Content Explorer, AutoCAD LT displays prompts on the command line that specify the xref's insertion point, scale, and rotation settings. You can also specify the insertion point of the xref by using your pointing device.

NOTE Multiple drawings can be selected but cannot be attached as xrefs to the current drawing. Attach xrefs one at a time.

Managing Xrefs

With the External Reference dialog box, you can manage all of your externally referenced drawings. The External Reference dialog box displays the status of each xref and the relationship of xrefs to one another.

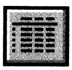
Draw Toolbar	Menu	Command
	Insert ➤ External Reference	XREF

In the dialog box, you can

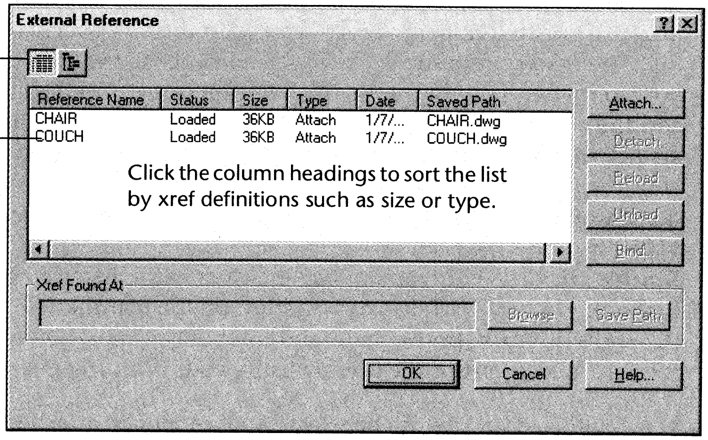
- Attach a new xref
- Detach an existing xref
- Reload or unload an existing xref
- Change an attachment to an overlay or vice versa
- Bind the entire xref definition to the current drawing
- Change the xref path

You can view the xrefs from either a list view (flat listing) or a hierarchical tree view. To choose a view, click either the Tree View or the List View icon in the upper-left corner of the dialog box. The list view is displayed by default.

List View



When the External Reference dialog box opens, the list view displays an alphabetical list of the xref definitions in the current drawing.



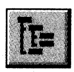
List view of the External Reference dialog box

The Status field indicates the state of the xref definitions in the drawing or the action to be taken when the dialog box is closed:

- ***Loaded*** Xref was found when the drawing was opened or reloaded.
- ***Unloaded*** Xref was unloaded by user.
- ***Unreferenced*** Parent xref is no longer present in the drawing. When an xref with nested xrefs is unloaded, not found, or unresolved, its nested xrefs become unreferenced. If a parent xref is unloaded, a message is displayed stating that its nested xrefs are "orphaned."
- ***Not found*** Xref was not found when the drawing was opened or reloaded.
- ***Unresolved*** Xref file was found, but could not be read by AutoCAD LT.

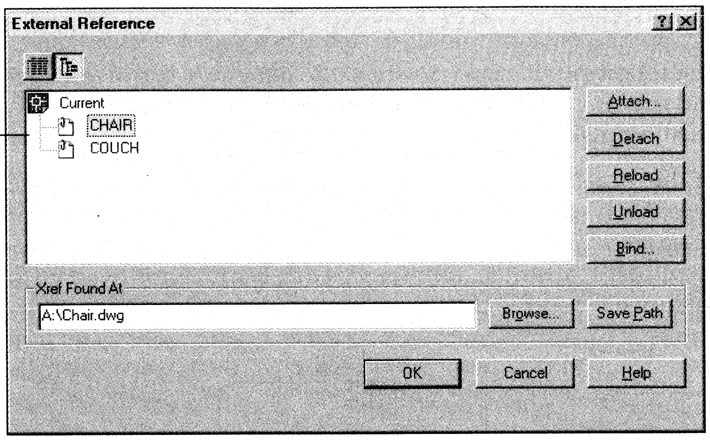
The Type column indicates whether the xref is an attachment or an overlay. By double-clicking the Type column, you can toggle an xref between an attachment and an overlay.

Displaying Tree View



When you click the Tree View icon, a hierarchical representation of the xrefs is displayed. The top level of the tree view is listed in alphabetical order.

Tree View displays the hierarchy of xrefs or nesting.



Overlays and repeat attachments are not displayed in the Tree View.

Tree view of the External Reference dialog box

Overlaying Xrefs

Overlaying also attaches a drawing as an external reference, but unlike blocks and attached xrefs, overlaid xrefs cannot be nested. If you reference a drawing that itself contains an overlaid xref, the overlaid xref does not appear in the current drawing. If another person is currently editing the xref, AutoCAD LT overlays the most recently saved version.

It is recommended that you use overlaying when you are referencing geometry that is not useful for other users to see when they reference your drawing. For example, you may have created a wiring plan for a house and need to reference the floor plan of the house. If you have chosen to overlay (rather than attach) the floor plan, then another user, who doesn't need to see the floor plan, could xref your wiring plan without the floor plan attached.

Overlaid xrefs are designed for data sharing. By overlaying an xref, you can see how your drawing relates to other drawings. Also, overlaying an xref reduces the possibility that you might create self-referencing drawings (circular xrefs).

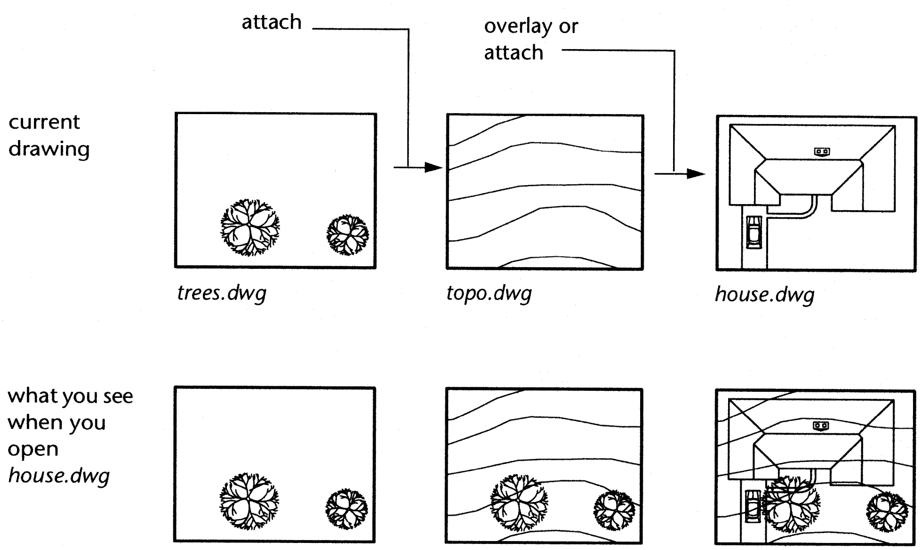
To overlay an xref

1 From the Insert menu, choose External Reference. In the External Reference dialog box, choose Attach.
2 The first time you attach an xref, the Select File dialog box is displayed. After you select a file to attach, the Attach Xref dialog box is displayed.
3 Click Browse and select a file. Make sure you select Overlay under Reference Type in the Attach Xref dialog box.

Deciding to Attach or Overlay an Xref

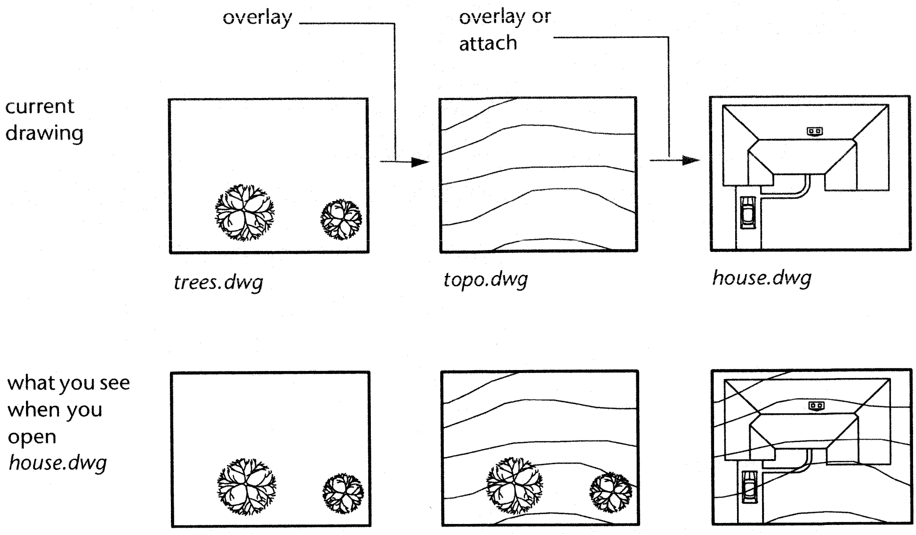
The following illustrations show the difference between attached and overlaid drawings. The upper illustrations show the type of xref used. The lower illustrations show what you see on your screen when you open each file.

In the "Attached Xrefs" illustration, *trees.dwg* is attached to *topo.dwg*. *Topo.dwg* is either attached to or overlaid on *house.dwg*. Notice that when you open *house.dwg*, you also see both *trees.dwg* and *topo.dwg*. You see *trees.dwg* because it is attached. You see *topo.dwg* because it is overlaid and is no more than one level of xref from *house.dwg*.



Attached xrefs

In the "Overlaid Xrefs" illustration, *trees.dwg* is overlaid on *topo.dwg*. *Topo.dwg* is either attached to or overlaid on *house.dwg*. Notice that when you open *house.dwg*, you see only *topo.dwg* and *house.dwg*. You do not see *trees.dwg* because it is overlaid and is more than one level from *house.dwg*.



Overlaid xrefs

Detaching Xrefs

You can detach xrefs to completely remove them from your drawing. You can also erase the individual xref instances by selecting and deleting them. Using the Detach option removes xrefs and all dependent symbols associated with that xref. If all the instances of an xref are erased from the drawing, AutoCAD LT removes the xref definition the next time the drawing is opened.

When an xref is detached, it is removed from the list view and tree view, along with any of its nested xrefs, unless the xref exists in another branch of the tree. You cannot detach a nested xref.

To detach an xref

1 From the Insert menu, choose External Reference.

2 In the External Reference dialog box, select an xref and then choose Detach.

Reloading Xrefs

When you reload an xref, the most recently saved version of the referenced file is read into the drawing, and the xref definition is updated to ensure that the current version of the reference is loaded.

If demand loading is turned on when the xref is loaded, the referenced drawing remains open during the current editing session, or a copy is created and remains open during the current editing session. When a demand loaded xref is reloaded, if the XLOADCTL system variable is set to 1, no one else can access the referenced drawing. If XLOADCTL is set to 2, AutoCAD LT loads a temporary copy of the most recently saved version of the referenced file. For more information about demand loading, see "Demand Loading and Maximizing Xref Performance" on page 307.

To reload an xref

1 From the Insert menu, choose External Reference.

2 In the External Reference dialog box, select an xref and then choose Reload.

Unloading Xrefs

When you unload a referenced file that is not being used in the current drawing, AutoCAD LT's performance is enhanced by not having to read and display unnecessary drawing geometry or symbol table information. The xref geometry and that of any nested xrefs is not displayed in the current drawing until the xref is reloaded.

Unloading is recommended when a reference file is not needed in the current drawing session but may be used eventually for plotting. For each drawing, you can maintain a working list of unloaded xrefs that can be loaded as needed.

To unload an xref

1 From the Insert menu, choose External Reference.

2 In the External Reference dialog box, select an xref and then choose Unload.

Binding Xrefs to Drawings

Binding xrefs to drawings is useful if you're archiving drawings and want to ensure that the xrefs do not change. It's also an easy way to send drawings to reviewers. Rather than send a master drawing plus each of the drawings it references, you can use the Bind option to convert those xrefs to blocks in the current drawing. Binding an xref to a drawing makes the xref a permanent part of the drawing and no longer an externally referenced file. The externally referenced information becomes a block. When the externally referenced drawing is updated, the bound xref is *not* updated.

Only loaded xrefs can be bound. If you select an unloaded xref, the Bind option is unavailable. You must reload the xref to bind it.

You can choose to either bind an xref or insert it. When you bind an xref to a drawing, the symbol table definitions associated with the externally referenced drawing are permanently added to the host drawing. The names in the symbol table are prefixed with a *blockname$x$* string, where *blockname* is the name of the bound xref. When you insert an xref, the symbol table definitions are added to the host drawing without the prefix.

For example, suppose a drawing has a layer named BLUE and an externally referenced drawing called *xref.dwg* also contains a layer named BLUE. If you bind *xref.dwg*, a new layer named XREF0BLUE is added to the host drawing. If you insert *xref.dwg*, no new layer is added to the host drawing because the layer named BLUE already exists.

You can choose to bind an xref to a drawing using either the Bind option or the Insert option. The option you choose determines how the file names are defined in the symbol table when the definitions are merged from the reference file into the host drawing.

To bind an xref to a drawing

Menu	Command
Insert ➤ External Reference	XBIND

Changing Xref Paths

If you open a drawing that contains an xref that has been moved to a different directory than the one that was saved with the xref, you can edit the file name and path.

Menu	Command
Insert ➤ External Reference	XREF

Changing Nested Xref Paths

If the path of a nested xref is changed in the current drawing, the change is saved only if the VISRETAIN system variable is set to 1. If AutoCAD LT cannot save a changed nested xref path with the current drawing, the following message is displayed.

WARNING VISRETAIN must be On to save nested Xref path.

Handling Xref Errors

You may encounter two types of error messages when you use the external reference feature: messages indicating missing reference files or circular references.

Missing Reference Files

AutoCAD LT stores the file name of the drawing used to create the external reference. Each time you open or plot the drawing or use the XREF Reload option to update the external reference, AutoCAD LT checks the file name to determine the name and location of the associated drawing file. If the name of the drawing associated with the xref has changed, AutoCAD LT cannot read the external reference file. If the drawing has been moved and doesn't exist in the AutoCAD LT support path or the current drawing directory, AutoCAD LT cannot resolve the xref.

If AutoCAD LT cannot load an external reference when it is opening your drawing, it displays an error message. In the following example of an error message, AutoCAD LT cannot find the xref *house*.

```
Resolve XREF House: \acadlt\dwg\house.dwg
Can't find \acadlt\dwg\house.dwg
```

For each insertion of the unresolved external reference, AutoCAD LT displays a single piece of text (at the insertion point, scale, and rotation angle of the original reference) that contains the path of the missing xref, and the External Reference dialog box displays the xref as "Not found." You can use the XREF Path option to update or correct the path.

Circular References

A reference file that contains a sequence of nested references that refers back to itself is considered a circular reference. For example, if drawing A attaches drawing B, attaches drawing C which attaches drawing A, the reference sequence A>B>C>A is a circular reference.

If AutoCAD LT detects a circular reference while attaching an xref, a warning is displayed asking you if you want to continue. If you respond with Yes, AutoCAD LT reads in the xref and any nested xrefs to the point where it detects the circularity. If you respond with No, AutoCAD LT halts the process and the xref doesn't get attached.

If AutoCAD LT encounters a circular reference while loading a drawing, it displays an error message and breaks the circular reference for the current session. For example, if you have the circular reference is A>B>C>A, and you open *a.dwg*, AutoCAD LT detects and breaks the circularity between *c.dwg* and *a.dwg*. The error message reads: "Breaking Circular Reference from C to current drawing."

Using the Xref Log File

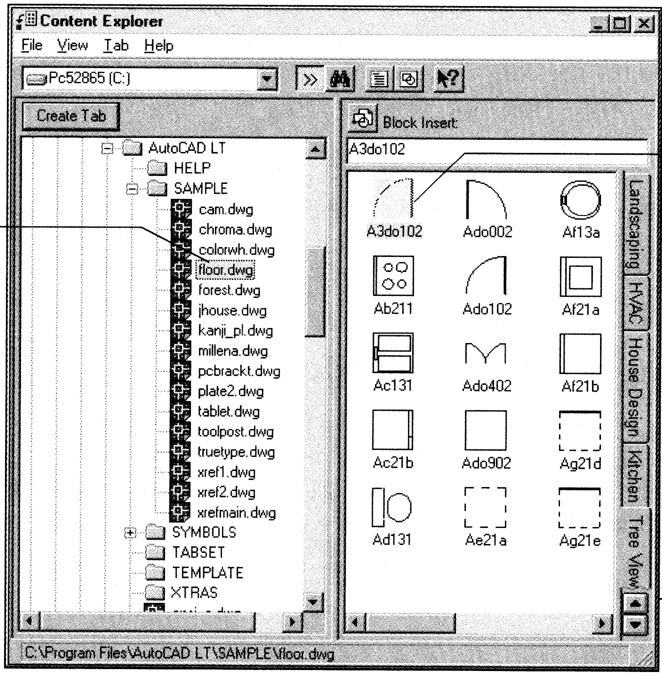
AutoCAD LT can maintain a log of its actions when attaching, detaching, and reloading xrefs and when loading a drawing containing xrefs. This log is maintained only if the XREFCTL system variable is set to 1. The system default setting is 0.

ONLINE HELP See "Resolving Xref Errors."

Managing Blocks and Xrefs with Content Explorer

You can now manage blocks and drawings on your system by using Content Explorer on the Insert menu. The options in the Content Explorer dialog box make it easier to navigate, locate blocks and drawings, drag and drop content into your current drawing, and organize data for your projects.

For example, you might navigate to a folder on your company's network and drag and drop a drawing to attach it as an external reference to your current drawing.



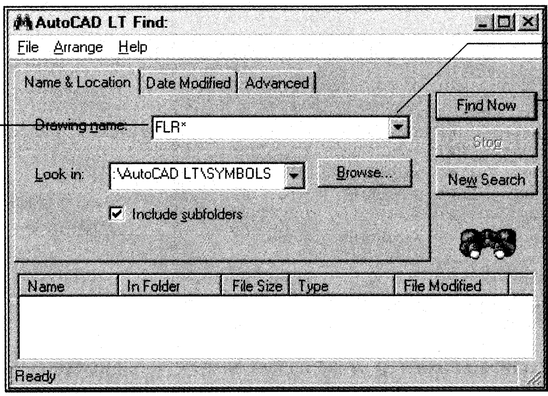
Content Explorer provides *tabs*, or shortcuts to file folders or drawings. By creating tabs, you can easily retrieve blocks and drawings. For example, you might create and save a set of tabs that contain all the block and external reference drawings for a specific project; when you work on that project, you restore the tab set for your current drawing session.

Finding Drawings and Blocks

In addition to using Tree View to search for files, you can locate blocks and drawings easily by using Find in Content Explorer.

To find drawings and blocks

Content Explorer Toolbar	Content Explorer Menu
	View ➤ Find

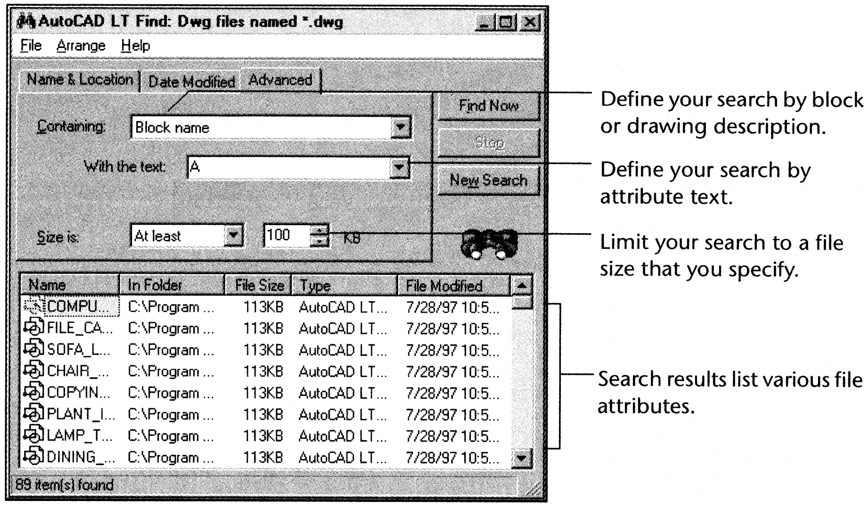


NOTE The Find dialog box searches apply only to files with a *.dwg* extension.

The Find dialog box provides three different search criteria that you can use to define your search.

- ***Name and Location*** Specifies the name and location of the drawing file. In the Drawing Name box, you can enter all or part of the drawing file name. Specify the location to start the search by clicking Browse. By default, the search path displays the current path shown in the Tree View window.
- ***Date Modified*** Specifies the date on which the files were created or last modified, or a date range or a period of days or months prior to the current date. By default, AutoCAD LT leaves the date unspecified.
- ***Advanced*** Specifies additional search parameters. Enter text to search for files containing the specified text, either in the drawing file as internal block names or attributes, or as part of block or drawing descriptions. You can also search for files that are at least as big as, or equal to, or smaller than the KB size you specify.

The search results include both drawing files and blocks that satisfy the search criteria. The search results can be sorted by any of the columns displayed. For example, to sort the list of found files by date, click Modified. Click Modified again to reverse the sort order.



Using a Block or Drawing from the Search Results

When you have located the file or block you searched for, you can quickly return to Content Explorer by double-clicking the file name or block name in search list. Tree View updates to reflect the current file and path, and the selected block or drawing file is displayed in Content View, available for insertion in the current drawing. To insert a block, see "Inserting Blocks" on page 295. To attach an xref, see "Attaching Xrefs" on page 308.

Saving Search Results

If you want to save the search results, choose Save Results from the File menu of the Find dialog box. The listed files are saved in a comma-delimited text (TXT) file, which you can then import into any spreadsheet application.

ONLINE HELP To find content using Content Explorer, see "Finding Content."

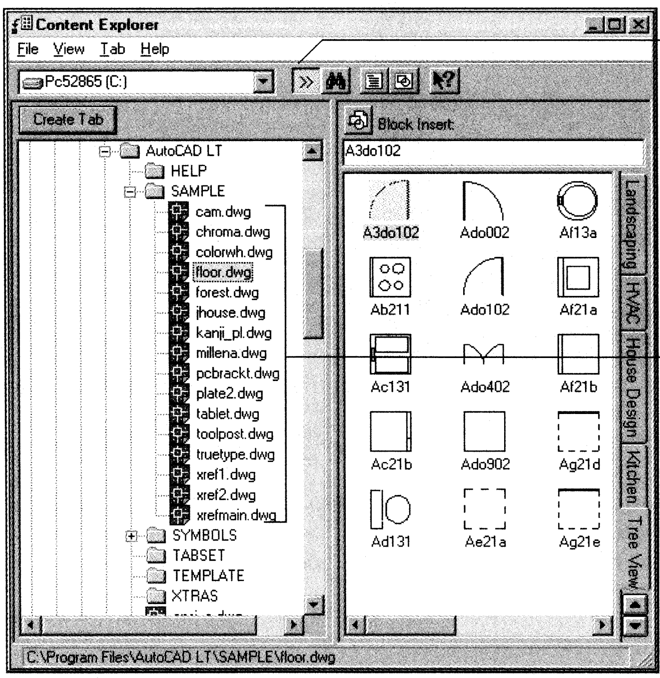
Using the Tree View

With the Tree View, you can display the contents of drawings, folders, and drives as you browse for drawings and blocks. When the Tree View is opened by clicking the Tree View button in the Content Explorer dialog box, a *tree view,* or hierarchical view of your system is displayed, including your current drawing.

To display the Tree View of Content Explorer

Content Explorer Toolbar	Content Explorer Menu
	View ➤ Tree View

With the Tree View, you can locate blocks that have been inserted into other drawings, making it easy to reuse block information. By clicking the file name of a drawing in the Tree View, you can quickly list the blocks within that drawing. The block information that Content Explorer provides can include a preview thumbnail and a description of the block, as well as the block name.



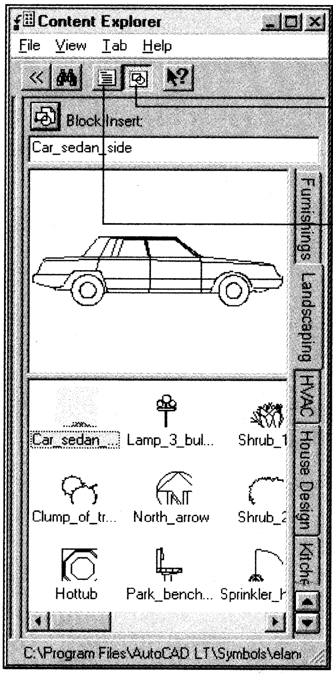
Toggle Tree View open or closed by clicking the Tree View icon.

Use Tree View to browse drawing files saved on your system, including the current drawing.

Viewing Content Information

The Content View in Content Explorer lists the drawings within a folder or the blocks within a drawing that you have selected in Tree View. You can also set up *tabs*, or shortcuts to folders and drawings that contain drawings and blocks you need to access. See the following section, "Organizing Drawing Content," for more information.

Content View provides viewing options to help you distinguish drawings and blocks more easily.



Click Preview icon to see a visual representation of the currently selected drawing or block.

Click the Description icon to see the associated description of the currently selected drawing or block.

Content View also provides different views for the listed drawings or blocks:

- ***Icons and Names*** Displays icons and names of the blocks or drawings in Content View.
- ***Names Only*** Displays the names of the blocks or drawings only.

To change the display of listed blocks and drawings

Content Explorer Shortcut Menu	Content Explorer Menu
Right-click ➤ Icons and Names	View ➤ Icons and Names
Right-click ➤ Names Only	View ➤ Names Only

Use the scroll bars in Content View to browse the listed blocks and drawings.

A block or drawing file created using BMAKE can include a preview image and a text description in its definition. For easier identification and classification of drawing data, Content Explorer can display either the preview image or the block description associated with any blocks or drawings listed in Content View.

To preview an image or description of a selected block or drawing

Content Explorer Toolbar	Content Explorer Menu
Preview	View ➤ Preview
Description	View ➤ Description

Related Use BMAKE to define internal blocks or drawing (DWG) files and associate a description with them. See "Defining Blocks" on page 290.

ONLINE HELP See "Finding Content."

Organizing Drawing Content

Once you start to use blocks and xrefs in your drawings, you usually need to organize the data in some way. Content Explorer provides different ways to accomplish this. Tabs are a convenient way to access file folders and drawings for insertion into your project. By default, AutoCAD LT provides two tabs automatically:

- ***Current DWG*** Lists the inserted blocks and attached xrefs for the current drawing.
- ***Tree View*** Lists the drawing files for the folder or the blocks for the drawing selected in the Tree View.

You can create additional tabs as shortcuts to file folders.

Creating Tabs

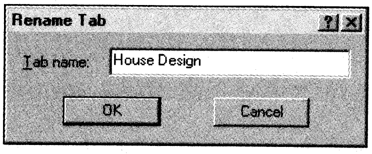
By creating additional tabs to the drawing files or folders that contain content you need for your projects, you can access data directly from within AutoCAD LT; when you find what you are looking for, you can drag and drop it directly into your current drawing.

To create a tab

1 Navigate to the folder or drawing for which you want to create a tab.
2 Click Create Tab. In the Create Tab dialog box, enter a tab name.

The newly created tab is displayed.

You can rename a tab by double-clicking it and then entering a new tab name in the Rename Tab dialog box.



The order in which tabs appear in Content View can be changed simply by dragging a tab to the position you want. The Current DWG tab is always the first tab, and the Tree View tab is always the last. Additional tabs you create can be reordered between these default tabs. If you create too many tabs to be viewed simultaneously, use the scroll arrows at the lower-right corner of the window to change the tabs in view.

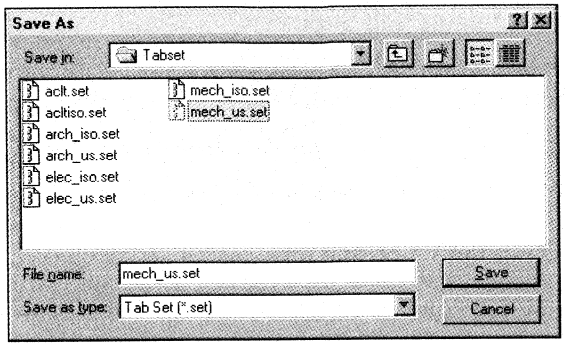
When you no longer need a tab, delete it by right-clicking it and then choosing Delete from the shortcut menu, or by choosing Delete from the Content Explorer Tab menu. You can also delete all the tabs (except for the Current DWG and Tree View tabs) by choosing Delete All from the Content Explorer Tab menu.

Creating Tab Sets

If you create several tabs, you might want to save them as a *tab set*, a reusable group of tabs that you can restore. For example, you might create tab sets that include different symbol libraries for different aspects of your project, from HVAC to lighting. When you switch to a particular drawing in your project, you can restore the tab set that contains all the relevant folders, providing direct access to the drawing and block data you need.

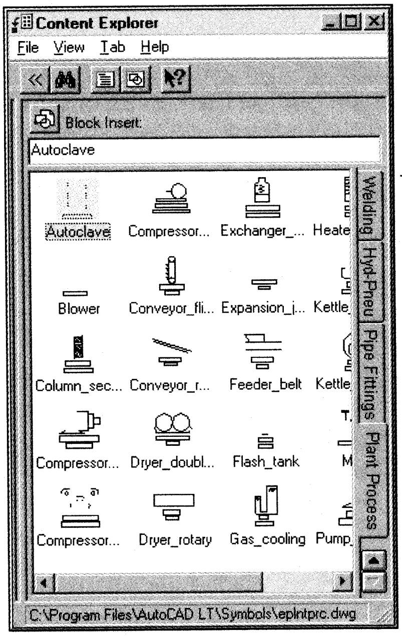
To create a tab set

1 Create the tabs that you want to be saved in your tab set.
2 Choose Save As Tab Set from the Content Explorer Tab menu. Enter a name for the tab set in the Save As Tab Set dialog box.



NOTE The Current DWG and Tree View tabs are not saved in a tab set, nor are they replaced when you restore a tab set.

When you need to restore the tab set, choose Open Tab Set from the Content Explorer Tab menu. Except for the Current DWG and Tree View tabs, all existing tabs in the Content View are replaced.



A new tab set is displayed when you pick a tab set option from the Tab ➤ Open Tab Set menu item.

If you need to use different tab sets simultaneously, you can *merge* them; merging appends the tabs in the second set to the end of the existing tab set. Choose Merge Tab Set from the Content Explorer Tab menu to combine two tab sets.

Working with Default Tab Sets

During installation, AutoCAD LT prompts you to specify the set of symbols used as the default tab set for Content Explorer. Tab sets are provided for architectural, mechanical, and electronic drawing projects, and are available in both English and metric measurements. You can turn the default tab set on and off, and you can specify a different default tab set.

To turn the default tab set on and off

- From the Content Explorer Tab menu, choose Load Default Tabs.

To specify a different default tab set

- From the Content Explorer Tab menu, choose Open Tab Set. The Open dialog box defaults to the Tabset folder. Choose the tab set to specify the default.

Moving and Copying Drawings

Tabs are also convenient for moving drawing files between folders. You can drag and drop any drawing file from any tab that points to a folder to another tab that points to a folder, or you can use the Move Content and Copy Content commands.

To move or copy content from one tab to another

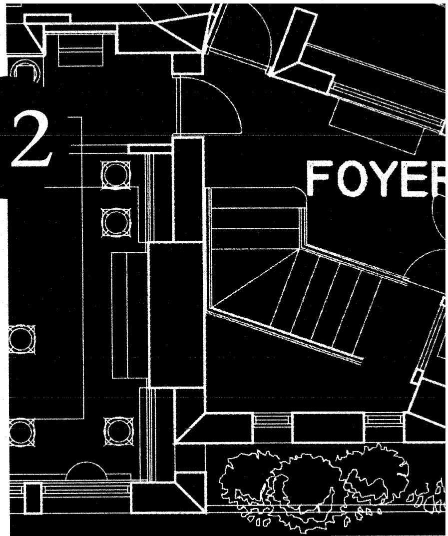
1 Navigate to the tab from which you want to move or copy content.

2 In the Content View, select the drawings, and then use *one* of the following methods:

- Drag the selected content to *move* it to the tab of the folder where you want it to be stored. Hold down CTRL and drag the selected content to *copy* it to the tab of the folder where you want it to be stored.
- Right-click in the Content View, and then choose Move Content or Copy Content from the shortcut menu.
- Choose Copy Content or Move Content from the Content Explorer File menu. In the dialog box, select the tab name to which the content is to be moved or copied.

NOTE Content can be moved or copied between folders *only*. Content cannot be copied from the Current DWG tab or any tab that points to a drawing rather than a folder.

ONLINE HELP See "Working with Tabs and Tab Sets."



12

Creating a Layout

After you've created your drawing with AutoCAD LT, you usually plot it on paper. A plotted drawing can contain a single view of your drawing or a more complex arrangement of views. AutoCAD LT provides a special drawing environment, known as paper space, that you can use to lay out various components of your drawing, including the annotation. In paper space, you can create windows called floating viewports, which display various views of the drawing. Depending on your needs, you can plot one or more viewports or set options that determine what is plotted and how the image fits the paper.

In this chapter

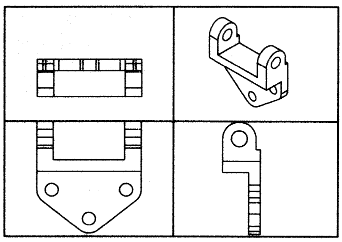
- Using paper space and model space
- Adding title blocks and borders
- Creating and using floating viewports
- Using named views in paper space

In online Help

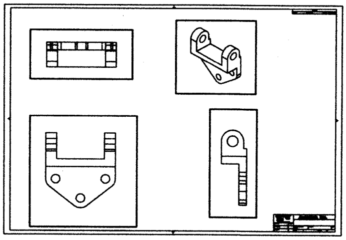
- When working in floating viewports of 3D models, you can also specify viewpoints. See "Changing 3D Viewpoints."

Using Paper Space and Model Space

As you create your drawing, you may want to display different views of it. You can split the screen to display different views called *tiled viewports*. When you work with tiled viewports, you are in *model space*, where you create the basic drawing, or model. If several tiled viewports are displayed, editing in one viewport affects all other viewports. However, you can set magnification, grid, and snap individually for each viewport.



tiled viewports



floating viewports

Same model displayed in tiled and floating viewports

When you switch to *paper space*, the graphics area displays a space that represents the "paper" on which you arrange your drawing. In this space, you create *floating viewports* to contain different views of your model. In paper space, floating viewports are treated as objects that you can move and resize in order to create a suitable layout. You are not restricted to plotting a single model space view, as you are with tiled viewports. Therefore, you can plot any arrangement of floating viewports. In paper space, you also can draw objects, such as title blocks or annotations, directly in the paper space view without affecting the model itself.

Because floating viewports are treated as objects, you *cannot* edit the model in paper space. To access the model in a floating viewport, you toggle from paper space to model space. As a result, you can work with the model while keeping the overall layout visible. In floating viewports, the editing and view-changing capabilities are almost the same as in tiled viewports. However, you have more control over the individual views. For example, you can freeze or turn off layers in some viewports without affecting others. You can turn an entire viewport display on or off.

When you work in tiled viewports, the TILEMODE system variable is on. When TILEMODE is off, you work in floating viewports, and you can switch between paper space and model space as needed. Paper space, model space, and TILEMODE settings are interrelated as shown in the table.

Using paper space and model space

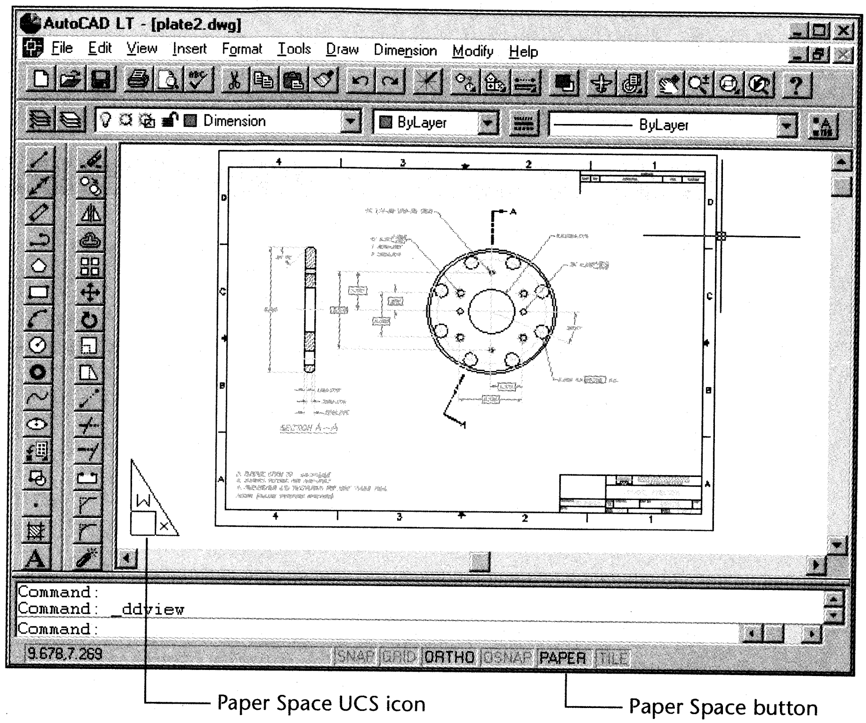
Space	Status	Purpose
Paper space	TILEMODE off	Arrange the layout by creating floating viewports and adding title block, borders, and annotation. Objects created in paper space can be edited, including viewports. Editing does not affect the model.
Model space in floating viewports	TILEMODE off	Work within floating viewports to edit the model or change views. You can turn off or freeze layers in individual viewports.
Model space in tiled viewports	TILEMODE on	Split the screen into tiled viewports to edit different views of the model. When you turn off layers, all viewports are affected.

To switch from model to paper space, use the toggles on the status bar. The combined status of the Paper/Model and Tile toggles determines the current space.

Switching to Paper Space

If you use the Advanced Setup wizard to start a new drawing, you have the option to use paper space layout capabilities. You can also switch to paper space at any time to arrange the drawing layout in preparation for plotting, including creating floating viewports and inserting a title block and border.

Use paper space also to create objects such as annotations and dimensions that you don't want to include in the model drawing. Objects created in paper space are visible only in paper space.

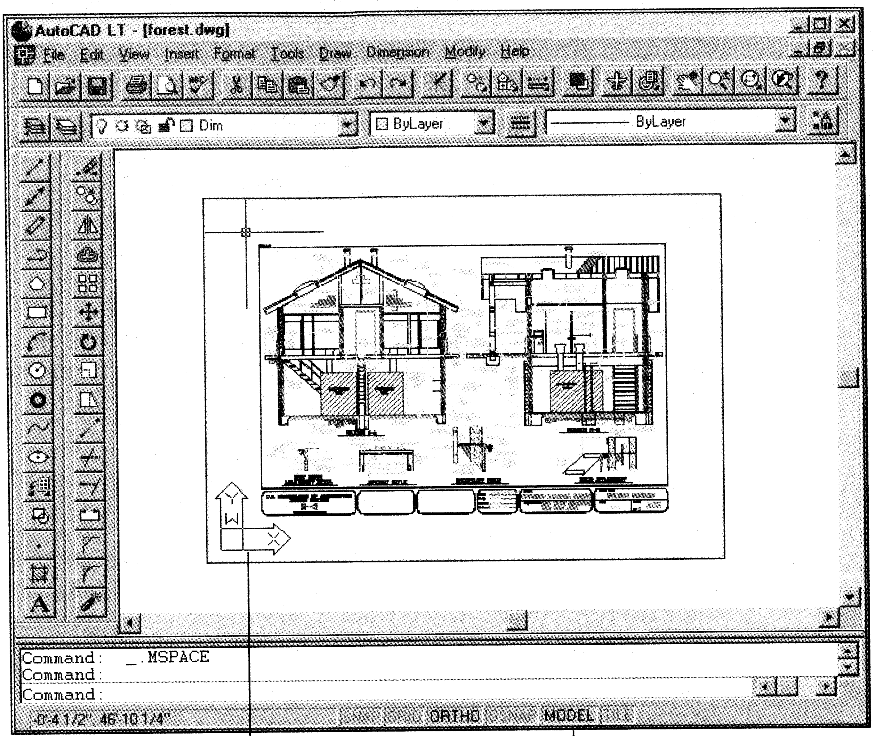


To turn on paper space

Status Bar	Menu	Command
MODEL Double-click	View ➤ Paper Space	Set TILEMODE = 0

Related If TILEMODE is off and model space is on, use the PSPACE command to turn on paper space.

Switching to Model Space



Model Space UCS icon

Model Space button

Once you have created floating viewports, you can edit the model itself by switching from paper space to floating model space. This switch allows you to edit the model in a floating viewport.

If you switch to model space before you have created floating viewports, AutoCAD LT prompts you to use MVIEW to create viewports. See "Creating Floating Viewports" on page 335.

To edit the model in floating viewports

Status Bar	Menu	Command
PAPER Double-click	View ➤ Model Space (Floating)	Set TILEMODE = 0. Then enter **mspace** to switch to model space.

ONLINE HELP See "Working with Paper Space and Model Space."

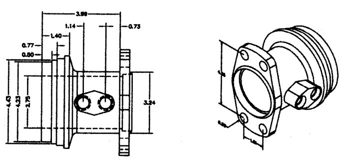
Preparing a Layout

The arrangement of a drawing on paper is an important part of the drawing process. The initial drawing may consist of a single model; however, you can display this model in many ways. You can arrange it within a border, show it from different perspectives, magnify details that you want to emphasize, or hide insignificant details.

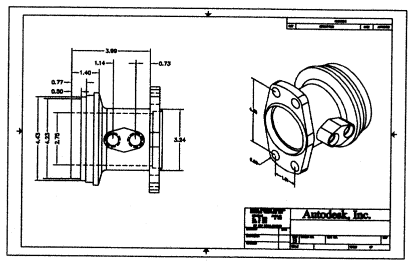
Adding a Title Block and Border

Because title blocks are part of the layout, they usually are drawn in paper space rather than model space. AutoCAD LT provides more than a dozen standard title blocks, which you can insert into your layout using the Advanced Setup wizard. For more information, see "Using the Advanced Setup Wizard" on page 65.

ONLINE HELP See "Adding a Title Block."



drawing in model space

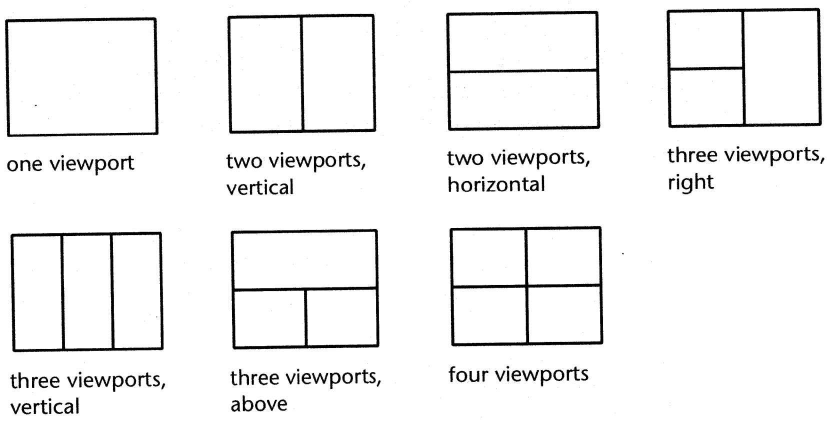


drawing with title block and border inserted in paper space

Creating Floating Viewports

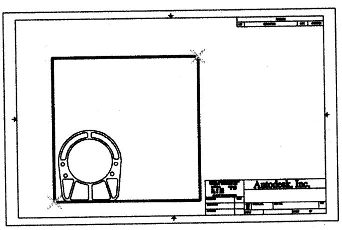
If you use the Advanced Setup wizard to open a new drawing and choose to work in paper space, you can insert a single floating viewport, depending on the title block you specify. You can also create floating viewports using MVIEW.

You can place the floating viewports you create anywhere in the drawing area. As with tiled viewports, you can choose one of several standard configurations. If you choose 1 Viewport, you can either create a viewport that fits the entire graphics area or specify a size and shape. If you choose multiple viewports, you can configure them in various arrangements. The illustrations show some of the options.

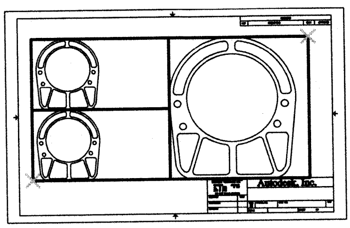


Examples of standard floating viewport configurations

The configuration you choose depends on how you plan to use the floating viewports. If you are designing a layout, you may want to use the 1 Viewport option to place each viewport individually. If you expect to use the viewports to edit the model, one of the other configurations may be more useful.



starting a layout with one viewport



using three viewports for editing

After you specify the number of viewports you want to create, you define an area to contain them. Although these floating viewports are created simultaneously, they are separate objects that can be moved, resized, or erased as needed.

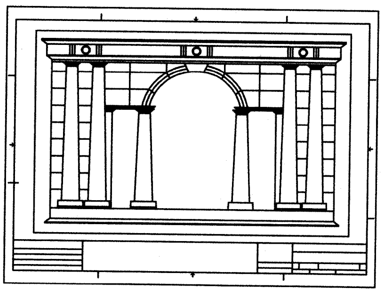
To create floating viewports

Menu	Command
View ➤ Floating Viewports (available in paper space only)	MVIEW

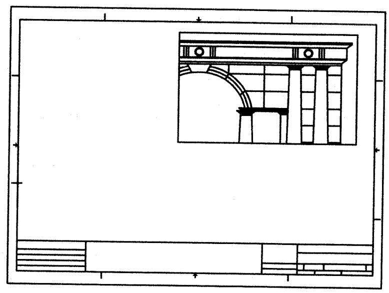
Rearranging and Removing Floating Viewports

After you insert floating viewports, you can rearrange them according to your needs. Floating viewports can be edited just as any other object in AutoCAD LT. You can snap to, copy, scale, stretch, erase, and move viewports using standard commands and most grip commands. To use these commands, you must be in paper space, and the viewport borders must be visible. For more information on editing objects such as viewports, see chapter 7, "Editing Your Drawing."

When you scale or zoom a view within a viewport, you may need to resize the viewport. Scaling or stretching the viewport border does not change the view within the viewport, as shown in the following illustration.



viewport to be moved and resized



resized viewport

By resizing the viewport borders, you can clip the view to display only a specific section of the drawing.

ONLINE HELP See "Working with Paper Space and Floating Viewports."

Controlling Visibility in Floating Viewports

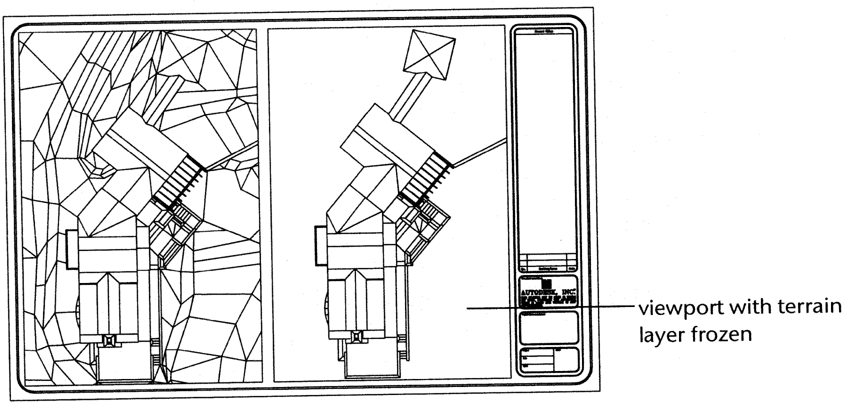
You can control the visibility of objects in a floating viewport using several methods. You can:

- Freeze or thaw layers in individual viewports
- Turn viewports on and off
- Limit the number of viewports that can be active

These methods are useful for limiting screen regeneration and for emphasizing or hiding different elements of your drawing.

Freezing or Thawing Layers in Individual Viewports

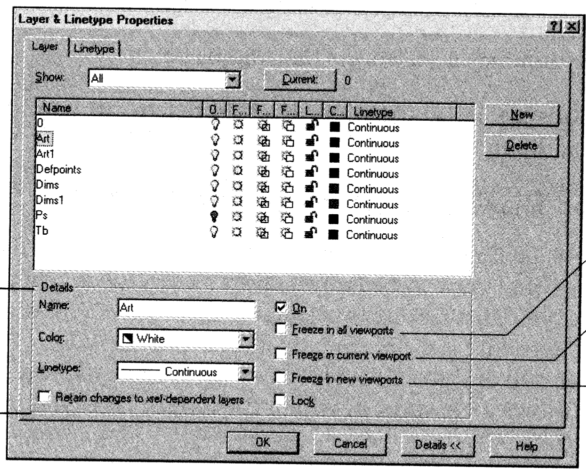
You can freeze or thaw layers in current and future floating viewports without affecting other viewports. Frozen layers are invisible. They are not regenerated or plotted. This feature is useful, for example, if you want to create an annotation layer that is visible only in a particular viewport. You can also set layers to be frozen or thawed for any new viewports you create. The following illustration shows a viewport in which the terrain layer has been frozen in one viewport.



Thawing restores the layer visibility. You can freeze or thaw layers in the current viewport with VPLAYER.

To freeze or thaw layers in the current floating viewport

Object Properties Toolbar	Menu	Command
	Format ➤ Layer	LAYER



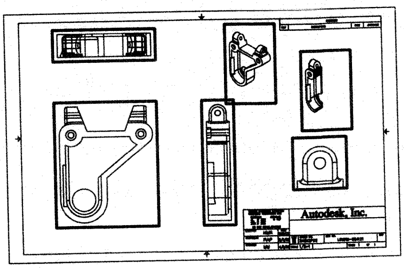
Listing Frozen Layers

When you need to know which layers are frozen in the current viewport, you can filter and list them. You can list frozen layers only for viewports whose borders are visible.

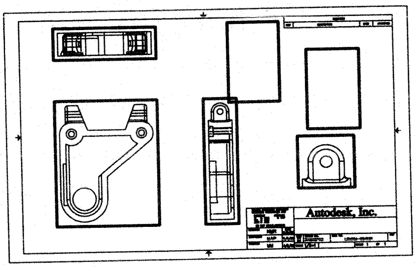
Command
VPLAYER, ?

Turning Floating Viewports On or Off

Displaying a large number of active floating viewports can affect your system's performance as the content of each viewport regenerates. You can save time by turning some viewports off or by limiting the number of active viewports. The following illustration shows the effects of turning off two viewports.



all viewports on



two viewports off

New viewports are turned on by default. If you turn off the viewports you aren't using, you can move or resize viewports without waiting for each one to regenerate. You can also turn off viewports that you don't want to plot.

To turn viewports on or off

Menu	Command
View ➤ Floating Viewports ➤ Viewports On or Viewports Off	MVIEW

Related If you want to hide the viewport borders as you work or when you plot, create a special layer that you use for inserting viewports. Then, turn off or freeze that layer.

Limiting Active Viewports

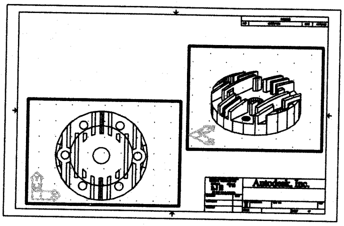
A floating viewport is considered to be active when it is on, visible in the graphics area, and not part of a block reference. (Viewports contained in blocks do not display model space views.)

Your drawing can display an unlimited number of viewports. However, your operating system and display driver determine the number of viewports that are active. Active viewports are also limited by the maximum value set by the MAXACTVP system variable. Lowering this limit can improve performance because inactive viewports are blank and their contents are not regenerated. The default limit is 48.

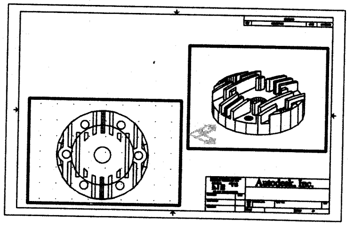
ONLINE HELP See "Managing Layer Visibility in a Layout."

Changing Viewport Views and Content

To change the view within a floating viewport, you must be in model space. Changes to the grid and snap settings or zoom values affect only the current viewport.



grid and UCS icon turned on in two viewports



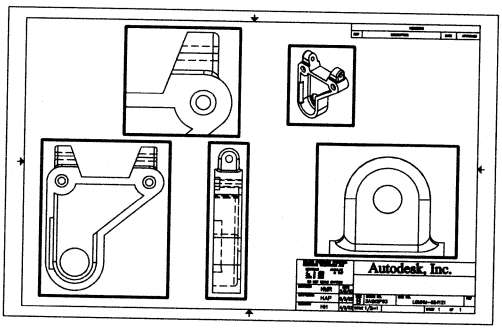
grid turned on in one viewport, UCS icon on in the other

To edit a drawing in a floating viewport, you must be in model space. Click inside the floating viewport you want to edit to make it current.

If you want to create, resize, or move a floating viewport, or edit any other object created in paper space, such as annotations, you must be in paper space.

Scaling Views Relative to Paper Space

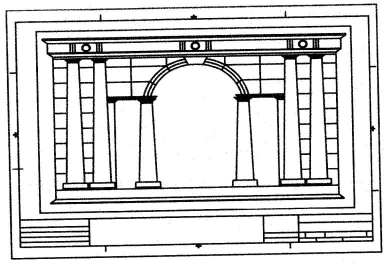
Before you plot, you can establish accurate zoom scale factors for each section of your drawing. Scaling views relative to paper space establishes a consistent scale for each displayed view. For example, the following illustration shows a paper space view with several viewports—each set to different zoom scale factors and views. To scale the plotted drawing accurately, you must scale each view relative to *paper space*, not relative to the previous view or to the full-scale model.



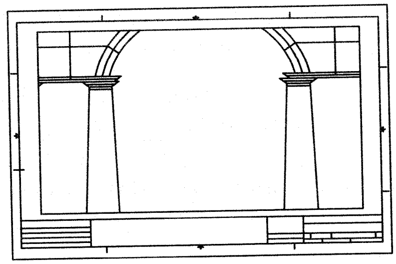
Viewports using different scales

When you work in paper space, the scale factor represents a ratio between the size of the plotted drawing and the actual size of the model displayed in the viewports. To derive this scale, divide paper space units by model space units. For a quarter-scale drawing, for example, you specify a scale factor of one paper space unit to four model space units (1:4).

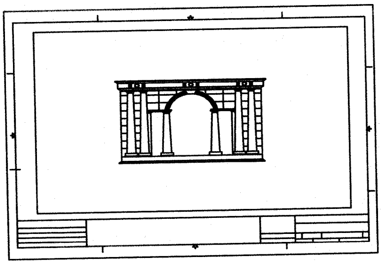
You can use ZOOM to scale viewports relative to paper space units. The scale factor you enter is relative to the current paper space scale. As shown in the illustrations, if you enter a scale of **2xp**, the scale in the viewport increases to twice the size of the paper space units. A scale of **.5xp** sets the scale to half the size of the paper space units. The model is plotted at half its actual size.



current view



zoomed to 2xp



zoomed to .5xp

Zooming to scale a view relative to paper space

For a 1:4 ratio, for example, you would enter **0.25xp** or **1/4xp**. You derive the 0.25 value by dividing 1 by 4.

To zoom to a scale factor

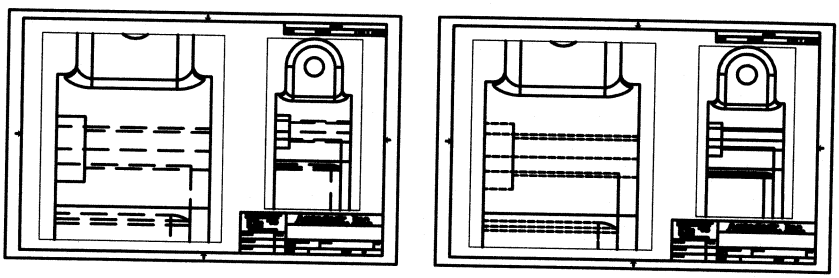
Standard Toolbar	Menu	Command
Zoom Flyout	View ➤ Zoom ➤ Scale	ZOOM Scale

Scaling Pattern Linetypes in Paper Space

When you scale views in floating viewports, you sometimes create inconsistency in the appearance of dashed or dotted lines. For example, when you decrease the scale of a drawing containing dashed lines, the spacing and dashes decrease in length accordingly. With AutoCAD LT, you can control the scaling of linetypes to correspond to the model or paper space scale or to remain the same at any zoom value.

In paper space you can scale linetypes in two ways. The scale can be based on the drawing units of the space in which the object was created (model or paper). The linetype scale also can be a uniform scale based on paper space units. To maintain the same linetype scaling for objects displayed at different zoom scales in different viewports, use the PSLTSCALE system variable.

In the following illustration, the pattern linetype of the lines in model space has been scaled uniformly in paper space using the PSLTSCALE system variable. Notice that the linetype in the two viewports has the same scale, even though the objects have different zoom scales.

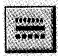


psltscale=1: dashes scaled to paper space

psltscale=0: dashes scaled to space where they were created

Scaled linetypes

To scale pattern linetypes evenly in paper space

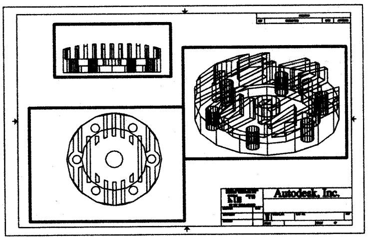
Object Properties Toolbar	Menu	Command
	Format ➤ Linetype. In the Layer & Linetype Properties dialog box under Details, specify the Global Scale Factor.	PSLTSCALE

Related LTSCALE sets the global linetype scale factor. The CELTSCALE command sets linetype scaling per object.

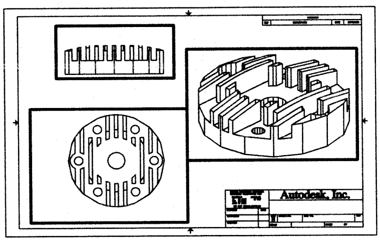
ONLINE HELP See "Scaling Views and Linetypes (PSPACE)."

Hiding Lines in Plotted Viewports

If your drawing contains 3D or thickened objects, you can direct AutoCAD LT to remove hidden lines from specific viewports when you plot the paper space view. The Hideplot option of MVIEW affects only the plotted output, not the screen display. To remove hidden lines on the screen, use HIDE.



Hideplot off



Hideplot on

You do not see the effect of this command until you plot the drawing.

To turn hidden lines on or off when the paper space view is plotted

Menu	Command
View ➤ Floating Viewports ➤ Hideplot	MVIEW Hideplot

ONLINE HELP See "Removing Hidden Lines for Printing (PSPACE)."

Using Named Views in Paper Space

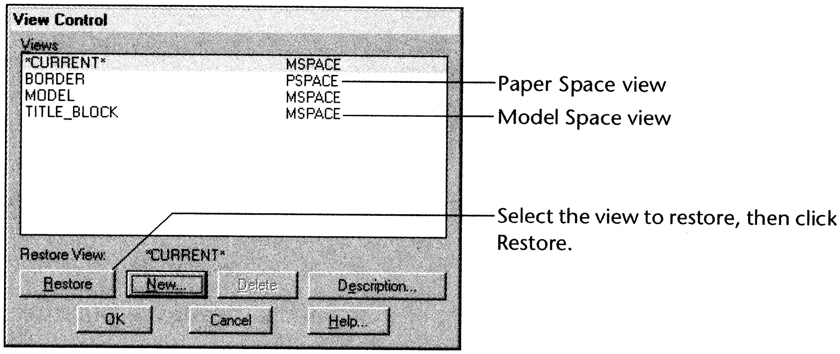
You can save views that you want to reuse or plot. Each view represents a specific pan and zoom position. For example, you can save a view of a room in a floor plan and then restore the view whenever you want to work on it.

In model space (tiled or floating), you save a view from a single viewport. When restored, the saved view replaces the view in the selected viewport. In paper space, a saved view may include one or more floating viewports. When you restore it, the zoom and pan position of the restored view replaces the current paper space display.

The procedure for saving the views in floating viewports and paper space is the same as for saving views in tiled viewports.

To restore a named paper space view

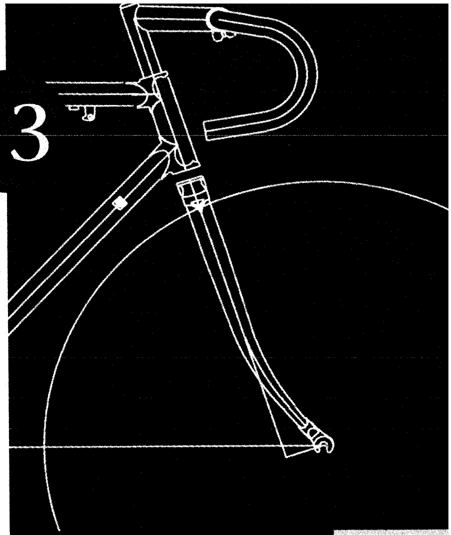
Menu	Command
View ➤ Named Views	DDVIEW



ONLINE HELP See "Using Named Views in Paper Space and Model Space."

13

Plotting Your Drawing



After you've created your drawing in AutoCAD LT, you usually plot it on paper. You can reproduce your drawing on either a plotter or a printer. If you need to use your drawing with other applications, you can also plot to a file in any of several formats.

In this chapter

- Preparing to plot
- Performing basic plotting
- Previewing the plotted drawing
- Setting up the plot configuration
- Saving and restoring plot configurations
- Plotting to a file

Getting Ready to Plot

You can produce hard-copy output on a variety of plotters and printers, including system printers and Océ, CalComp, Houston Instrument, and Hewlett-Packard printers and plotters (HP-GL and HP-GL/2).

By default, AutoCAD LT is set to plot to the Windows system printer specified in the Windows Printers folder. However, you might want to specify a different output device and plotting settings. In addition to the controls you use to load and specify different printers and plotters, AutoCAD LT provides the capability to save, reuse, and modify the settings you specify, including the device and device-independent settings such as paper size and scale factors. The following table shows the plotting settings and controls and where to find information in the AutoCAD LT documentation. You may also need to refer to the documentation provided with your output device for hardware-specific details.

Where to find more information

To do this	Use
Add or remove a plotter	File ➤ Printer Setup See "Loading a New Printer or Plotter" on page 348
Modify basic plotter configuration	File ➤ Printer Setup See "Reconfiguring a Printer or Plotter" on page 349
Change plot settings such as size, scale, plot area, and rotation	File ➤ Print See "Setting Plot Area and Output Parameters" on page 355 and "Setting the Plot Scale" on page 360
Modify device driver configuration	HPCONFIG CCONFIG OCECONFIG See "Configuring CalComp, Hewlett-Packard, and Océ Plotter Drivers" in online Help
Set up cables, parity, baud rate	HPCONFIG CCONFIG OCECONFIG See "Configuring CalComp, Hewlett-Packard, and Océ Plotter Drivers" in online Help

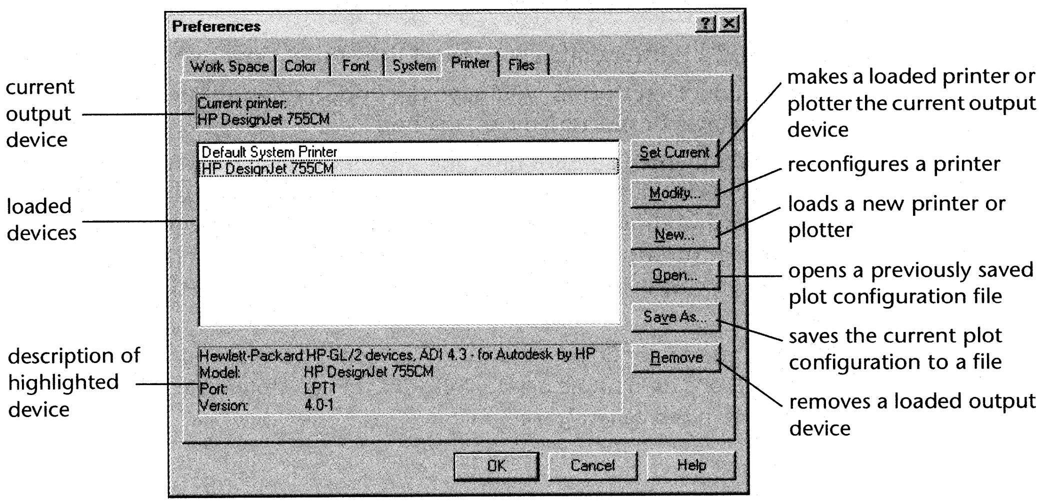
Where to find more information (*continued*)

To do this	Use
Save and reuse plot and plotter configurations	File ➤ Printer Setup File ➤ Print See "Reusing a Plot Configuration" on page 361

NOTE This guide uses the term *plot* to refer to both printing and plotting, unless they need to be distinguished.

Setting Printer and Plotter Preferences

When you install AutoCAD LT, the current system printer is automatically configured as your output device. If you intend to use the system printer for printing or plotting you usually don't need to perform any additional configuration. However, by choosing Printer Setup from the File menu to display the Printer tab in the Preferences dialog box, you can load a new print driver, and change, reconfigure, or delete a printer or plotter. AutoCAD LT also provides the option to save and reuse plotter configuration settings. See "Reusing a Plot Configuration" on page 361.



ONLINE HELP For device-specific configuration information, see "Configuring CalComp, Hewlett-Packard, and Océ Plotter Drivers."

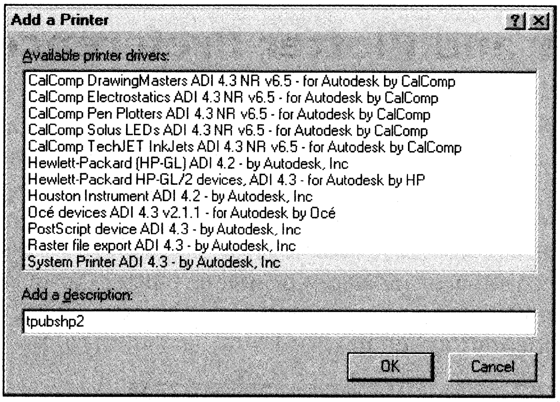
Loading a New Printer or Plotter

You can add a new printer or plotter to the list of loaded printers and plotters at any time. You load new printers or plotters by first selecting a driver and then responding to a series of prompts displayed in the AutoCAD LT text window.

To load a new printer or plotter

1 From the File menu, choose Printer Setup.

2 On the Printer tab in the Preferences dialog box, choose New.



3 In the Add a Printer dialog box, select the printer driver you want to load.

4 Under Add a Description, you have the option of entering a description for the printer or plotter you are loading.

If you do not enter a description, the default name of the printer or plotter you select is used. Because it is possible to configure the same printer or plotter several times and several different ways, the same device name may appear multiple times in the list of loaded printers and plotters. This can lead to confusion later when you try to select a loaded printer or plotter.

NOTE You can configure AutoCAD LT for many devices and store multiple configurations for a single device. AutoCAD LT can store up to 29 device configurations in the *aclt4.cfg* file.

5 Choose OK.

The AutoCAD LT text window is displayed so you can configure the new printer.

6 Respond to the prompts in the text window.

When you have responded to all the prompts, the Preferences dialog box is displayed again. The name of the printer you just configured is displayed in the list.

7 To make the new printer current, select the printer name from the list and choose Set Current.

8 Choose OK.

Setting the Current Printer or Plotter

A printer or plotter must first be loaded before it can be selected as the current output device. The Printer tab lists the loaded printers and plotters and displays the name of the current printer or plotter. At any time you can change the current printer or plotter to any of the previously loaded printers or plotters in the list.

To set the current printer or plotter

1. From the File menu, choose Printer Setup.
2. On the Printer tab in the Preferences dialog box, select a device from the list of loaded printers and plotters.
3. Choose Set Current.
4. Choose OK.

Reconfiguring a Printer or Plotter

You can reconfigure a loaded printer or plotter at any time.

To reconfigure a printer or plotter

1. From the File menu, choose Printer Setup.
2. On the Printer tab in the Preferences dialog box, select a device from the list of loaded printers and plotters.
3. Choose Modify.
4. In the Reconfigure a Printer dialog box under Description, you have the option of entering a new description.
5. Choose Reconfigure.

   The AutoCAD LT text window is displayed so that you can reconfigure the printer.

6 Respond to the prompts.

When you have responded to all the prompts, the Reconfigure a Printer dialog box is displayed again.

7 Choose OK to close each dialog box.

Removing a Printer or Plotter

You can remove a printer or plotter from the list of loaded devices.

To remove a printer

1 From the File menu, choose Printer Setup.
2 On the Printer tab in the Preferences dialog box, select a device from the list of loaded printers and plotters.
3 Choose Remove.
4 Choose Yes.
5 Choose OK.

You can reload a printer or plotter at any time.

Plotting Your Drawing

You can plot your drawing on any Windows-supported output device, and on any plotter supported by ADI drivers supplied with AutoCAD LT. To specify, or modify device settings or load new settings, use the Printer tab of the Preferences dialog.

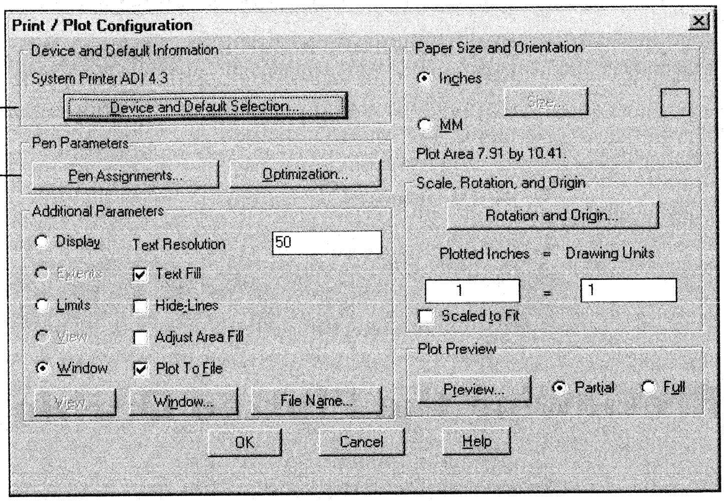
You can also plot your drawing to a specified file format if you need to use your drawing with other applications. For more information, see "Plotting to a File" on page 364.

NOTE AutoCAD LT does not store plot configuration information in the drawing files. For each drawing, you can define the settings for the desired output by selecting an existing plot configuration as a starting point, modifying plotter settings in the Print/Plot Configuration dialog box. Changes made in the Print/Plot Configuration dialog box are saved as new default plotter settings only if a successful plot is completed using the new settings.

The section describes how to plot without changing a plot configuration. For information about changing the plot configuration, see "Setting Up a Plot Configuration" on page 354.

changes plotters, plot configuration, and setup

assigns widths and colors to pens



Notice that the current plotter driver is displayed under Device and Default information.

In the Print/Plot Configuration dialog box, change any or all of the following settings:

- ***Device and Default Selection*** Displays information about the current plotter. In the Device and Default Selection dialog box, specify the device configuration or plot configuration. Modify the settings for your particular output device by choosing Change under Device Specific Configuration.
- ***Pen Parameters*** Assigns pen settings including pen widths, pen speeds, and colors. See "Setting Pen Assignments and Colors" on page 354.
- ***Paper Size and Orientation*** Specifies inches or millimeters as the units for plotting, and a paper size for your current output device. See "Setting Paper Size" on page 358.
- ***Additional Parameters*** Specifies the settings for the plot area and plotted output. See "Setting Plot Area and Output Parameters" on page 355.
- ***Scale, Rotation, and Origin*** Specifies angular and displacement settings for the output. The current scale factor for plotting is indicated in the boxes. Edit the values to change the scale factor, or select Scaled to Fit to plot the area at maximum size for the specified paper size. See "Positioning the View on the Paper" on page 358.

- *Plot Preview* Shows final output before it is printed. Select Partial to display a schematic image of the plot area and the specified paper size. Use this option to check the position of the plot on the paper. Select Full to display the actual plot area of the drawing. See "Previewing the Plotted Drawing," next.

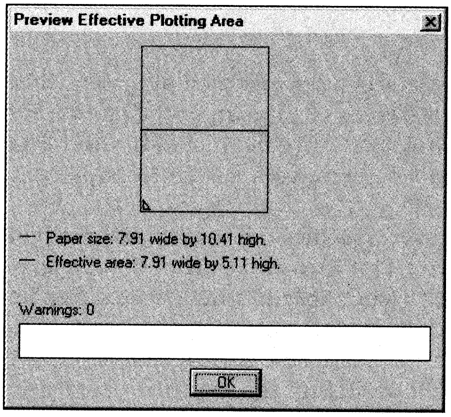
To plot a drawing

Standard Toolbar	Menu	Command
	File ➤ Print	PRINT or PLOT

Previewing the Plotted Drawing

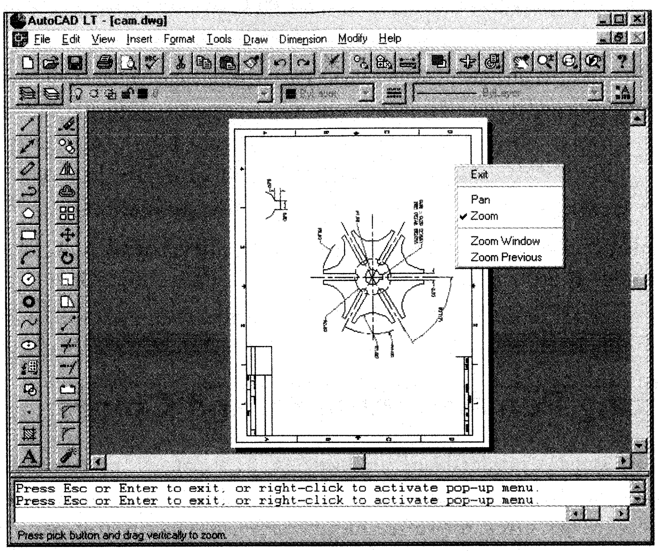
You can preview the final output of your drawing by using the Print Preview option in the Print/Plot Configuration dialog box.

Partial Preview shows a schematic image of the paper area and the position of the plot. This option does not display the actual drawing, but you can check the position of the plot image on the paper. The Warning list in the Preview Effective Plotting Area dialog box indicates if any of the plotted image is outside the effective plotting area, for example, because the paper size is too small or print margin settings are incorrect.



Example of a partial preview of the effective plotting area

Use Full Preview to view the drawing in full detail as it would be plotted using the current plot configuration and paper settings. Select Full, and then click Preview to open the Full Preview view. A progress bar indicates that the preview is being created using the current settings.



Example of full preview of effective plotting area

A representation of the paper area and the plot image is displayed. Use the shortcut menu to zoom and pan the previewed plot image. Choose Exit from the shortcut menu to exit the Full Preview view and return to the Print/Plot Configuration dialog box.

For more information on zooming and panning, see "Using Zoom and Pan" on page 196.

During Plot Preview, zoom-in is limited by the resolution of your plotter. Plot Preview stops zoom-in at the point where one pixel on your display is equal to one pixel (or plotter step) on your plotter. You can only zoom in to the level of detail that your plotter or printer is capable of plotting.

Setting Up a Plot Configuration

You can choose which plot configuration to make current by using the Device and Default Selection option in the Print/Plot Configuration dialog box.

Use the options in the Device and Default Selection dialog box to specify the output device driver, display or change the settings for that device driver, and load or create plot configuration files that contain both device-independent and device-specific plot information. For information on saving and reusing plot configuration files, see "Reusing a Plot Configuration" on page 361.

The plot configuration settings control the final output. These settings affect pen assignments, plot area, scale, paper size, and rotation. Understanding how to use these settings helps ensure that your drawing is plotted as expected.

Setting Pen Assignments and Colors

Each object you create has an AutoCAD LT color that is assigned either by layer or specifically to the object. Although these colors can correspond to plotted colors, they more frequently are used to distinguish different layers on the screen. Colors also are used to assign line widths or weights. How you assign pens depends on whether you use a pen plotter or a raster printer.

Pen Width

For most pen plotters, the physical width of the pen determines the plotted line weight. However, some plotters can simulate a specified pen width by using multiple pen strokes. If you plot filled solids, wide polylines, or trace fills, be sure to configure the pen width to match the actual width of the pen.

Many plotter pens are marked with pen width in millimeters. Be sure that the pen width is accurately entered in the current units (inches or millimeters). If you are working in inches but your pen tip measurements are in millimeters, you can select Millimeters in the Print/Plot Configuration dialog box, make your pen assignments, and then switch back to inches.

NOTE You can specify pen widths up to a value of 25% of the smallest dimension measurement for the current paper size setting.

If you use a single-pen plotter, you can assign the same pen number, width, linetype, and speed to several screen colors, causing them to be drawn similarly. You can set the plot configuration to prompt you to change pens each time a new color is needed. During configuration, answer **y** when you see this prompt: Do You Want to Change Pens While Plotting?

With raster printers such as laser or electrostatic printers that do not have physical pens, you still can plot various line widths by assigning a pen width to a color. You may want to configure these widths to match the most common pen sizes: 0.18 mm, 0.25 mm, 0.35 mm, 0.50 mm, and 0.70 mm. If your printer allows you to specify pen widths on the hardware, be sure to change the hardware settings to read the software pen-width values.

With both pen plotters and raster printers, you can associate the same line width with several layers by assigning the same pen to more than one AutoCAD LT color. For example, suppose you set up a blue and a red layer. If pen 1 has a width of 0.25 mm (whether physical or assigned), you can plot 0.25 mm lines for all objects on those layers by assigning pen 1 to both blue and red. This assignment also causes those colors to be plotted in the same color on a multiple-pen plotter.

Plotter Linetypes

Some plotters support hardware linetypes. Hardware linetypes are faster, more device-dependent, and less precise than software linetypes. They may be useful if you want to plot a different linetype without changing the linetype style in the drawing. Software linetypes are more precise. If you change hardware linetypes in the Print/Plot Configuration dialog box, avoid unexpected results by making sure that the affected objects use the CONTINUOUS linetype style. Hardware linetypes work best with narrow lines. Wide hardware linetypes produce unpredictable results on some devices.

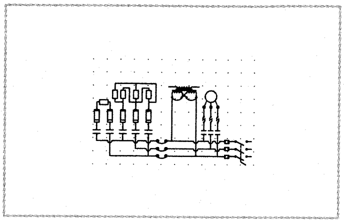
Pen Speed

For some plotters, you can adjust pen speed on a pen-by-pen basis. This feature is useful, for example, for slowing down pens that are skipping. Each pen manufacturer recommends a pen speed for each type of medium. For best results, use those values.

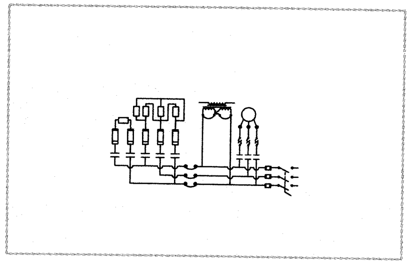
Setting Plot Area and Output Parameters

For any given drawing view, you can plot the current display, the extents of the objects, an area you define, the limits, or a named view. In model space, the options apply to the current viewport. If you are in paper space, the options apply to the paper space view. You can also specify output parameters, including plotting to a file.

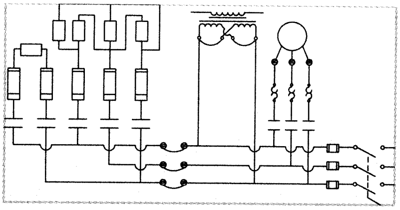
The following examples illustrate the different plot area options you can specify.



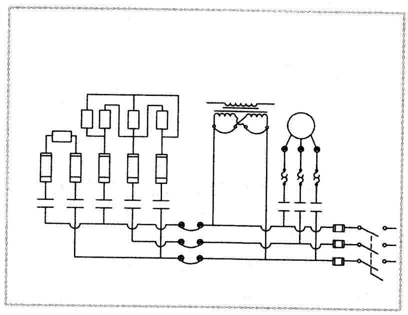
current graphics display, with a grid defining time limits



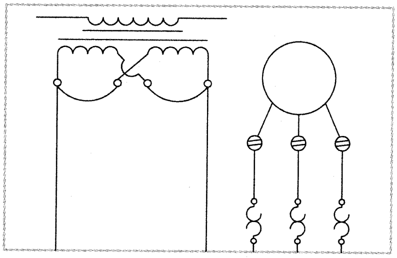
display option



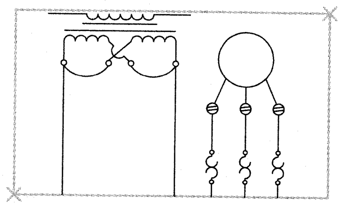
extents option



limits option



view option



window option

Examples of the options specifying the plot area

Text Resolution

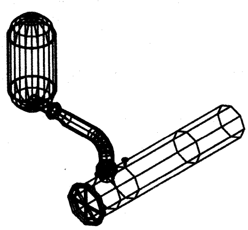
If you are working with TrueType fonts in your drawings, you can refine the appearance of text in your plotted drawings. The Text Resolution option sets the tessellation or amount of *smoothing* applied to TrueType fonts while plotting, exporting with PSOUT, and rendering. Lower values apply less smoothing and increase plotting speed. Higher values apply more smoothing and decrease plotting speed. The value for this entry (between 0 and 100) is stored in the TEXTQLTY system variable.

Text Fill

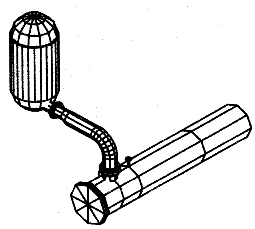
Use the Text Fill option to control the filling of TrueType fonts while plotting, exporting with PSOUT, and rendering. If Text Fill is not selected, text is output as outlines. The value for this entry is stored in the TEXTFILL system variable.

Hide Lines

If you are viewing a 3D drawing created with AutoCAD, or working with thickened objects, you also can specify whether hidden lines are plotted. The Hide Lines option affects views in the current space only. If you are plotting from paper space and want to hide lines within specific viewports, use the Hideplot option of MVIEW. See "Hiding Lines in Plotted Viewports" on page 343.



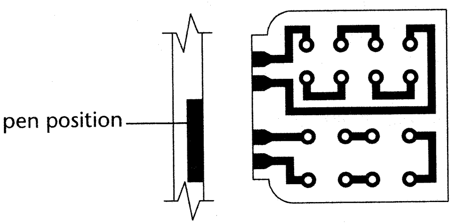
Hide Lines off



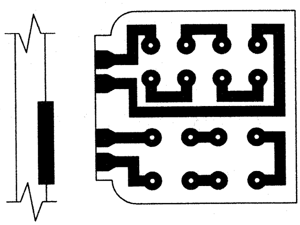
Hide Lines on

Adjust Area Fill

When you plot 2D solids and wide polylines, AutoCAD LT uses the pen-width value specified in the Pen Assignments dialog box. The pen normally aligns with the center of the object boundaries. To ensure greater accuracy, you can use the Adjust Area Fill option to move the pen inward by half a pen width.



Adjust Fill Area on



Adjust Fill Area off (default)

Use this option for drawings such as circuit diagrams in which the plot must be accurate to half the pen width.

Plot to File

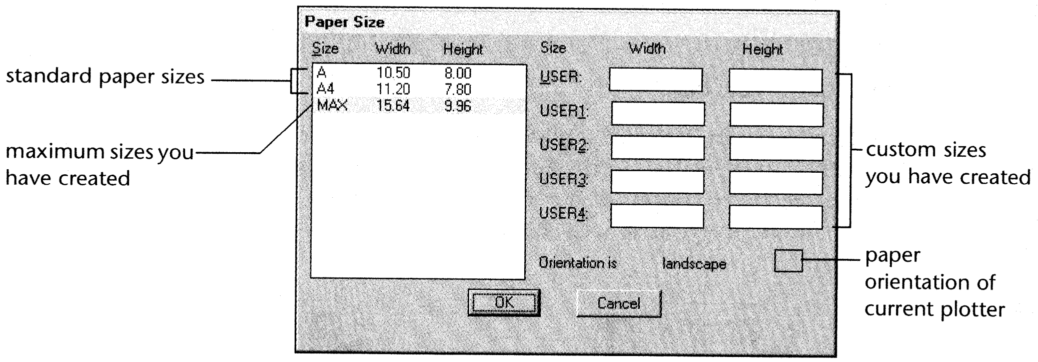
Use the Plot to File option to output to a file rather than directly to the plotter. The default plot file name is the drawing name with a *.plt* file extension. To specify a name other than the default name, choose File Name. The plot commands for the currently configured plotter are sent to the named file.

Setting Paper Size

You can choose a paper size from a standard list or create a custom size. You can base this size on millimeters or inches, depending on which option you choose. You also can specify the unit type used for plotting (millimeters or inches).

The Paper Size and Orientation area of the Print/Plot Configuration dialog box also indicates the dimensions of the plot area in the specified measurement units. If you use a Windows system printer, the paper size is determined in the Windows Control Panel. Therefore, the Size option is not available.

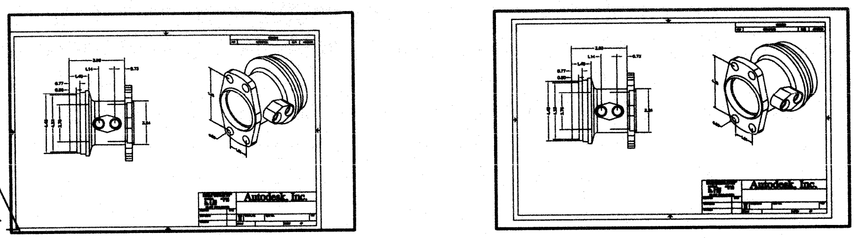
NOTE If you create a custom paper size, make sure the specified printer or plotter supports the paper dimensions. If the paper size you specify is too large, the plotted drawing may be clipped.



Positioning the View on the Paper

You can rotate the image to adjust its orientation on the paper. You also can reposition the image by changing its plot origin. The home position, or lower-left corner of the plotted area, is determined by your plotter. If necessary, you can change the plot origin (normally set to 0,0) to adjust the position of the plotted drawing relative to the home position. For example, the home position for pen plotters often is well within the edges of the paper.

You can access the plot origin settings by choosing Rotation and Origin in the Print/Plot Configuration dialog box. To plot the lower-left corner of the drawing nearer to the lower-left corner of the paper, you might set *X* Origin to –1.0 and *Y* Origin to –0.5.



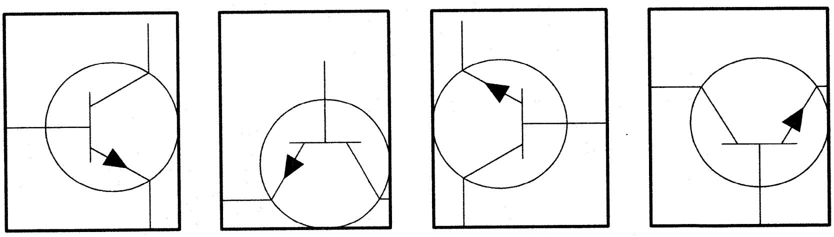
home position

lower-left corner of paper

plot with origin 0,0 plot with origin 1,2

For raster printers, the home position often is as close as 0.25 from each edge of the paper. However, many printers do not print that close to the edge. To avoid clipping off part of the drawing when you print, you can avoid drawing outside the printer-defined margins, or you can specify a new plot origin. For example, set *X* Origin and *Y* Origin to 0.5 to establish a plot margin 0.50 units in and up from the plotter's home position.

If you change the plot rotation, the home position remains in the lower-left corner of the rotated plot area. Positive plot-origin offsets shift the rotated image up or to the right. Negative plot-origin offsets shift the rotated image down or to the left.



0° plot rotation 90° plot rotation 180° plot rotation 270° plot rotation

The effect of the rotation and origin settings varies with the plotter you use. For information about the home position or clipping regions for your output device, see its documentation.

While you make adjustments, you can preview the image. In the Print/Plot Configuration dialog box under Plot Preview, select Full or Partial. Choose Preview to see the position of the image in the plotted drawing.

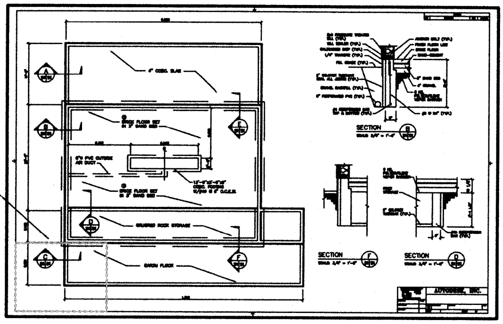
Setting the Plot Scale

You generally draw objects at their actual size. When you plot the drawing, you either specify a precise scale or fit the image to the paper.

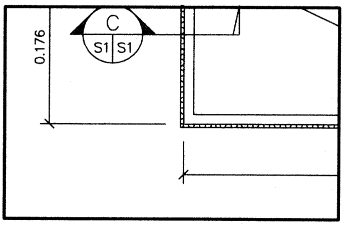
Most final drawings are plotted at a precise scale. To develop a scale, enter the ratio of plotted units to drawing units under Scale, Rotation, and Origin in the Print/Plot Configuration dialog box. You can choose the unit type, inches or millimeters, under Paper Size and Orientation. For example, if MM is selected, entering **1** under Plotted MM and **10** under Drawing Units produces a drawing in which each plotted millimeter represents 10 actual millimeters.

The following illustrations show a full-scale drawing scaled to fit a C-size sheet using both a 1/10th scale and the Scaled to Fit option.

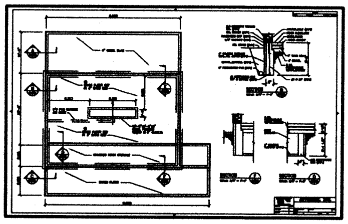
C-size paper compared with actual size



drawing shown at 1=1 scale



plotted at 1=1 on C-size paper



scaled to fit C-size paper

When you are reviewing drafts, a precise scale is not always important. You can use the Scaled to Fit option to plot the view at the largest possible size that fits the paper. AutoCAD LT fits either the horizontal or vertical dimension to the corresponding dimension of the paper. Perspective views are scaled to fit the paper, even when you enter a scale.

NOTE Do *not* use Scaled to Fit with the long-axis plots that are available on some plotters.

When you choose the Scaled to Fit option, the text boxes change to reflect the actual ratio of plotted units to drawing units. AutoCAD LT updates this scale whenever you change the size of the plotted area, unit type, paper size, plotter, plot origin, or orientation.

If you plot from paper space, you can use ZOOM Scale to adjust the scaling in viewports relative to paper space. See "Scaling Views Relative to Paper Space" on page 340.

Reusing a Plot Configuration

Before you plot, you need to specify the output device and other device-specific information, as well as settings such as paper size and scale factors. AutoCAD LT stores this information in a *plot configuration file*.

Each plot configuration contains device-specific information such as the device driver name, device model name, the output port to which the device is connected, and various device-specific driver settings. A plot configuration can also contain device-independent information, including paper size, orientation, plot scale factors, pen parameters, optimization, plot origin, and rotation. For more information, see "Setting Up a Plot Configuration" on page 354.

To save time, you may want to reuse the more complicated setups in plot configuration files. You can create plot configurations for different drawings, for each available plotter, or for plotting the same drawing on the same plotter in different ways.

For a single plotter, use *partial* plot configuration (PCP) files. A PCP file contains the device-independent configuration of a single plotter. Partial plot configuration (PCP) files can be used for exchanging device-independent plot configuration information with users of AutoCAD LT Release 2 (and AutoCAD Release 12) or AutoCAD LT for Windows 95 (and AutoCAD Release 13). PCP files can be used to merge pen parameters and device-independent settings from one device to another similar device's configuration.

If you use several different plotters, you can use PC2 files to save your setups. A PC2 file contains a *complete* plot configuration, including both device-dependent and device-independent settings. For example, by including complete plot configuration (PC2) files with drawing files that you send to another site, you can save other users from having to set up a detailed plot configuration.

You can save up to 29 plot configurations in the *aclt4.cfg* file. This file can contain configurations for 29 different plotters, or it can specify 29 configurations for a single plotter.

If you need more than 29 configurations, you can save an unlimited number of complete plot configuration (PC2) files. PC2 files contain the device-independent and device-dependent parts of a single plot configuration. If you plan to load a PC2 file from the Device and Default Selection dialog box, and you want to preserve the current default configuration, save it in another PC2 file before loading the new PC2 file. When you plot, you can choose the PC2 file you need.

NOTE Use any ASCII text editor to edit plot configuration PC2 or PCP files.

Creating a Complete Plot Configuration File

You can create and save the current plot configuration directly to a complete plot configuration file by choosing Save (Complete PC2) in the Device and Default Selection dialog box. Complete plot configurations are saved with the *.pc2* file extension. When you save the PC2 file, it copies the plot configuration information from one section of *aclt4.cfg* to a separate PC2 file.

You can also create PC2 files by choosing the Printer tab in the Preferences dialog box. Choose Save As to save the current configuration to a PC2 file. The PC2 file configuration information is added to the *aclt4.cfg* file.

Using a Complete Plot Configuration File

You can use the default configuration stored in a specific complete plot configuration file whenever you plot a drawing. Loading a PC2 file changes the current device as well as its configuration. Use a PC2 file when you want to change devices.

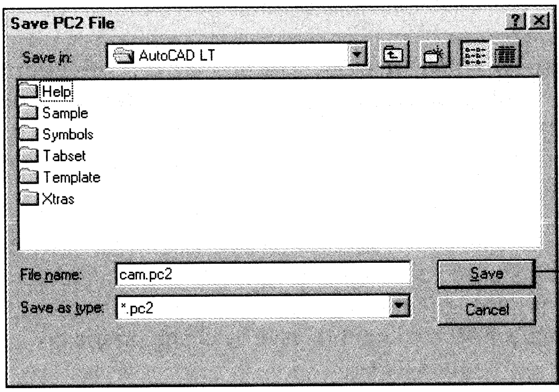
You can also add a configuration stored in the PC2 file by choosing the Printer tab in the Preferences dialog box. Choose Open to select the PC2 file you want. This adds the configuration stored in the PC2 file to the list of configurations in *aclt4.cfg*.

NOTE Adding a plot configuration from the Printer tab does *not* replace the current configuration; to change the current configuration, select a device from the list of printers and plotters, and then select Set Current. See "Setting the Current Printer or Plotter" on page 349.

ONLINE HELP See "Reusing a Plot Configuration."

Creating a Partial Plot Configuration File

You can create and save the device-independent part of the current plot configuration directly to a partial plot configuration file by choosing Save (Partial PCP-LT R2/LT 95) in the Device and Default Selection dialog box. Partial configurations are saved with the *.pcp* file extension.



Save a PCP file to reuse pen and paper settings without changing the output device.

Using a Partial Plot Configuration File

You can use the settings defined in a specific partial plot configuration PCP file whenever you plot a drawing. The information in the PCP file is merged into your current plot configuration, replacing existing settings for pen parameters, paper size and orientation, and plot scale and units.

Loading a PCP file changes the configuration but does not change the current device. For example, you can use PCP files to copy the pen parameters from one device's configuration to another or to save configurations for the same device with different paper sizes.

ONLINE HELP See "Reusing a Plot Configuration."

Plotting to a File

You can plot a drawing to a file rather than to a plotter or printer. Because you can plot later without opening AutoCAD LT, you can send the file to a plotting service or use a spooler to plot the file.

You can output files in a variety of formats; the type of file you create depends upon the device driver that you use to generate the file. Choose any of the three output options:

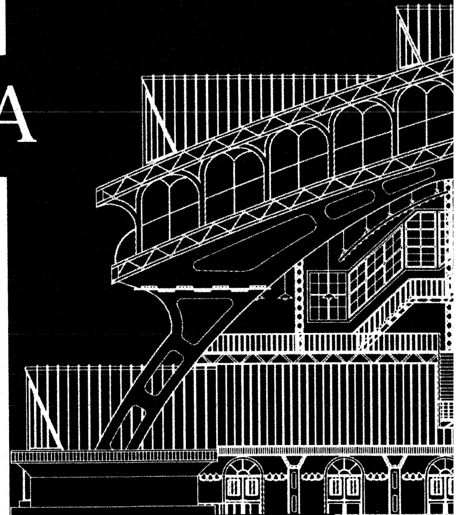
- ***Raster file export*** Use the raster file format driver to create files in BMP, TGA, PCX, and TIFF file formats. See "Configuring Raster File Output" on page 413.
- ***PostScript*** Use any PostScript printer driver and set the device to create EPS or PS file formats.
- ***Other printer or plotter drivers*** Use any other printer or plotter driver to create an output file by checking Plot to File under Additional Parameters in the Print/Plot Configuration dialog box. Your plotter driver may specify a file extension automatically, or you can specify one when you plot. Otherwise, AutoCAD LT adds a *.plt* file extension.

NOTE When you plot to file using either the raster file format driver or a PostScript driver, the Plot to File setting in the Print/Plot Configuration dialog box is checked but unavailable. The raster file format driver outputs to file only, and PostScript drivers are set to output to a file by changing the driver settings in the Windows Settings, Printers folder.

The plotted file uses the current drawing name with a different extension.

ONLINE HELP To plot a drawing to file, see "Plotting to a File." To set up the raster file format driver, see "Configuring Raster File Output."

The CAD Drawing Tutorial



In this appendix

- Setting up the drawing area
- Creating and editing a drawing at full scale
- Dimensioning the drawing
- Adding annotation
- Plotting the drawing

This tutorial provides an introduction to creating and plotting a simple drawing. Follow the step-by-step procedures—from how to start the drawing process to how to plot the final output on paper. You will begin with a hand-drawn sketch. The tutorial takes approximately one hour to complete.

For detailed information about some of the commands you will use, see online Help or the appropriate sections in this guide.

Preparing to Do the Tutorial

This tutorial guides you through the fundamental procedures to produce drawings with AutoCAD LT. In order for the tutorial to run smoothly, you need to set up the AutoCAD LT work space.

- ***Right-Click Mouse Button*** Choose Preferences from the Tools menu to display the Preferences dialog box. Make sure that the right-click mouse button option on the Work Space tab is set to display a shortcut menu.
- ***Toolbars*** Make sure that the following toolbars are displayed: Standard, Object Properties, Draw, Modify. When you first install the program, AutoCAD LT displays these toolbars by default. Choose Toolbars from the View menu to turn on any toolbars that are not displayed.
- ***Object Snap and AutoSnap*** For the purposes of this exercise, turn off any object snaps you may have set previously by choosing Object Snap Settings from the Tools menu. Then click Clear All on the Running Osnap tab of the Osnap Settings dialog box. You will turn on selected running object snap settings later in this tutorial. In the same dialog box, click the AutoSnap tab, and then make sure that the following settings are checked: Marker, Magnet, SnapTip. For details, see "Using AutoSnap" on page 133.

Setting Up the Drawing Area

Before you begin drawing on paper, you need to set up your drawing area by organizing where each specific plan or detail will go, what type of units and scale you want to use, and what size paper you need.

In AutoCAD LT setup, you specify what type of units you want to use, but you draw at "full scale." Drawing at full scale represents the actual size of what you are drawing based upon the type of drawing units you use. For example, a single unit could represent an inch or a kilometer.

You also establish drawing limits, which define an imaginary boundary, usually slightly larger than the drawing you want to make. You can see the drawing limits by setting a grid, since the grid extends to the limits.

The first exercise involves setting up the drawing area in AutoCAD LT. You will:

- Set the type of units you want to draw in
- Establish the drawing size by setting the limits
- Set up the drawing aids by setting grid and snap intervals
- Separate the drawing into components by creating layers
- Load linetypes for different types of lines

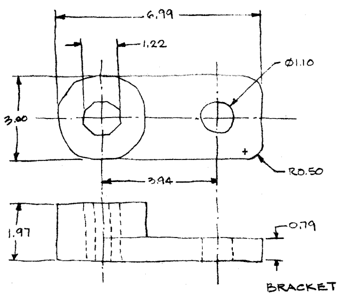
To start AutoCAD LT

1 On the taskbar, click Start, and then choose Programs.

2 Choose the AutoCAD LT 97 folder, and then AutoCAD LT 97 from the menu.

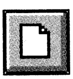
Beginning a New Drawing

You start with a hand-drawn sketch of what you will draw in AutoCAD LT.

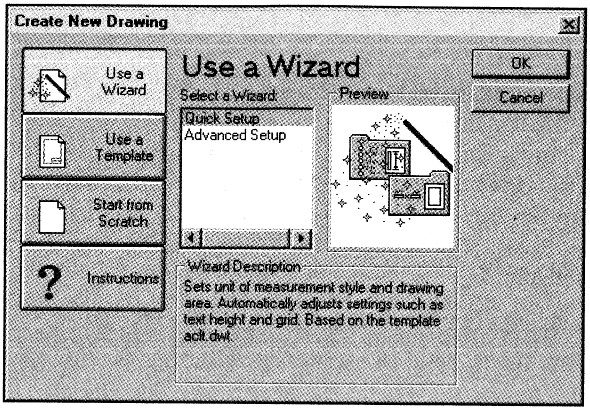


One easy way to start a new drawing is to use the Quick Setup wizard, which is one of the setup options. In this exercise, you will use the Quick Setup wizard to establish the unit type and the general size of your drawing area.

To begin a new drawing



1 The Start Up dialog box is displayed automatically when you start up AutoCAD LT. If you are already running AutoCAD LT, display the Create New Drawing dialog box by choosing New from the File menu.



2 In the Start Up or Create New Drawing dialog box, select Use a Wizard.
3 Select Quick Setup.
4 Choose OK.

The Quick Setup wizard is displayed.

Setting the Unit Type

One of your first tasks before you start to draw in AutoCAD LT is to select the type of unit you want to use for drawing: scientific, decimal, engineering, architectural, or fractional units. The unit type specifies the notation style for your drawing.

In this exercise, you will choose the Decimal style.

To set unit type

- In the Quick Setup wizard on the Step 1: Units tab, specify the unit type by selecting one of the available options. AutoCAD LT uses decimal units by default, which is the correct setting for this exercise.

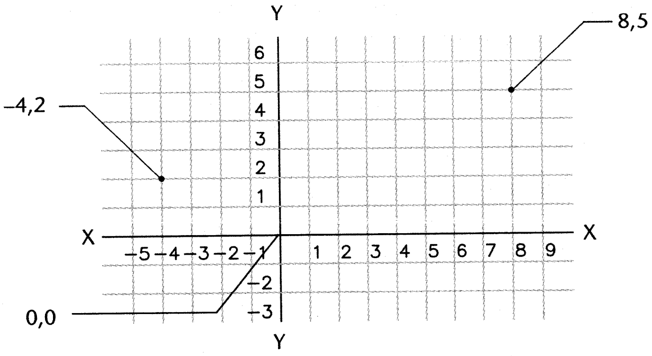
NOTE For the purposes of this tutorial, the system of measurement (English or metric) is not specified. For more information, see "Setting the System of Measurement" on page 68.

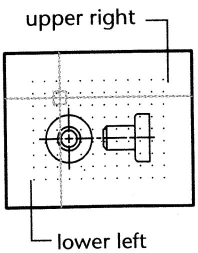
Specifying Limits

Now set your drawing boundaries. In AutoCAD LT, drawing boundaries are known as limits. If you visualize your screen as a sheet of paper, the limits represent the drawing area, including the border around the edge of the paper. Setting the limits defines the full-scale size of the objects you draw plus an invisible margin around those objects.

Limits can be set to represent any size. For example, if you want to draw a 35 × 25 hinge block, you would set your limits to be at least 50 × 40. The limits would include the 35 × 25 hinge block plus the invisible margin around your object. Limits are always specified by two coordinates.

A coordinate is a set of numbers that specifies a point location on an axis, a flat two-dimensional (2D) surface, or a drawing plane. In 2D drawing, you can specify a coordinate location with two numbers separated by a comma, such as 0,0. The first number represents the *X* value of the coordinate, which specifies horizontal distance. The second number represents the *Y* value, which specifies vertical distance.





In the following exercise, the origin point (0,0) has been automatically established for you with the Quick Setup wizard. Where the two axes meet at the origin point is equivalent to the lower-left corner of a piece of paper. By entering **12,9** for the upper-right corner, you establish the opposite corner of your drawing area.

To specify limits

1 In the Quick Setup wizard, choose Step 2: Area.

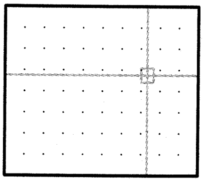
2 If the Windows Regional Settings specify U.S. as the system of measurement, AutoCAD LT sets 12,9 as the area by default, which is the correct setting for this exercise.

If the Windows Regional Settings specify Metric as the system of measurement, AutoCAD LT sets 420,297 by default. For this exercise, enter **12,** and **9** in the Width and Length boxes.

3 Choose Done to exit the wizard.

Unlike the physical limits imposed by paper, these limits can be changed at any time using the LIMITS command.

Setting the Grid and Snap



grid turned on

Grid and snap are tools that help you draw and align your objects. Grid gives you a visual representation of the drawing limits by displaying a pattern of vertical and horizontal dots on your screen. These dots are a visual aid only and are not printed. Snap restricts the movement of your pointing device so that selecting precise points is easier.

You can set grid and snap to specific values. For example, setting grid to 1 displays a pattern of dots one unit apart. Setting snap to 0.50 restricts pointing device movement to every 0.50 units, or half the grid spacing. Note that the grid display is limited by the limits values.

In the following exercise, set Snap to 0.25.

To set grid and snap

1 From the Tools menu, choose Drawing Aids.

2 In the Drawing Aids dialog box under Snap, click On. The Quick Setup wizard automatically turns grid on.

When grid and snap are turned on, you can enter values for *X* Spacing and *Y* Spacing.

3 Under Snap, enter **0.2500** next to *X* Spacing.

4 Under Grid, enter **0.5000** next to *X* Spacing.

Y Spacing changes automatically to match *X* Spacing.

5 Choose OK.

As you move your pointing device, notice that the cursor "snaps" to an evenly spaced pattern of points between the dots, or one-half the grid spacing.

Setting Unit Precision

Next, set unit precision, which controls the number of decimal places displayed. For example, a mechanical drawing might require a high degree of precision for the bore hole on a metal plate. By default, AutoCAD LT sets precision to four decimal places. In this exercise, you set the precision to two decimal places.

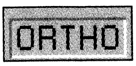
To set the unit precision

1 From the Format menu, choose Units.
2 In the Units Control dialog box under Units, click the down arrow to display the list under Precision.
3 Select a two-place (0.00) decimal precision.
4 Choose OK.

Setting Ortho

The Ortho tool restricts the cursor to horizontal and vertical movement. Ortho makes it easier for you to draw straight horizontal and vertical lines. For this exercise, set Ortho on.

To set Ortho

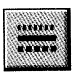


■ On the status bar, double-click Ortho.

Loading Linetypes

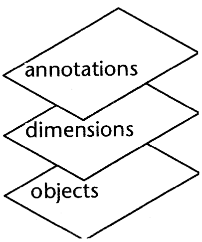
You need a continuous line, centerline, and dashed, or hidden, line to draw the bracket in this tutorial. When you begin a new drawing, to save file space and loading time, only the CONTINUOUS linetype is loaded. You must load any other AutoCAD LT linetypes before you can use them. By loading the linetypes first, you can assign different linetypes to each layer as you create it.

To load linetypes



1 From the Format menu, choose Linetype.
2 In the Layer & Linetype Properties dialog box, choose Load.
3 In the Load or Reload Linetypes dialog box, the linetype file *aclt.lin* is selected.
4 Scroll the linetype list and select CENTER. Hold down the CTRL key, scroll the linetype list again, and select HIDDEN. Then choose OK.

   The CENTER and HIDDEN linetypes are loaded. The CONTINUOUS linetype is already loaded by default.
5 Choose OK to close the Layer & Linetype Properties dialog box.



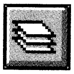
sample layers

Creating Layers

Your drawing consists of many components. By creating layers you can group various types of objects. For example, you can put objects, text, dimensions, and even title blocks on separate layers. This helps you separate plans, details, and shapes.

In the following exercise, you create five layers. You create a layer named CENTER where you draw centerlines, DIMS for dimensions, HIDDEN for hidden lines, OBJECTS for the model itself, and TEXT for annotations.

To create layers

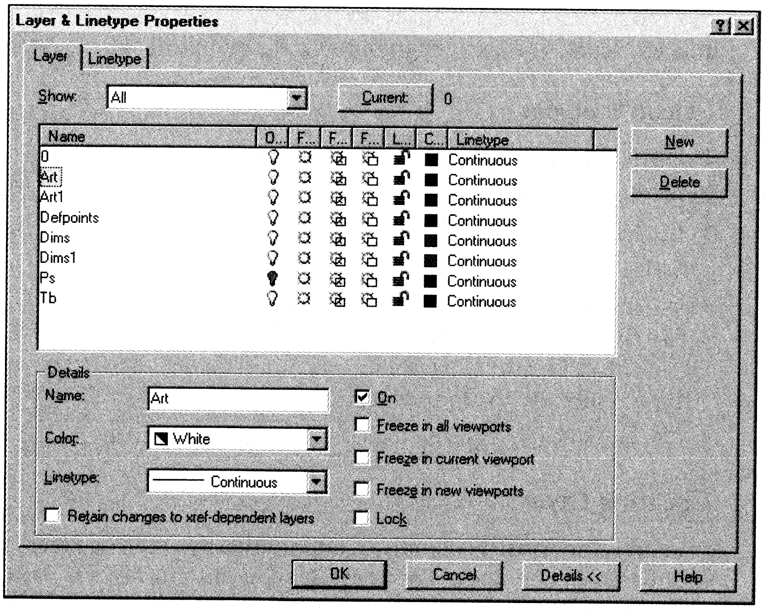


1 From the Format menu, choose Layer.
2 In the Layer & Linetype Properties dialog box, choose New.

   Notice that the layer you've created, Layer1, now appears in the Layer Name list.
3 Rename this layer by typing **center** on the highlighted name. Press ENTER to update the layer name.
4 Repeat steps 2 and 3 to create the following layers: DIMS, HIDDEN, OBJECTS, and TEXT.
5 Choose OK.

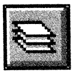
Assigning Colors and Linetypes to Layers

When you draw manually, you use centerlines, hidden lines, and heavy or light lines to distinguish parts of your drawing. With AutoCAD LT, you can assign a specific linetype and color to each layer.



NOTE The colors available depend on your video graphics card and configuration.

To assign colors and linetypes to layers



1 From the Format menu, choose Layer.
2 In the Layer & Linetype Properties dialog box, select CENTER in the Name list.
3 Under the column heading C (Color), click the colored square.

   The Select Color dialog box displays the standard nine colors and a full color palette (depending on your video graphics card).
4 In the Select Color dialog box under Standard Colors, select the Red square. Then choose OK.
5 Next, select the linetype for the CENTER layer by clicking CONTINUOUS under the Linetype column heading.
6 In the Select Linetype dialog box, select the CENTER linetype. Then choose OK.
7 For the remaining layers, DIMS, HIDDEN, OBJECTS, and TEXT, assign the colors and linetypes shown in the following table.

Layer specifications

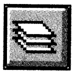
Layer name	Color	Linetype
CENTER	Red	CENTER
DIMS	Magenta	CONTINUOUS
HIDDEN	Green	HIDDEN
OBJECTS	Blue	CONTINUOUS
TEXT	Cyan	CONTINUOUS

If you are not sure which color to choose because you don't know its hue (for example, cyan), click the colors in the Select Color dialog box, and look at their names displayed in the Color text box.

Making a Layer Current

Once you've assigned a color and a linetype to each layer, you need to make a specific layer current. When a layer is current, everything you draw is drawn on that layer. Start drawing on the CENTER layer.

To make a layer current



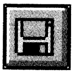
1 In the Name list, select CENTER. Then choose Current.
2 Choose OK.

Saving the Drawing

By saving the drawing, you store both the drawing itself and the drawing environment settings, such as grid, snap, and Ortho mode, loaded linetypes, and layer names. It is important that you save your drawing frequently to avoid losing it in the event of a power failure or other unexpected event.

It is good practice to save your drawing files in a directory or folder other than where you installed AutoCAD LT. For instance, you might create a folder called *project*, in which you store all your drawings for a project you're working on. In this exercise, you will save the drawing and its settings.

To save a drawing



1 From the File menu, choose Save.
2 In the Save dialog box, under File Name, enter **bracket**. Choose Save.
   Remember to check which directory you are saving your file in so that you can easily find it later.

You are ready to start drawing.

Creating and Editing Objects at Full Scale

When drawing with AutoCAD LT you draw at full scale (a 1:1 scale). You can alter the drawing easily by changing the scale.

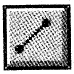
Drawing the Top View

AutoCAD LT provides several methods for drawing objects. This exercise demonstrates only one approach to drawing the sketch shown in "Beginning a New Drawing" on page 367.

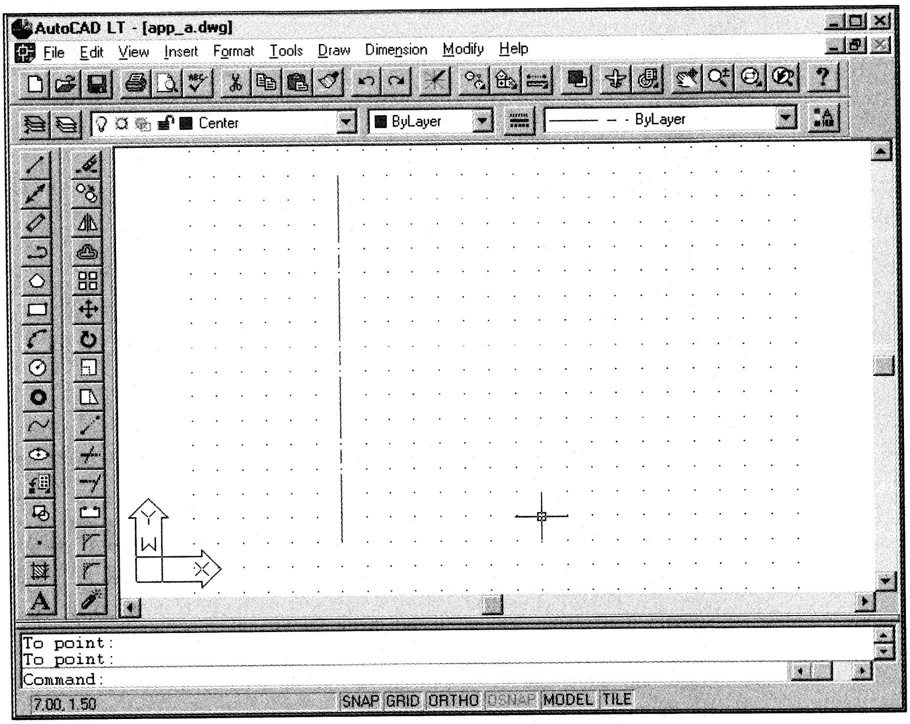
You start by drawing lines and circles to create the top view. The drawing starts to take shape when you copy the lines and circles and use fillets to round off the corners. You use object snaps to maintain accuracy.

You start by drawing lines. First, create two lines on the CENTER layer. These lines are used to place the circles in the drawing.

To draw the vertical centerline



1 From the Draw menu, choose Line.
2 Use your pointing device to specify a point near the top of your screen. (Refer to the following illustration for placement.)
3 Drag your cursor vertically down to the third grid dot from the bottom of the graphics area.
4 Click to place the endpoint of the line.
5 Press ENTER to end the LINE command.



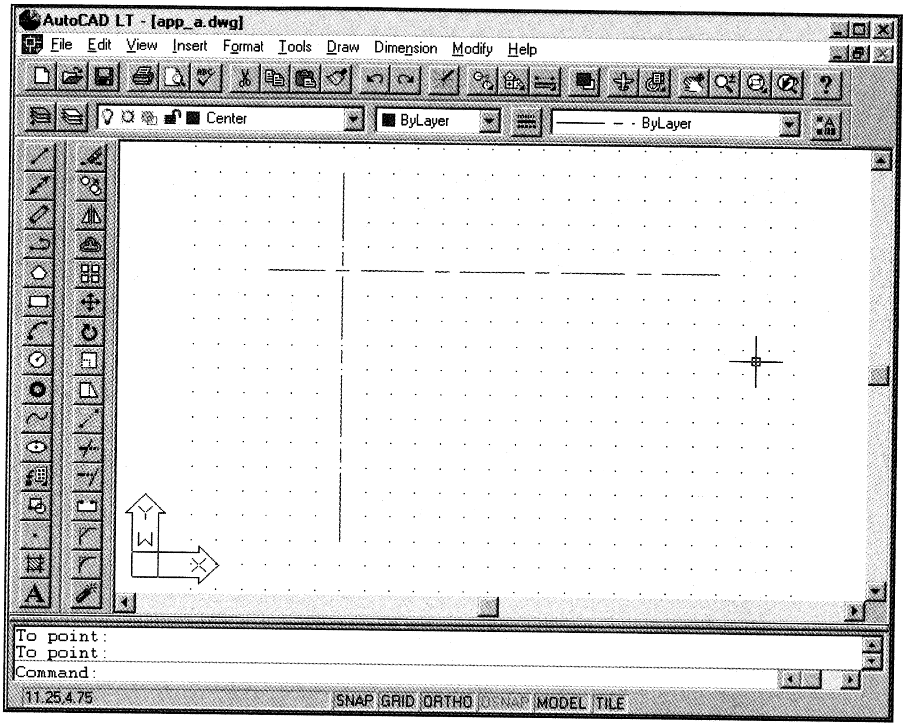
Now, draw a horizontal line to finish the centerlines.

To draw the horizontal centerline

1 Press ENTER to repeat the LINE command.
2 Click a point left of the vertical line and one quarter of the way down, or approximately at the sixth grid dot from the top of the graphics area (based on 0.5000 grid setting).

3 Drag your cursor horizontally across the graphics area and click on a point to place the endpoint of the line.

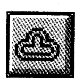
4 Press ENTER to complete the LINE command.



You can save time by using the copying feature: you draw an object once and then copy it to other locations. Using an offset creates copies of an object a specified distance and direction from the original object.

In the following exercise, you use the OFFSET command to copy the line 3.94 units away from the original line.

To offset the vertical line



1 From the Modify menu, choose Offset.

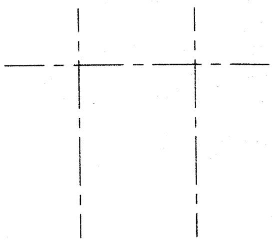
2 On the command line, type **3.94** for the offset distance. Then press ENTER.

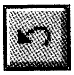
Notice that the cursor changes from crosshairs to a square box, called a pickbox. With the pickbox, you select the objects you want to copy, rotate, edit, and so on.

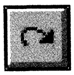
3 Select the vertical centerline.

4 Specify any point on the right side of the line.

5 Press ENTER to complete the command.







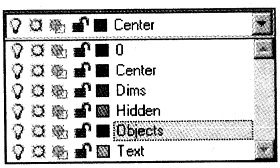
If you make a mistake while drawing the lines, choose Undo from the toolbar. (You can also enter **u** for UNDO on the command line.)

If you need to bring back the objects that UNDO removed, choose Redo from the toolbar. (You can also enter **redo** on the command line.) The REDO command must be issued immediately after UNDO.

Now that you have drawn all the centerlines, you can draw the rest of the objects. Before you start, you need to make the OBJECTS layer current. This places the rest of the drawing on the OBJECTS layer.

To make a different layer current

1 On the Object Properties toolbar, click the Layer Control arrow.



A list of layers is displayed.

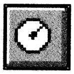
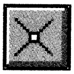
2 Select the layer called OBJECTS.

To begin drawing the top view of the object, you draw two circles.

Object snaps help you draw with precision. When you select an object, the cursor snaps to a geometric point, such as the center point of a circle. For example, if you want to draw a line from the center point of an existing circle, use the Center object snap. If you select the edge of the circle, the cursor snaps to the center point of the circle as the start point of the line.

In this exercise you draw a circle by specifying its center and radius, using the Intersection object snap to specify a center point.

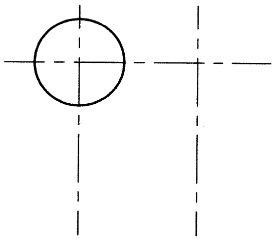
To draw the large circle

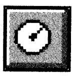
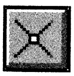
1. From the Draw menu, choose Circle ➤ Center, Radius.
2. To display the cursor menu, press SHIFT and right-click your pointing device. Then choose Intersection.

   Notice that the crosshairs change to an **X** as you move the cursor over the intersection of the centerline.
3. Select a point near the intersection of the centerlines on the left to specify the center of the circle.

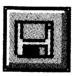
   Although the centerlines are on a different layer, you can still use them as snap points when drawing on the OBJECTS layer.
4. On the command line, Radius is the default. Enter **1.5** for the radius.
5. Press ENTER to complete the circle.



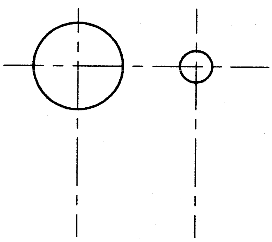
To draw the small circle

1. Press ENTER to restart the CIRCLE command.
2. From the cursor menu, choose Intersection.
3. Select a point near the intersection of the centerlines on the right to specify the center of the circle.
4. On the command line, enter **.55** for the radius.
5. Press ENTER to complete the circle.



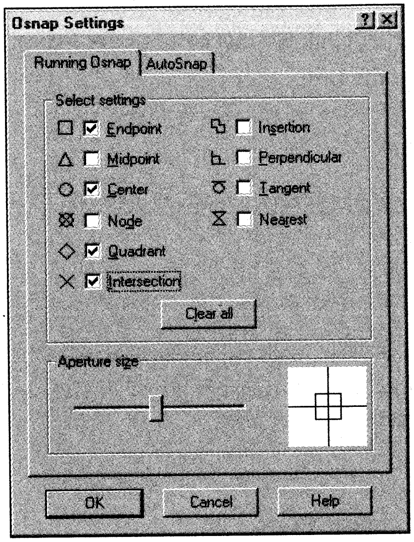
Save your drawing.



Now that you have used single object snaps for accurate placement of objects, you can set AutoCAD LT to identify specific object snaps automatically whenever your pointing device moves over the objects in your drawing. For the purposes of this exercise, you set the running objects snaps for the following points: Center, Endpoint, Quadrant, Intersection.

To set running object snaps

1 From the Tools menu, choose Object Snap Settings.

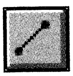


2 In the Osnap Settings dialog box on the Running Osnap tab, check the boxes next to Center, Endpoint, Quadrant, and Intersection.

3 Choose OK.

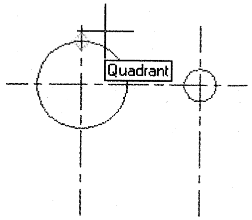
Having created the circles, you can now start on the lines of the object. You can draw one line, and then copy it to another location. The next procedure gives you more practice with object snaps and teaches you how to copy an object.

To draw the top line



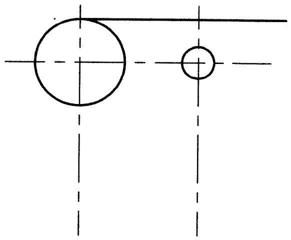
1 From the Draw menu, choose Line.

2 Move the cursor to a point near the top (90-degree) point of the larger circle to specify the top quadrant point of the circle. The quadrant AutoSnap marker is displayed at the circle's quadrant. Click to specify the first point of the line. Notice that the AutoSnap magnet automatically locates the circle's quadrant; AutoCAD LT does the precision work for you.



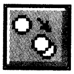
quadrant AutoSnap marker and SnapTip displayed at the circle's top quadrant

3 Move the pointing device to draw a line about the same length as the horizontal centerline, and then click to select the endpoint of the line.
4 Press ENTER to complete the line.

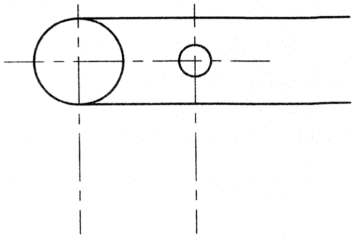


Next, copy the line at the lowest (270-degree) point of the circle. Use the Quadrant object snap for precise placement.

To copy a line



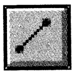
1 From the Modify menu, choose Copy.
2 Click the line at the top of the circle to select it.
3 Press ENTER to complete object selection.
4 For a base point, select the top quadrant of the large circle. Move the cursor to a point near the top (90-degree) point of the larger circle to specify the top quadrant point of the circle. The quadrant AutoSnap marker is displayed at the circle's quadrant. Click to specify the base point.
5 For the displacement point, select the bottom quadrant of the large circle. The quadrant AutoSnap marker is displayed at the circle's quadrant. Click to specify the displacement point.



Finishing the Top View

Only a few steps remain to finish the top view. In this exercise, you round off the corners of the bracket by using a fillet, which connects two nonparallel objects with a smooth arc. You also draw an eight-sided hole inside the larger circle.

To draw a vertical line

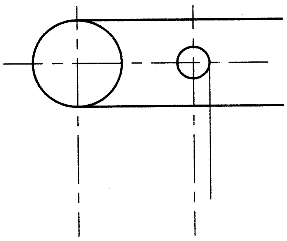


1 From the Draw menu, choose Line.

2 To begin the line at the right quadrant of the small circle, move the cursor to a point near the right (0-degree) point of the small circle to specify the right quadrant point of the circle. The quadrant AutoSnap marker is displayed at the circle's quadrant. Click to specify the right quadrant.

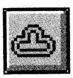
3 Move the cursor down vertically. Then specify a point to draw the endpoint of the line.

4 Press ENTER to complete the line.



Next you can offset the line at a distance of one unit from the original line.

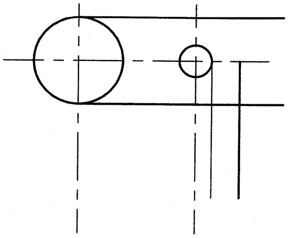
To offset the vertical line

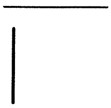


1 From the Modify menu, choose Offset.

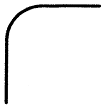
2 On the command line, enter **1** for the offset distance.

3 Select the vertical line on the small circle.
4 Specify a point on the right side of the line.
5 Press ENTER to complete the command.





before FILLET

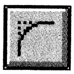


after FILLET

With two lines to work with, you can create a fillet to round off the corners of your bracket. Creating a fillet involves two steps; setting the radius of its arc, and then selecting the objects to fillet.

Before you can create a fillet, you set the radius of its arc. The value for a radius can be from 0 to an unlimited value restricted only by the size of your object. A fillet with a 0 radius is equivalent to a square corner. Any other value will round the corners. In this procedure, you round off the corners of the bracket using a fillet with a radius of 0.50.

To set the fillet radius

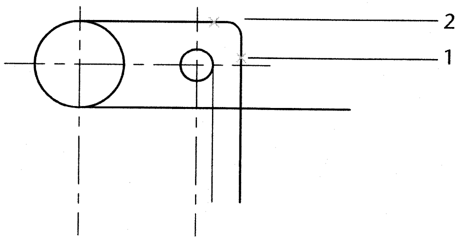


1 From the Modify menu, choose Fillet.
2 On the command line, enter **r** (Radius). Then press ENTER.
3 Enter **.5** for the radius of the fillet. (Because this value is the default for the radius, you may not have to enter anything.) Then press ENTER.

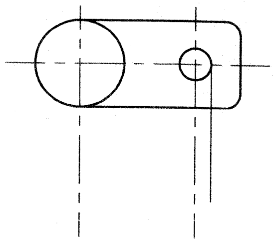
Now you need to select the objects that you want to fillet.

To fillet two lines

1 Press ENTER to repeat the FILLET command.
2 Select the offset vertical line on the right (1).
3 Select the line at the top of the large circle (2).

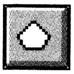


4 Follow steps 1 through 3 to create another fillet for the line at the bottom of the larger circle.

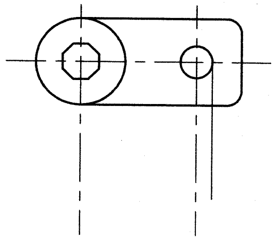


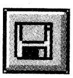
Now create an eight-sided hole inside the larger circle by drawing a polygon with a radius of 0.61. Use the Center object snap in this exercise.

To draw a polygon



1 From the Draw menu, choose Polygon.
2 On the command line, enter **8** to specify the number of sides.
3 Move the cursor to the circumference of the larger circle to specify the Center object snap point of the circle. By default, the quadrant AutoSnap marker is displayed when you first position your cursor. Press the TAB key on your keyboard until the center AutoSnap marker is displayed at the circle's center. Click to specify the center point.
4 Enter **i** to specify to draw a polygon that is inscribed within the circle radius you specify.
5 On the command line, enter **.61** for the radius of the circle whose circumference is tangent to the polygon segments.



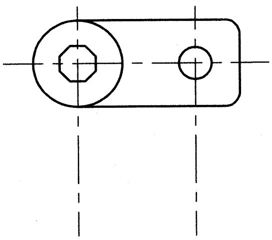


Save your drawing.

Now that you have finished part of the top view, you may notice that there's an unnecessary construction line on the right quadrant of the smaller circle. You can use the ERASE command to delete unwanted objects.

To erase an object

1 Select the vertical line at the right quadrant of the small circle to activate its grips.
2 Press DEL to delete the line.



Exiting AutoCAD LT

If you want to do the rest of the exercise later, exit the program.

To exit AutoCAD LT

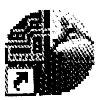
- From the File menu, choose Exit.

If you have not saved your most recent changes, choose Yes when the dialog box is displayed.

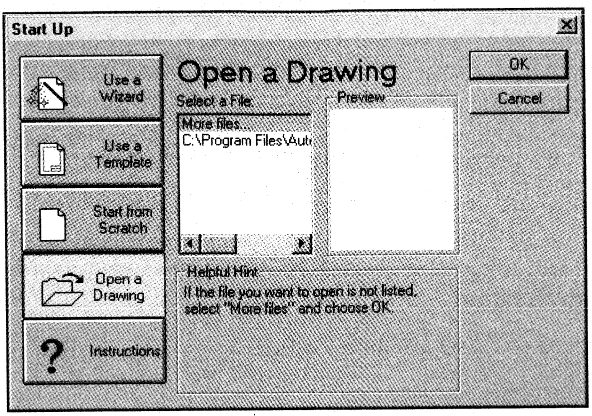
Opening an Existing Drawing

Once a drawing file exists, you can open it at any time. When you open a drawing, AutoCAD LT loads all the drawing settings and any objects that you have created since the last time the file was saved.

To start AutoCAD LT



1 From the Start menu, choose Programs ➤ AutoCAD LT 97. Then choose the AutoCAD LT icon.
2 In the Start Up dialog box, select Open a Drawing. Then choose OK.



A list of drawing files is displayed under Select a File.

3 If the file you are looking for is displayed in the list, select it, choose OK, and skip the remaining steps of this procedure. Otherwise, double-click More files.

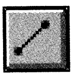
4 In the Select File dialog box, navigate to the folder in which you created the drawing. Select *bracket.dwg* from list of files displayed, using the scroll bars if necessary. Then choose OK.

Drawing the Front View

You can use the top-view drawing as a guide to draw the front view. A front view is a view looking at the front of an object. You use the LINE and OFFSET commands, as well as the TRIM command, to complete the front view.

In this exercise, you first need to draw some lines near the bottom of the screen.

To draw a line

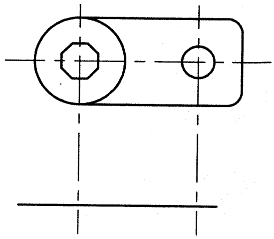


1 From the Draw menu, choose Line.

2 Specify a point approximately one quarter of the way up from the bottom of the screen, near the left edge of the graphics area, using the start point of the horizontal centerline as a guide.

3 Move your cursor horizontally from one side of the screen to the other. Specify a point before the edge of the bracket for the endpoint of the line, as shown in the illustration.

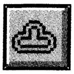
4 Press ENTER to complete the command.



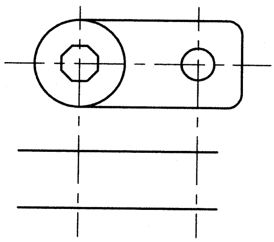
If the line is not long enough, don't worry. You can use the EXTEND command to join the lines.

Next, you need to offset the line for the top portion of the front view of the bracket.

To offset a line



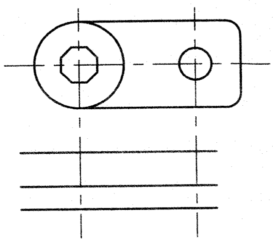
1 From the Modify menu, choose Offset.
2 On the command line, enter **1.97** for the offset distance.
3 Select the horizontal line at the bottom of the screen.
4 Specify a point anywhere above the line.
5 Press ENTER to complete the command.



Now you should have two lines, as shown in the illustration. To create the middle line, use OFFSET again to copy the line from the bottom of the front view.

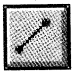
To offset a line

1 Press ENTER to repeat the OFFSET command.
2 On the command line, enter **.79** for the offset distance.
3 Select the same horizontal line at the bottom of the screen.
4 Specify a point anywhere above the line.
5 Press ENTER to complete the command.



Now that you have baselines for the front view of the bracket, you can draw construction lines, called xlines, from the top view to the front view. These lines help you align the top view with the front view. Using the Quadrant and Endpoint object snaps, draw the lines to complete the figure.

To draw the left xline from the large circle



1 From the Draw menu, choose Line.
2 Move your pointing device to a point near the left (180-degree) point of the larger circle to specify the left quadrant. The quadrant AutoSnap marker is displayed at the circle's quadrant. Click to specify the first point of the line.
3 Drag your cursor down vertically. Then click a point to specify the endpoint of the line, as shown in the illustration on page 388.
4 Press ENTER to complete the line.

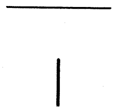
To draw the right xline from the large circle

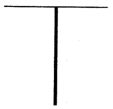
1 Press ENTER to repeat the LINE command.
2 Move your pointing device to a point near the right (0-degree) point of the larger circle to specify the right quadrant. The quadrant AutoSnap marker is displayed at the circle's quadrant. Click to specify the start point of the line.
3 Drag your cursor down vertically. Then click a point to specify the endpoint of the line, as shown in the next illustration.
4 Press ENTER to complete the line.

To draw the right end of the bracket

1 Press ENTER to repeat the LINE command.
2 Move the cursor to a point near the lower endpoint of the right vertical edge of the top view to specify the endpoint of the line. The endpoint AutoSnap marker is displayed at the line's endpoint. Click to specify the start point of the line you are drawing.

3 Move the cursor down vertically. Click a point to specify the endpoint of the line, as shown in the illustration on page 388.
4 Press ENTER to complete the line.


before EXTEND


after EXTEND

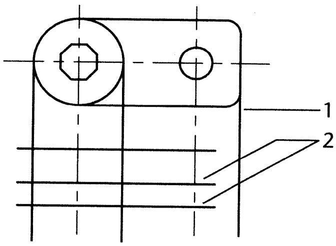
Now that you have drawn the vertical ends of the bracket, notice that the horizontal lines do not overlap or touch the right vertical end line. The EXTEND command extends objects such as lines and arcs to other objects that you define as boundary edges.

In this exercise, you extend the bottom and the middle horizontal lines to the right vertical end line of the bracket.

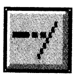
You might find it easier to select the lines with snap off.

To turn Snap off

- On the status bar, double-click Snap.



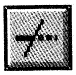
To extend the horizontal lines



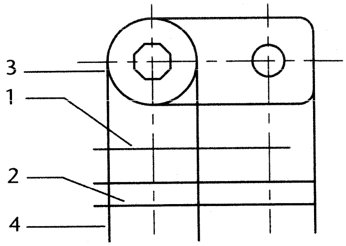
1 From the Modify menu, choose Extend.
2 Select the vertical line at the right end of the bracket as the boundary edge (1). Then press ENTER.
3 Click near the right edge of the bottom and the middle horizontal lines to select the objects to extend (2).
4 Press ENTER to end the command.

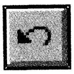
Now that all the lines overlap or touch one another, you can trim or cut the lines to create the outline of the front view. Using TRIM removes portions of lines.

To trim lines where they intersect



1 From the Modify menu, choose Trim.
2 Select the top (1) and bottom (2) lines of the front view as the cutting edges. Then press ENTER to complete the object selection set.
3 Select the lines (3, 4) as shown in the illustration for the objects to trim.

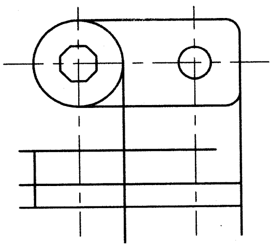




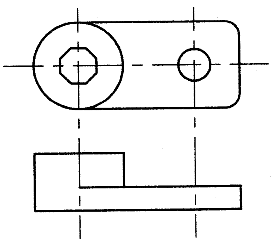
If you make a mistake, choose Undo from the toolbar.

4 Press ENTER to end the command.

Your drawing should look like the following illustration.



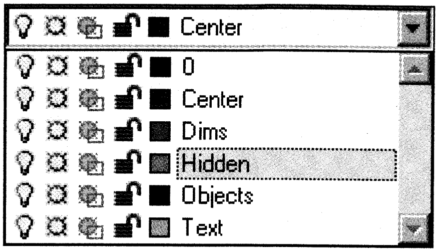
5 Use TRIM to remove the rest of the lines to complete the figure as shown in the following illustration.



To represent the holes in the plan view, you draw hidden lines. First, you'll need to make HIDDEN the current layer.

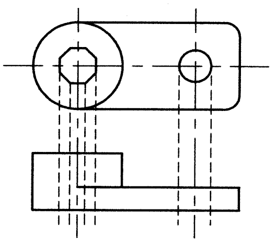
To make a new layer current

1 On the Object Properties toolbar, click the Layer Control arrow.



2 From the layer list, select HIDDEN.

Next, you will draw construction lines from the polygon and the small circle to represent the holes in the front view, as shown in the following illustration.



To draw the hidden lines from the polygon

1 From the Draw menu, choose Line.

2 Move the cursor to a point near the right edge of the polygon in the top view of the bracket. The endpoint AutoSnap marker is displayed at the edge's lower endpoint. Click to specify the start point of the line you are drawing.

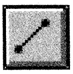
3 Move the cursor down vertically. Then click a point to draw the endpoint of the line.

4 Press ENTER to complete the line.

5 Draw the remaining vertical lines from the edges of the polygon by repeating steps 1 through 4.

Now draw lines from the small circle.

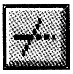
To draw the hidden lines from the small circle



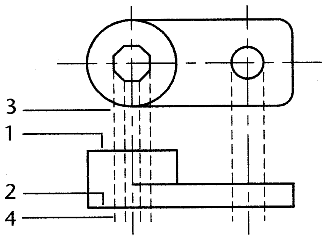
1 From the Draw menu, choose Line.
2 Move the cursor to a point near the right (0-degree) quadrant of the small circle. The quadrant AutoSnap marker is displayed at the circle's quadrant. Click to specify the start point of the line you are drawing.
3 Move the cursor down vertically. Then click a point to draw the endpoint of the line.
4 Press ENTER to complete the line.
5 Draw the left vertical line from the left quadrant of the small circle by repeating steps 1 through 4. In step 3, select the 180-degree point of the circle to start your line.

Next, using TRIM, remove the unwanted portions of the lines.

To trim lines where they intersect



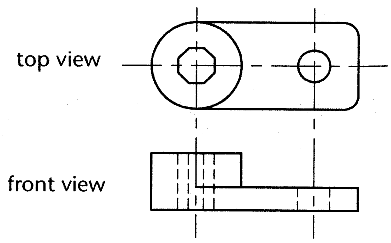
1 From the Modify menu, select Trim.
2 Select the top and bottom lines of the front view as the cutting edges (1, 2), as shown in the illustration. Press ENTER to complete the object selection set.
3 Select one of the hidden lines of the front view for the object to trim (3, 4).



4 Press ENTER to end the command.

Use TRIM to remove the rest of the hidden lines to complete the figure as shown in the following illustration.

Save your drawing.



Now that you've completed the top and front views, you are ready to dimension your drawing.

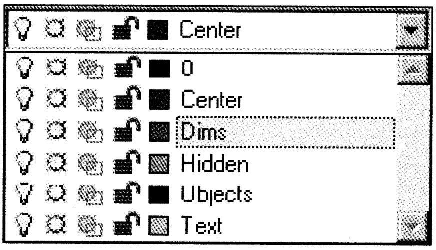
Dimensioning the Drawing

Dimensions show the exact measurement for your objects. With the help of object snaps, you can dimension each detail.

By placing all your dimensions on the DIMS layer, you can separate them from the objects you have drawn. Separating your dimensions gives you more precise control over their appearance. The dimensions for this drawing consist of horizontal, vertical, radius, and diameter associative dimensions.

To make the DIMS layer current

1 On the Object Properties toolbar, click the Layer Control arrow.



2 From the layer list, select DIMS.

You will dimension the top view using linear dimensioning. Use object snaps to specify the first and second extension line origins.

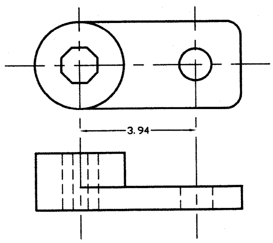
Before you add dimensions to the drawing, you can change the dimension style to suit your particular needs. In this procedure, you change the default precision and specify the placement of arrowheads.

To change the default dimension style

1 From the Format menu, choose Dimension Style.
2 In the Dimension Styles dialog box, choose Format.
3 In the Format dialog box under Fit, select Text and Arrows.
   When space between the extension lines is limited, AutoCAD LT places the both text and arrowheads outside the extension lines.
4 Choose OK.
5 In the Dimension Styles dialog box, choose Annotation.
6 In the Annotation dialog box under Primary Units, choose Units.
7 In the Primary Units dialog box under Dimension, select 0.00 as the precision.
8 Choose OK to close each dialog box.

To create a linear horizontal dimension

1 From the Dimension menu, choose Linear.
2 Move the cursor to a point near the intersection of the left vertical centerline and the lower quadrant of the large circle to specify the first extension line origin. The intersection AutoSnap marker is displayed at the intersection of the large circle and the centerline. Click to specify the first extension line origin.
3 Move the cursor to a point near the intersection of the other vertical centerline and the lower quadrant of the small circle to specify the second extension line origin. The intersection AutoSnap marker is displayed at the intersection of the small circle and the centerline. Click to specify the second extension line origin.
4 Using your pointing device, specify a location for the horizontal dimension. Refer to the following illustration.



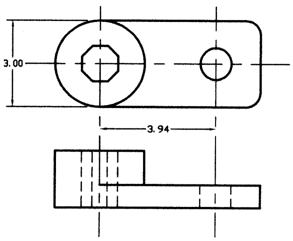
In the illustration, the extension lines may not be visible, because they coincide with the construction lines.

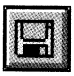
NOTE If you are using Metric units, your dimensions may not look exactly like the ones in the illustrations.

You use vertical dimensioning and object snaps to create the next dimension.

To create a linear vertical dimension

1 From the Dimension menu, choose Linear.
2 In the top view, move the cursor to a point near the top quadrant of the large circle to specify the first extension line origin. The quadrant AutoSnap marker is displayed at the circle's quadrant. Click to specify the first extension line origin.
3 Move the cursor to a point near the bottom quadrant of the large circle to specify the second extension line origin. The quadrant AutoSnap marker is displayed at the circle's quadrant. Click to specify the second extension line origin.
4 Using your pointing device, specify a point for placement of the vertical dimension.

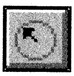




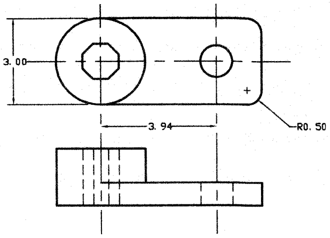
Save your drawing.

To dimension the fillet on the top view, use radial dimensioning. Radial dimensions automatically place an *R* symbol in front of the dimension text.

To create a radial dimension

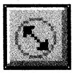


1 From the Dimension menu, choose Radius.
2 In the plan view, select one of the fillets.
3 Using your pointing device, specify a point for placement of the radial dimension.



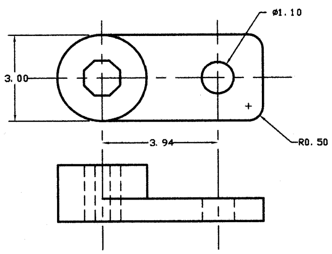
To dimension the small circle in the top view, use diameter dimensioning. Diameter dimensions automatically place a ∅ symbol in front of the dimension text.

To create a diameter dimension

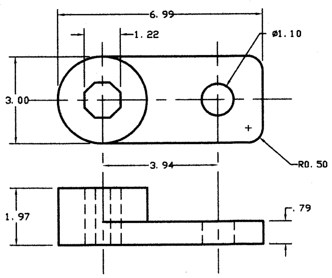


1 From the Dimension menu, choose Diameter.
2 Select the smaller circle.
3 Using your pointing device, specify a location for placement of the diameter dimension.

   Because of the changes you made to the dimension style, text and arrowheads are both placed outside the circle.



4 Using linear dimensioning, continue to dimension the top and front view as shown in the following illustration.



For the 0.79 linear dimension at the lower-right corner, specify the endpoint at the bottom of the vertical line first, and then specify the top of the vertical line.

Adding Annotations

After you complete a drawing, you can add text or annotations. You can enter a single line or multiple lines of text and see the text on the screen as you enter it. A variety of text fonts are available, and each text font can be assigned to a specific text style. For example, the font style contained in the file *txt.shx* is assigned to the default text style named Standard. If you want to use different text fonts in your drawing, you must assign a unique text style name to each font.

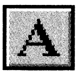
In this exercise, you will use the default text style and font to add the drawing name and your name at the lower-right corner of the drawing. Before starting, set the current layer to TEXT to ensure that all annotations are placed on the correct layer.

To make a layer current

1 On the Object Properties toolbar, select Text in the Layer Control.

Next, you can create several lines of text.

To create multiple lines of text



1 From the Draw menu, choose Paragraph Text.

2 Specify a point near the lower-right corner of your drawing area.

This is the start point for a boundary that defines the width of the paragraph of text.

3 Move the cursor down and to the right and then click to specify the right-hand limit of the bounding box.

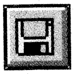
The Multiline Text Editor dialog box appears.

4 Enter **BRACKET** and press ENTER to start a new line.

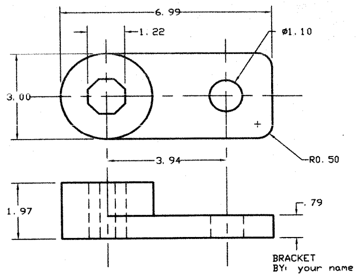
5 Enter **BY:** ***Your Name*** and press ENTER.

6 Choose OK.

Notice that the text wraps according to the width of the bounding box you specified.



7 From the File menu, choose Save to save your drawing.

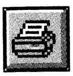


Plotting the Drawing

You are now ready to print or plot your drawing on paper. Although there are more advanced methods of printing available for 3D or complex drawings, these methods are not necessary for a simple 2D drawing such as this one.

Before you start, you will need to make sure the printer or plotter is turned on and the correct size paper is set for use. If you are using a pen plotter, check to see that the pens are working and are secure in the pen holder.

To select a printer or plotter



1 From the File menu, choose Print.

The Print/Plot Configuration dialog box is displayed. If you have more than one printer or plotter available, you need to set which printer or plotter you want to use.

2 In the Print/Plot Configuration dialog box, choose Device and Default Selection to display a list of the current printers or plotters.

3 In the Device and Default Selection dialog box, select the printer or plotter you want to use as your default printer or plotter.

4 Depending upon which printer or plotter you have chosen, use *one* of the following methods:

- If you have selected Default System Printer as your current device, click Change under Device Specific Configuration to set the paper size and orientation.
- If you have selected a printer or plotter other than Default System Printer, click Change to make any modifications to the device settings.

5 Choose OK.

The Print/Plot Configuration dialog box is displayed again.

Once you have selected the printer or plotter, you need to select the paper size. The Default System Printer is the default printer for AutoCAD LT.

NOTE Because Windows has its own paper size and orientation controls, the Paper Size and Orientation features are not available when AutoCAD LT is configured for the Default System Printer.

To select a paper size for printers other than the Default System Printer

1 In the Print/Plot Configuration dialog box under Paper Size and Orientation, select Inches or MM (millimeters) for the units you want to be printed or plotted.

2 Under Paper Size and Orientation, choose Size.

In the Paper Size dialog box, select the size of the paper that is currently loaded in your printer or plotter.

3 Choose OK.

Having selected the paper size, you can select the area you want to be printed or plotted. In the following exercise, you will use the drawing extents as your plotting area.

To set the plot area

- In the Print/Plot Configuration dialog box under Additional Parameters, select Extents.

Once the plotting area is established, you can set the plotting scale, as well as the plot rotation and the origin. Although you've drawn the objects at their full size, you can plot the drawing either to a specific scale or to fit the paper size.

You are going to plot this drawing at 0.50=1, that is, half scale.

To set the plotting scale

1 In the Print/Plot Configuration dialog box under Scale, Rotation, and Origin, clear the Scale to Fit check box.

2 Enter **0.50** in the box below Plotted Inches. Make sure that 1 is the setting in the Drawing Units box.

You can save time and paper by previewing the plot before it is printed. There are two preview options: Partial and Full. The Partial preview shows an outline representation of the plotting area in relation to the paper size. The Full preview displays the plotting area as it will be plotted on the paper.

To preview a drawing

1 In the Print/Plot Configuration dialog box under Plot Preview, select Full. Then choose Preview.

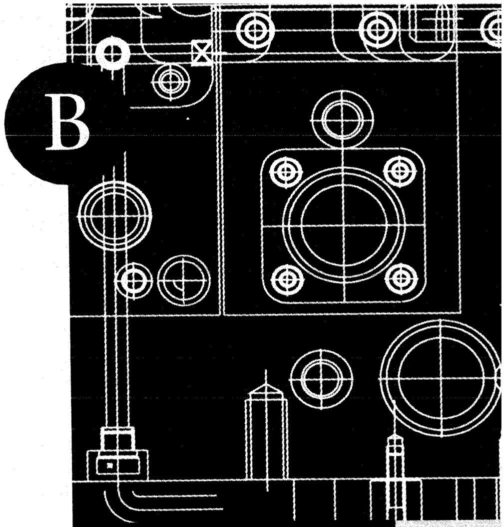
2 In the Plot Preview dialog box, right-click to display the shortcut menu, and choose Exit.

   The drawing is displayed as it will look when plotted. If the drawing did not fit on the paper during a preview, you can change the plot rotation to 90 or set the plotting scale to another value. If the drawing was not centered on the paper, you may need to change the plot *X,Y* values to a coordinate other than 0,0. Once you have made any adjustments, do another plot preview.

Once you have set the correct plotting scale and previewed the drawing, you are ready to plot your drawing.

To send the drawing to the printer or plotter

- In the Print/Plot Configuration dialog box, choose OK.



B

Configuring Plotters and Printers

AutoCAD LT supports many plotters and printers for producing hard-copy output of your drawings. In addition, you can output to files in a variety of formats. This chapter provides basic printer and plotter configuration information. Configuration information specific to a supported device, such as hardware connections, switch settings, cabling diagrams, and software settings, can be found in online Help under "Configuring CalComp, Hewlett-Packard, and Océ Plotter Drivers."

In this appendix

- Supported output devices
- Using the Preferences dialog box for basic configuration
- Using the Windows system printer
- Outputting raster files
- Setting device time-out values
- Testing pen configurations

Using Drivers for Plotter Support

AutoCAD LT uses Autodesk Device Interface (ADI) drivers to communicate with output devices. These drivers fall into three categories: file format drivers, nonsystem drivers, and the system printer driver.

ADI is a standard specification used by manufacturers and software developers to write drivers that allow communication between AutoCAD LT and peripheral hardware. Use ADI drivers to ensure the best output results when printing, plotting, or generating output files from AutoCAD LT.

ADI System Printer Driver

The system printer driver, *plsys.dll*, acts as a gateway to any previously installed Windows system printers. The printers or plotters can be locally connected or networked. The *plsys.dll* file sends raster information to a raster-capable Windows system printer.

ADI Nonsystem Drivers

Nonsystem drivers plot directly to many types of printers and plotters. The printers or plotters can be locally connected or networked. Models marked "obsolete" might not be supported in future releases. Models marked with an asterisk (*) support raster output.

Nonsystem drivers

Driver file	Plotter or printer	Model
plpcc.dll ADI 4.3	CalComp DrawingMaster plotter	800 53436 600 53336 Plus 52436 Plus 52236 Plus 52424 Plus 52224
plpcc.dll ADI 4.3	CalComp Electrostatic plotter	68444 Color EPP 68436 Color EPP 58444 Color EPP (obsolete) 58436 Color EPP (obsolete) 58424 Color EPP (obsolete) 67436 Monochrome 57444 Monochrome (obsolete) 57436 Monochrome (obsolete) 57424 Monochrome (obsolete)

Nonsystem drivers (*continued*)		
Driver file	**Plotter or printer**	**Model**
plpcc.dll ADI 4.3	CalComp pen plotter	1043 (obsolete) 1044 (obsolete) 1077 (obsolete) Artisan 1023 (obsolete) Artisan 1026 (obsolete) Artisan 1025 (obsolete) DesignMate 3024 DesignMate 3036 Pacesetter 2024 Pacesetter 2036 Pacesetter Classic 4036
plpcc.dll ADI 4.3	CalComp Solus plotter	Solus 4 LED 54436 Solus 4 LED 54424
plpcc.dll ADI 4.3	CalComp TechJET inkjet plotter	TechJET 5536 TechJET 5524 TechJET 175i 5336i TechJET 5336GT TechJET 5336GT-PS TechJET Color 5336 TechJET Color 5324 TechJET 720c 5636 TechJET 720c 5624 TechJET 720 5436 TechJET 720 5424
plphplp.dll ADI 4.2	Hewlett-Packard plotter (HP-GL)	7475 7550 7580 7585 7586 7586B & 7596A Roll Feed DraftPro DXL DraftMaster I DraftProEXL
plhpgl2.dll ADI 4.3	Hewlett-Packard ink-jet plotter (HPGL/2)	DesignJet 755CM* DesignJet 750C* DesignJet 750C PLUS* DesignJet 700* DesignJet 650C* DesignJet 350C* DesignJet 250C* DesignJet 330* DesignJet 230* DesignJet 220* DesignJet 200* DesignJet 600

Nonsystem drivers (*continued*)		
Driver file	**Plotter or printer**	**Model**
plhpgl2.dll ADI 4.3	Hewlett-Packard printer (HP-GL/2)	LaserJet III LaserJet 4 PaintJet XL300
plhpgl2.dll ADI 4.3	Hewlett-Packard pen plotter (HP-GL/2)	DraftMaster Roll Feed DraftMaster Plus Sheet Feed DraftMaster Sheet Feed DraftPro Plus HP 7600 Color (obsolete) HP 7600 Monochrome (obsolete)
plphip.dll ADI 4.2	Houston Instrument DMP plotter	DMP-51 (obsolete) DMP-51MP (obsolete) DMP-52 DMP-52MP (obsolete) DMP-56 (obsolete) DMP-56MP (obsolete) DMP-61 DMP-61MP (obsolete) DMP-62 DMP-62MP DMP-161 DMP-162 DMP-162R
plppost.dll ADI 4.3	PostScript laser printer	300 dpi* 600 dpi* 1270 dpi* 2540 dpi*
plpoce.dll ADI 4.3	Océ plotter device	G9034-S* G9035-S* G9054-S* 9055-S/95xx-S* 5104* 5105* 5100C A1 EC_2.x* 5100C A0 EC_2.x* 5120 A1 LZ_1.x* 5120 A0 LZ_1.x* 5200* 9400 LV_3.x* 9700 R1.0* 9800 R3.x*

File Format Drivers

File format drivers send data to files and do not print on paper.

File format drivers

Driver file	File format
plexport.dll ADI 4.3*	BMP PCX TGA TIFF

Using the ADI System Printer Driver

With the Windows system printer driver, you can use whatever plotter or printer you have configured with Windows. Any printing device supported by Windows can be used with the Windows system driver configured as the AutoCAD LT plotter.

The system printer driver, *plsys.dll*, acts as a gateway to any previously installed Windows system printers. The printers or plotters can be locally connected or networked. The *plsys.dll* file sends raster information to a raster-capable Windows system printer.

For pen plotters, it is recommended that you use the ADI nonsystem driver appropriate to your device rather than the Windows system printer driver. For raster output devices, it is recommended that you use the Windows system printer driver.

NOTE The system printer driver supports raster output. However, the ability of the device connected as the system printer to output raster and vector data sent by AutoCAD LT may be limited.

If you have multiple Windows system printers, you can select the device from which to plot from within AutoCAD LT. This feature is useful, for instance, for using a LaserJet printer for word processing documents and a BubbleJet for AutoCAD LT drawings.

Configuring the ADI System Driver

Setting up the Windows system printer for AutoCAD LT consists of two parts:

- Configuring the system printer in Windows (see the Microsoft documentation for your operating system)
- Configuring the Windows system printer as the AutoCAD LT plotter (see the following procedure)

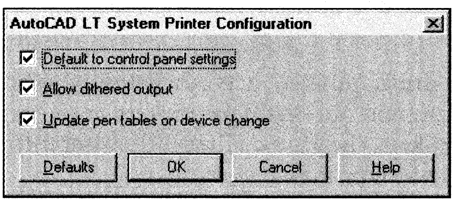
To configure the system printer as the AutoCAD LT plotter

1 From the File menu, choose Printer Setup.

2 By default, the Windows system printer is configured for AutoCAD LT. If it is not listed on the Printer tab in the Preferences dialog box, choose New.

3 In the Add a Printer dialog box, select System Printer ADI 4.3—by Autodesk, Inc.

4 Enter a description in the Add a Description box. Then choose OK.



5 In the AutoCAD LT System Printer Configuration dialog box, check or clear the following check boxes as required. Then choose OK.

- ***Default to Control Panel Settings*** Specifies the Control Panel settings when you plot. Clear this option to use the printer settings you selected the last time you plotted with the system printer.
- ***Allow Dithered Output*** Specifies print in gray scale on devices that print only black. For color devices, the pure-color option limits the number of colors plotted to the number of colors the device supports. The dithering option instructs the Windows printer driver to simulate more colors using pixel patterns from the palette of pure colors the device supports.
- ***Update Pen Table On Device Change*** Specifies that you want to update your pen assignments when you choose a new printer using the Control Panel.

6 Respond to the prompts in the text window until you are returned to the Preferences dialog box. Then choose OK to close the Preferences dialog box.

If you use the Windows system printer, all handshaking operations and the I/O port handling are managed by Windows. You cannot change the I/O port used by AutoCAD LT by using the AutoCAD LT configuration process. You must do this through the Windows Control Panel.

It is possible to reconfigure AutoCAD LT at plot time to send your plot to a different Windows system printer.

To send the plot to a different Windows system printer

1. From the File menu, choose Print.
2. In the Print/Plot Configuration dialog box, choose Device and Default Selection.
3. In the Device and Default Selection dialog box under Device Specific Configuration, choose Change.
4. In the Print Setup dialog box under Name, select the printer you want to use and other print settings.

To configure multiple Windows system printers

1. From the File menu, choose Print.
2. In the Print/Plot Configuration dialog box, choose Device and Default Selection.
3. In the Device and Default Configuration dialog box, select the printer you just configured. Then choose Change.
4. In the Print Setup dialog box, select a different printer from the Printer list.
5. Adjust any other settings in the Print Setup dialog box. Then choose OK.
6. Choose OK to close the Device and Default Configuration dialog box.
7. Adjust any other settings in the Print/Plot Configuration dialog box and choose OK.

   You must actually complete a plot for your changes to be saved.

When you are using a Windows system printer, the printer configuration is specified by settings in the Windows Print Setup dialog box. AutoCAD LT is not in control of these settings.

If you reuse an AutoCAD LT saved plot configuration for a Windows system printer, the current Windows settings for the printer apply. AutoCAD LT restores only the settings saved in AutoCAD LT.

Configuring a Windows System Network Printer

You can print to any networked Windows system printer.

To plot using a Windows system network printer

1 Set the Windows system printer driver as your current plotter.

2 From the File menu, choose Print.

3 In the Print/Plot Configuration dialog box, choose Device and Default Selection.

4 In the Device and Default Selection dialog box under Device Specific Configuration, choose Change.

5 In the Print Setup dialog box under Printer, select any configured Windows system printer, either local or networked.

   In Windows 95, this list contains the printers you have configured using Print Manager. In Windows NT, you can also browse for any shared printer on the network.

Using a Nonsystem Driver

For pen plotters, it is recommended that you use the nonsystem driver appropriate to your device rather than the Windows system printer driver.

AutoCAD LT uses ADI 4.2 and 4.3 plotter drivers for AutoCAD LT running on Windows. The plotter driver names are in the form *pl*.dll*. If you have a plotter and the Windows system printer, you can configure both of them as AutoCAD LT plotters and select one of them when you plot a drawing.

Configuring a Nonsystem Driver

During the configuration of nonsystem drivers, you are prompted to specify the kind of port to which the device is connected: serial (local), parallel (local), or network.

If the hard-copy device has only one type of port, you are offered only two choices, the relevant local port (serial or parallel) or a network connection.

If the hard-copy device is connected to a local port, you are prompted for the port to which it is connected. For parallel ports, the default is LPT1. For serial ports, the default is COM1. Change the port name if your device is connected to a different port. You can specify the port by name or by hexadecimal address.

If you specify that the device is connected to a network port, AutoCAD LT displays the Browse for Printer dialog box, in which you can specify a network printer. This usually involves browsing for the computer to which the device is connected and then browsing for the particular remote plotter.

If you choose a network printer connected to a computer that is running a different version of Windows, you might see a one-time prompt to install the Windows system driver for the remote device on your system.

To view a remote plotter on the network, you must install a Windows system printer driver for the plotter on the remote computer and set permissions so that the plotter is shared. If the device is connected to the remote system using a serial port, you will have to use the Windows Control Panel to set the serial port parameters correctly to match the device's communication parameters and cabling. This involves setting the serial port's baud rate, data bits, parity, stop bits, and handshaking. All of these must be properly configured before the device can be used. Verify that you can plot or print with the device by using the Windows system printer driver.

You can now use an ADI driver to plot to the device across the network. The Windows system printer driver is not involved in this, except to make the device browsable. Of course, the ADI driver must be configured for the specific make and model of the device in use.

Some printers or plotters require advanced settings that are controlled by commands entered at the Command prompt. This guide discusses the basic configuration of nonsystem device drivers only.

ONLINE HELP For information on port, cabling, and switch settings, see "Configuring CalComp, Hewlett-Packard, and Océ Plotter Drivers."

To configure a nonsystem driver as the AutoCAD LT plotter

1 From the File menu, choose Printer Setup.
2 On the printer tab in the Preferences dialog box, choose New.
3 From the list of available printer drivers in the Add a Printer dialog box, select the ADI nonsystem driver that supports the output device you are using. For specific information on the supported output devices, and the specific ADI drivers, see the table "Nonsystem drivers" on page 402.
4 Enter a description in the Add a Description box. Then choose OK.

   The AutoCAD LT text window is displayed so you can configure the new printer.
5 Respond to the prompts.

When you have responded to all the prompts, the Preferences dialog box is displayed again. The name of the printer you just configured is displayed in the list.

6 To make the new printer current, select the printer name from the list and choose Set Current.

7 Choose OK.

Configuring a Nonsystem Driver for a Network Plotter in Windows NT

In the Windows NT environment, you can set up a network plotter using one of the following techniques with a nonsystem driver.

To plot to a Windows NT network resource

1 Configure a nonsystem driver for the desired port.

2 Open a DOS command prompt, or choose Run from the Windows Start menu. Use the `net use` command to set up the redirected port by entering

```
net use LPTx \\computer\share /persistent:yes
```

where *x* is the port number; *computer* is the computer name; and *share* is the share name.

For more information about the `net use` command, see your Windows NT online documentation.

To plot to a Windows NT device file

1 Set up a redirected port as described in the preceding procedure.

2 Plot to a file named *lpt*x, where x is the port number that has been redirected.

Resolving Windows Print Manager Conflicts

If you configure a nonsystem driver for a locally connected plotter and you also configure a Windows system printer for the same locally connected plotter, the driver is unable to connect directly to the local port, because the Windows system has control over it. The driver output is rerouted to the Windows spooler.

To plot when your nonsystem driver output is rerouted, you must perform the following tasks:

- If the plotter is connected to a serial port, be sure that the Windows Control Panel settings for that serial port are correct. They should match the settings of the plotter and should be appropriate for the cabling you have used. You can verify that these settings are correct by printing to the conflicting Windows system printer.
- Refer to the information provided online in "Configuring CalComp, Hewlett-Packard, and Océ Plotter Drivers" for your specific device.

Plotting from a nonsystem driver through the spooler is similar to plotting to a file because there is only one-way communication with the plotter. Performance varies depending on the device.

Setting the Time-Out Value for Devices

The time-out value determines how long the plotter can instruct AutoCAD LT to stop sending data while it empties its buffer. After the plotter empties its buffer, it accepts more data from AutoCAD LT. When you are prompted for a time-out value, specify how much time you want to elapse before AutoCAD LT prompts you to abort the plot. If your drawings are complex, or contain OLE objects, or if your pen speed is very slow, set the time-out value higher than the default (30 seconds). If you begin to receive numerous time-out prompts, your time-out setting is probably too low. You can set the time-out value during initial configuration or during configuration at plot time.

The following procedure applies only to nonsystem drivers when plotting to a local device.

To set the time-out value for local devices

1. From the File menu, choose Printer Setup.
2. In the Preferences dialog box on the Printer tab, choose Modify.
3. In the Reconfigure a Printer dialog box, select Reconfigure.

   The AutoCAD LT text window is displayed.

4 Press ENTER until the time-out value prompt is displayed and enter the number of seconds.

How many seconds should AutoCAD LT wait for the plotter port to time-out (0 means wait forever), 0 to 500 <30>:

5 If the plotter pause is longer than the time-out setting, the following message is displayed.

An error has occurred while writing to the hardcopy device. Please check that it is turned on, online, and has paper. Abort? <N>

6 Press ENTER.

The following procedure applies to network plotting, plotting to files, or plotting to Windows system printers.

To set the time-out value for networked devices or the system printer on Windows 95 or NT 4.0

1 From the Start menu, choose Settings ➤ Printers.
2 Right-click the printer you want, and then choose Properties.
3 In the Properties dialog box, choose the Ports tab.
4 On the Ports tab, select the LPT port that the printer uses, and then choose Configure Port.
5 In Transmission Retry, enter the number of seconds.

Testing Color and Pen Configuration

Many plotters provide multiple pens, programmable linetypes, and programmable pen speeds. For such plotters, you may choose a pen number, linetype, and pen speed to use for all 255 AutoCAD LT drawing colors. You may establish permanent settings or change them when you begin a plot.

For some plotters, you can change the pen width. If your plotter supports configurable pen widths, you can set them in the Pen Width column of the Pen Assignments dialog box. For most plotters the default is 0.010 inch or 0.254 millimeter.

You can change the pen widths individually for each AutoCAD LT color, or you can change a group of them at once. If your plots are too dark or too low in resolution, or if the dimension arrows look odd, the pen width setting may be too wide.

The drawing *chroma.dwg* in the *sample* directory contains a grid of color swatches representing 255 colors. It is recommended that you plot *chroma.dwg* to see the colors your plotter assigned to the AutoCAD LT colors.

Configuring Raster File Output

With the raster file format driver, AutoCAD LT can export raster files in formats listed in the following table, with options and variants allowed within those standards.

The driver behaves as a plotter, and represents the width and height of the image in pixels. The screen sizes are correct for pixel aspect ratio. If you choose one of these screens, you shouldn't have to calibrate to correct the results.

Supported raster file formats

File format	Options
Device-independent bitmap for Microsoft Windows (BMP)	Monochrome, 16-color, and 256-color
TrueVision (TGA) Targa	TGA 8, 16, 24, and 32 formats Bottom-up or top-down order of scan lines Noninterleaved, 2-to-1, and 4-to-1 line interleaving image compression
PCX format	Monochrome, 16-color, and 256-color
Tagged image file format (TIFF)	Monochrome, 16-color, and 256-color

After you select the output file format and supply the unique data for each configuration for that format, you are prompted to select the color resolution of the file. The following procedure is an example of the configuration format for a raster output file.

To configure for output to raster files

1 From the File menu, choose Printer Setup.

2 On the Printer tab in the Preferences dialog box, choose New.

3 In the Add a Printer dialog box, select Raster file export ADI 4.3—by Autodesk, Inc.

4 Enter a description in the Add a Description box. Then choose OK.

The AutoCAD LT text window is displayed so you can configure the new printer.

5 Respond to the prompts.

   When you have responded to all the prompts, the Preferences dialog box is displayed again. The name of the printer you just configured is displayed in the list.

6 To make the new printer current, select the printer name from the list and choose Set Current.

7 Choose OK.

NOTE The background color that the driver uses for generating images can be set only when the driver is configured to output color images.

BMP File Format

The BMP files on Microsoft Windows are device-independent bitmap files. They can be read and written by Windows Paint, and virtually all Windows applications import them. AutoCAD LT creates BMP files according to the definition given in the *Microsoft Windows SDK Reference Manual*. There is a configuration option for background color.

TGA File Format

AutoCAD LT writes all of the commonly used TGA formats. To configure the format for a raster output file, use the following procedure.

To configure for output to TGA files

1 From the Tools menu, choose Preferences.

2 On the Printer tab in the Preferences dialog box, choose New.

3 In the Add a Printer dialog box, select Raster file export ADI 4.3—by Autodesk, Inc.

4 Enter a description in the Add a Description box. Then choose OK.

   AutoCAD LT displays prompts in the text window.

5 Select a resolution. Then choose TrueVision TGA format.

6 At the prompts that follow, respond as necessary to meet your output requirements.

- Choose the Targa display board you have.
- Choose whether to use compression, change scan-line order, and interleave lines.
- Specify a background color.

7 AutoCAD LT displays the current plot settings. If you want to change any settings, enter **y**.

8 In the Preferences dialog box, choose OK.

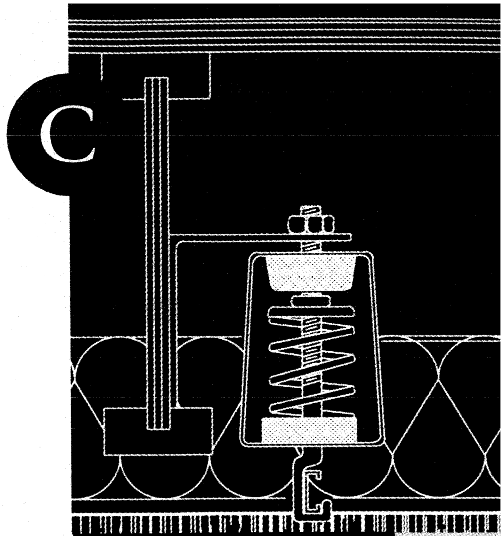
PCX File Format

The PCX file format has as its ancestor the files written by the original PC Paintbrush program, Z-Soft, which currently markets various paint products for the PC.

TIFF File Format

TIFF (tagged image file format) is a raster file format jointly defined by Aldus and Microsoft. AutoCAD LT writes "Little-endian" (Intel byte order) TIFF files containing bitmaps, 4-bit pixels, or 8-bit pixels, depending on the color resolution you select.

C



Configuring a Digitizing Tablet

In this appendix

- Configuring the Wintab digitizing tablet driver
- Configuring and calibrating the digitizing tablet
- Testing tablet calibration
- Reinitializing the digitizing hardware

This appendix provides the information you need to configure a digitizing tablet using the Wintab driver. Each section has information that helps you configure your digitizing tablet to work with AutoCAD LT. A basic tablet configuration with default selections is provided.

AutoCAD LT can be set up to accept input from both a digitizing tablet and your mouse, or a tablet can be used in place of a mouse as a system pointer to make full use of the graphical interface.

Configuring the Wintab Driver

AutoCAD LT supports Wintab-compatible digitizing tablets. The digitizing puck or stylus can be used in place of a mouse as a system pointer, and it can also be used as an absolute pointing device to trace drawings into AutoCAD LT (by turning TABLET mode on).

Wintab is a Windows specification that independent developers use so that the digitizing tablet can be used as both a system pointer and an absolute pointing device. The ADI Wintab driver developed by Autodesk is installed with AutoCAD LT. It provides an interface so a Wintab-compatible digitizing tablet can be used as a system pointer to choose menu items and drawing objects in AutoCAD LT.

To configure Windows 95 or Windows NT 4.0 for the Wintab driver provided with the digitizer, follow the setup installation procedure supplied by the digitizer manufacturer. Wintab drivers are not distributed with AutoCAD LT. The digitizer must work properly with Windows to work with AutoCAD LT. You can use a digitizer with AutoCAD LT only by configuring Windows for use with the Wintab driver. Make sure that the Wintab driver is configured for the correct digitizer model and the correct number of buttons on the puck or stylus.

Windows NT must have both a Wintab System driver and an ADI Wintab driver compiled for 32-bit Windows.

With Windows 95 you can use either a 16- or 32-bit Wintab driver but only with the 32-bit ADI Wintab driver. If you use a 16-bit Wintab driver with Windows 95, it is necessary to install a driver in order for the 32-bit ADI Wintab driver to call the 16-bit system resources of the 16-bit Wintab System driver.

The following table displays the modules required to run Wintab drivers on both platforms. The translation program is provided by the Wintab driver manufacturer and should be on the diskette that accompanies the driver. For more information or to obtain the Wintab driver, contact the hardware manufacturer.

Required Wintab module drivers

Platform	System driver	Translation program	ADI Wintab driver
Windows 95	16-bit 32-bit	Required None	32-bit 32-bit

Required Wintab module drivers (*continued*)			
Platform	**System driver**	**Translation program**	**ADI Wintab driver**
Windows NT 4.0	32-bit	None	32-bit

Using a Mouse versus a Digitizing Tablet

You can configure AutoCAD LT for a mouse, a digitizing tablet, or both. Before you configure the digitizing tablet, the entire tablet area is a fixed screen-pointing area. Inside a screen-pointing area, the tablet pointing device (puck or stylus) acts like a mouse.

Using Mouse Devices

A mouse is a relative screen pointing device. Because it does not provide a one-to-one correspondence between the mouse position and the drawing, it cannot be used to trace existing paper drawings or to support tablet menus. A mouse usually returns coordinates relative to its last placement. This makes a mouse the best tool to use to access the Windows graphical user interface.

Although you can use Microsoft Windows and AutoCAD LT without a mouse, a pointing device is recommended to take full advantage of the AutoCAD LT graphical interface.

To configure the mouse

1 On the taskbar, click Start.
2 From the Start menu, choose Settings ➤ Control Panel.
3 In the Control Panel window, double-click the Mouse icon.
4 In the Mouse Properties dialog box, you can swap the functions of the left and right mouse buttons, change the mouse tracking speed, and alter the double-click speed of the mouse. By convention, the left mouse button is the pick button.

Using Digitizing Tablets

A digitizing tablet is an absolute pointing device: each point on the tablet has a one-to-one correspondence to a specific location in the drawing, making the tablet the most useful tool for digitizing drawings in AutoCAD LT. Tablets are typically configured in two ways, for a tablet overlay or digitizing existing drawings or photographs.

The purpose of calibration is to establish a proportional relationship between the tablet surface and the actual size of the object being drawn. After a tablet is calibrated, it can be used to trace geometry from an existing drawing or photograph into an AutoCAD LT drawing, simultaneously creating accurate dimensions in the AutoCAD LT model. Points digitized from the drawing or photograph will represent actual coordinates. In AutoCAD LT, using a tablet in this manner is called Tablet mode. You can turn Tablet mode on and off by using TABLET, choosing Tablet on the Tools menu, or by pressing F4 on your keyboard.

When the tablet is configured, and Tablet mode is off, portions of the tablet surface are designated as menu areas and a screen-pointing area. A tablet configured with the AutoCAD LT tablet overlay provides convenient access to the most common AutoCAD LT drawing tools and can be customized to include your favorite features and macros. Cells A1 through I25 on the AutoCAD LT tablet overlay are intentionally left blank to accommodate user-defined shortcuts and macros.

Before you configure the tablet, attach the tablet overlay to the tablet so that it does not move during use, assuring that you track to the menu areas that you configure.

NOTE Although the tablet overlay is not required for tablet configuration, it is recommended that you use it.

Using a Tablet in Multiple Sessions of AutoCAD LT

Running multiple sessions of AutoCAD LT on Windows NT and Windows 95 is typically very similar to running single sessions. If you start each AutoCAD LT session from the same configuration file, the tablet is shared between sessions, with the active session (title bar highlighted) receiving tablet input. Only the AutoCAD LT session that is currently active can receive keyboard input. You can share one tablet among multiple AutoCAD LT sessions, or each AutoCAD LT session can have its own physical tablet, with each hooked to a different I/O port.

AutoCAD LT does not restrict the use of multiple sessions with a tablet. However, you must configure the tablet in the first session to be able to use it in subsequent sessions.

Configuring Fixed and Floating Screen-Pointing Areas

AutoCAD LT uses the concept of screen-pointing areas for tablets. A screen-pointing area is a rectangular region on the tablet surface within which the tablet pointer mimics system mouse movement. The screen areas map to the computer display *absolutely.* In other words, when the cursor is within a screen-pointing area, the tablet pointer can access windows, menus, and other applications outside the AutoCAD LT drawing area. You can configure a maximum of two screen-pointing areas for a tablet; one fixed and one floating.

After you configure AutoCAD LT, but before you configure your tablet, the entire surface of the tablet is actually the *fixed screen-pointing area.* At that point, the tablet pointer performs like a mouse. You can change the size of the fixed screen-pointing area by specifying a new size using the cfg (Configuration) option of TABLET. It is recommended that you specify a small screen area on your tablet that maps to the entire screen on your monitor.

If you want to turn on Tablet mode and digitize points into AutoCAD LT, you cannot use a fixed screen-pointing area. When Tablet mode is on, there is a one-to-one correspondence between the digitizer and the drawing.

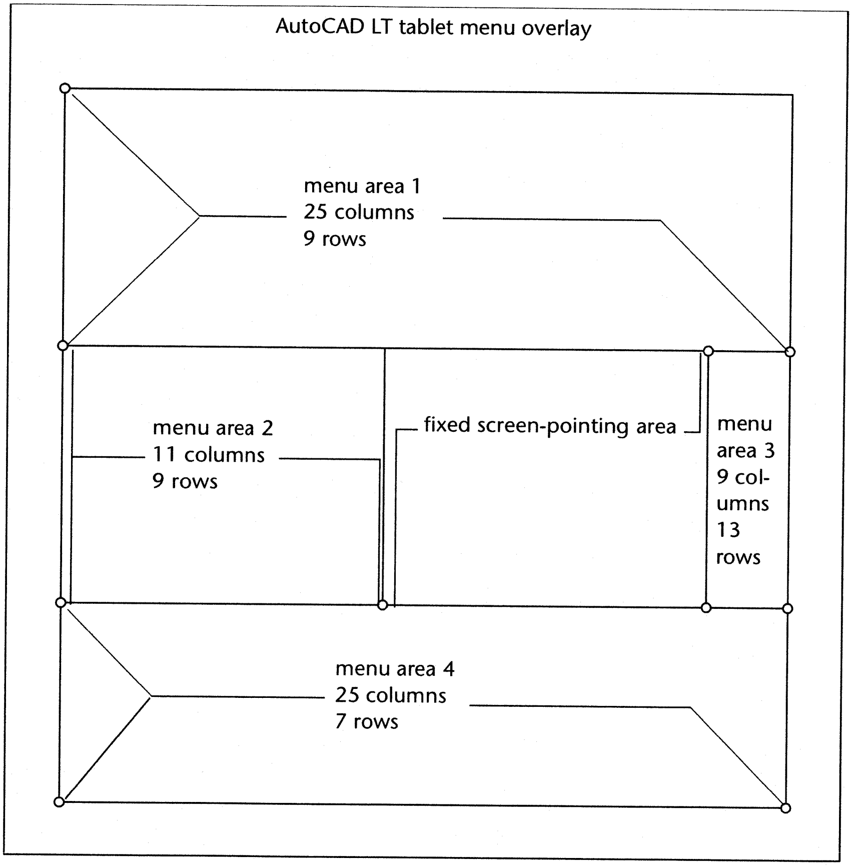
You can configure the *floating screen-pointing area* to expand the size of the pointing area on the tablet for creating and selecting objects. With a floating screen-pointing area, you can access toolbars and pull-down menus anywhere on the tablet surface and toggle them off when you have finished. When prompted, you can choose to specify the size of the floating screen area and special toggling options. However, the floating screen-pointing area cannot be calibrated for tracing, and does not support tablet menu access.

NOTE The floating screen pointing area takes precedence over the fixed screen pointing area, which in turn takes precedence over the tablet.

You can switch between the fixed and floating screen-pointing areas by using F12 or by using a toggle button that you designate during configuration. Some Wintab drivers allow access to button commands only if the cursor is outside the screen-pointing area. If you notice that the buttons on your tablet pointer are not responding appropriately, move the pointer anywhere outside the fixed screen-pointing area and then press the button again.

Configuring the Tablet for a Menu Overlay

To configure your tablet to use an overlay, it is recommended that you select the default tablet menus, columns, and rows that are provided. The following illustration shows the tablet overlay supplied by AutoCAD LT. After the tablet menus have been configured, the procedure prompts you for the fixed and floating screen-pointing areas, along with the option to use a tablet pointer button to toggle between the fixed and floating screen-pointing areas.



The following procedure configures your tablet for the AutoCAD LT tablet overlay. Defaults that are displayed may depend upon the specific digitizer you are configuring.

To configure the tablet for use with the tablet overlay

1 At the Command prompt, enter **tablet**, and then enter **cfg** at the following prompt.

Option (ON/OFF/CAL/CFG):

2 At the following prompt, enter **4**.

Enter number of tablet menus desired (0-4) <0>:

3 When prompted, digitize the upper-left, lower-left, and lower-right corners of menu area 1.

In response to the following prompts, press ENTER.

Enter the number of columns for menu area 1, 1 to 4991 <25>:
Enter the number of rows for menu area 1, 1 to 1839 <9>:

4 When prompted, digitize the upper-left, lower-left, and lower-right corners of menu area 2.

In response to the following prompts, press ENTER.

Enter the number of columns for menu area 2, 1 to 2202 <11>:
Enter the number of rows for menu area 2, 1 to 1809 <9>:

5 When prompted, digitize the upper-left, lower-left, and lower-right corners of menu area 3.

In response to the following prompts, press ENTER.

Enter the number of columns for menu area 3, 1 to 539 <9>:
Enter the number of rows for menu area 3, 1 to 1806 <13>:

6 When prompted, digitize the upper-left, lower-left, and lower-right corners of menu area 4.

In response to the following prompts, press ENTER.

Enter the number of columns for menu area 4, 1 to 5004 <25>:
Enter the number of rows for menu area 4, 1 to 1407 <7>:

7 In response to the following prompt, enter **y**.

Do you want to respecify the Fixed Screen Pointing Area? <N>:

When prompted, digitize the lower-left and upper-right corners of the fixed screen-pointing area.

8 Respond to the following prompts to designate your floating screen-pointing area.

In response to the following prompts, enter **y**.

Do you want to specify the Floating Screen Pointing Area? <N>:

In response to the following prompt, enter **y** or **n**.

Do you want the Floating Screen Pointing Area to be the same size as the Fixed Screen Pointing Area? <Y>:

If you entered **n**, digitize the lower-left and upper-right corners of the floating screen-pointing area.

9 In response to the following prompt, enter **n** or press any button other than the pick button on the tablet pointer to designate it as the on/off button for the floating screen-pointing area.

The F12 key will toggle the Floating Screen Pointing Area ON and OFF. Would you like to specify a button to toggle the Floating Screen Area? <N>:

Calibrating the Tablet

If you want to digitize drawings, photographs, or other graphic material, you must first calibrate the tablet. If you want to digitize points in AutoCAD LT with Tablet mode turned on, you may want to configure the template for zero menus with a screen-pointing area of any reasonable size. The fixed screen-pointing area can be bigger or smaller than the default screen-pointing area on the AutoCAD LT template (tablet menu).

Using Tablet Menus in Tablet Mode

You can also calibrate the tablet to digitize drawings and keep the tablet menu areas configured; however, make sure that the configured area does not overlap the area you are tracing. Depending on the size of your digitizer and the size of the paper drawing, you may have to turn off your tablet menu area until you have finished digitizing your paper drawing.

If you want to switch from a tablet calibrated for tracing drawings to a tablet configured for a tablet overlay with a different size screen-pointing area, the tablet must be reconfigured.

To configure or reconfigure the tablet for digitizing drawings

1 Turn on your digitizer.

2 At the Command prompt, enter **tablet**, and then enter **cfg** at the following prompt.

3 At the following prompt, enter **0**.

Enter number of tablet menus desired (0-4) <0>:

4 At the following prompt, enter **y**.

Do you want to respecify the Fixed Screen Pointing Area? <N>

5 At the following prompts, digitize an area that is greater than or equal to the area you will use to digitize drawings.

Digitize lower left corner of Fixed Screen Pointing Area:
Digitize upper right corner of Fixed Screen Pointing Area:

6 At the following prompt, enter **y** if you want a floating screen pointing area.

Do you want to specify the Floating Screen Pointing Area? <N>

7 If you entered **y** in step 6, at the following prompt, enter **y** or **n**.

Do you want the Floating Screen Pointing Area to be the same size as the Fixed Screen Pointing Area? <Y>:

If you entered **n**, digitize a floating screen pointing area of the size you want. If you need to, you can repeat this step to resize the area.

8 In response to the following prompt, enter **y** or **n**.

The F12 key will toggle the Floating Screen Pointing Area ON and OFF. Would you like to specify a button to toggle the Floating Screen Area? <N>:

9 If you entered **y** at the prompt in step 8, at the following prompt, press any button other than the pick button.

Press any nonpick button that will become the toggle for the Floating Screen Area:

Calibration creates a mapping of the points on the tablet to the points on the paper drawing you want to trace. When you use the Cal option of TABLET, you specify points on the tablet with the tablet pointer and then give those points coordinate values. You can specify a minimum of two points that define a rectangular area.

Calibration requires specifying a minimum of two points, but specifying five points is recommended. Choose points that are not close to one another and for which you can enter precise *X*,*Y* coordinate values.

ONLINE HELP See "Calibrating the Tablet for Tracing."

When calibrating the tablet, it is recommended that you enter lower-left coordinates before upper-right coordinates. If you calibrate only two points and the second point is below and to the right of the first point (with positive coordinates), AutoCAD LT point mapping becomes unreliable.

To calibrate the tablet

1 Tape the paper drawing at the top of the active tablet surface area.
2 At the Command prompt, enter **tablet**. Then enter **cal**.
3 Using the first button on the tablet pointer, click a point on the tablet for which you know the *X,Y* coordinate.

   NOTE When clicking points, it is recommended you hold the puck styles in both hands and, looking down from the top, align the puck crosshairs as accurately as possible with the specified location in the drawing. An error or inaccuracy when picking the calibration points represents a residual error throughout your work.

   AutoCAD LT prompts you for the *X,Y* coordinate values.
4 Enter the *X,Y* coordinate value for the point you specified.
5 Repeat steps 3 and 4.
6 Press ENTER to end the command, or repeat steps 3 and 4 to calibrate more points.

After calibrating the tablet, set the transformation type to Orthogonal, Affine, or Projective.

- ***Orthogonal*** Specifies translation, uniform scaling, and rotation with two calibration points. Use Orthogonal for dimensionally accurate paper drawings and paper drawings in which the portion to be digitized is long and narrow, with most points confined to single lines.
- ***Affine*** Specifies arbitrary linear transformation in two dimensions consisting of translation, independent *X*- and *Y*-scaling, rotation, and skewing with three calibration points. Use Affine when horizontal dimensions in a paper drawing are stretched with respect to vertical dimensions and lines that are supposed to be parallel actually are parallel.
- ***Projective*** Specifies a transformation equivalent to a perspective projection of one plane in space onto another plane with four calibration points. A projective transformation provides a limited form of what cartographers call rubber sheeting, in which different portions of the tablet surface are stretched by varying amounts. Straight lines map into straight lines. Parallel lines do not necessarily stay parallel. Projective transformation corrects parallel lines that appear to converge.

If only two points are entered, AutoCAD LT automatically computes an orthogonal transformation. If it is successful, AutoCAD LT ends the command.

If three or more points are entered, AutoCAD LT computes the transformation in each of the three types to determine which best fits the calibration points. If more than four points are entered, computing the best-fitting projective transformation can take a long time. You can cancel the process by pressing ESC.

When the computations are complete, AutoCAD LT displays a table with the number of calibration points and a column for each transformation type. The following information is provided under each column: Outcome of fit, RMS error, Standard deviation, Largest residual/At point and Second-largest residual/At point.

If there have been no failures of projection transformation, AutoCAD LT prompts:

Select transformation type . . .
Orthogonal / Affine / Projective?<Repeat table>: Enter an option or press ENTER

Only transformation types for which the outcome was Success, Exact, or Canceled are included in this prompt. A projective transformation can be specified even if it was canceled. AutoCAD LT uses the result computed at the time you canceled.

After you have calibrated the tablet, the coordinates showing on your screen should be coincident with the coordinates in the paper drawing. You can now trace a drawing with the tablet and the tablet pointer. Enter an AutoCAD LT drawing command, and use the pointer to specify points on the material you are tracing. The points you specify become digitized to create a drawing.

Using Editing Commands in Tablet Mode

Any command that requires you to select objects with the tablet pointer still works in Tablet mode. For example, to erase an object, start the ERASE command and move the tablet pointer until the pickbox is over the object.

Restricting Mouse Input

AutoCAD LT can accept input from both a mouse and a digitizer. Use the following procedure to designate input from both pointing devices.

To set up AutoCAD LT to use both a mouse and a digitizer

1 From the Tools menu, choose Preferences.
2 In the Preferences dialog box, choose the System tab.
3 On the System tab, select Wintab Digitizer. Then select Digitizer Input from the Tablet or Mouse.

If you are digitizing a drawing into AutoCAD LT, you may not want accidental movement of the mouse to move the crosshairs.

When AutoCAD LT first starts up, the entire tablet is a fixed screen area. Inside a screen area, the tablet pointer acts like a mouse. If you select Digitizer Only input, you won't see the crosshairs moving. But, remember, the tablet pointer acts as a mouse inside a screen area.

To designate the tablet as the only input device for AutoCAD LT, use the following procedure.

To limit AutoCAD LT input to the digitizer

1. Calibrate the tablet as described in the procedure "To calibrate the tablet" on page 426.
2. From the Tools menu, choose Preferences.
3. In the Preferences dialog box choose the System tab.
4. On the System tab under Wintab Digitizer, select Digitizer Input from Calibrated Tablet Only. Then choose OK.

   NOTE You are now limited to using the tablet for input and only when Tablet mode is turned on. You cannot specify points in the drawing area with the mouse.

Testing Tablet Calibration

If your drawing is not correct after you calibrate your tablet, double-check the accuracy of your tablet. To test the scaling accuracy of your tablet, use the CAL option of TABLET.

To test the tablet calibration

1. At the Command prompt, enter **tablet**. Then enter **cal**.
2. Specify a point in the lower-left corner of the tablet.

   Write down the coordinate values AutoCAD LT displays on the status bar. These values represent the tablet's raw coordinate values before they are processed by AutoCAD LT.
3. In response to the prompt for point 1, enter the values shown in the coordinate display, and mark that point on the tablet.
4. Specify a point in the lower-right corner of the tablet.

   Move the cursor to the right of the first point, keeping the current *Y* value as close to the *Y* value of the first point as possible. Write down the current coordinate values AutoCAD LT displays on the status bar.

5 In response to the prompt for point 2, enter the current values shown in the coordinate display, and mark that point on the tablet.

   Move the cursor so that the *X* value is the same as the *X* value of the first point, and the *Y* value is the same as the *Y* value of the second point.

6 Enter the *X* and *Y* values of the coordinate and mark those points on the tablet.

7 Press ENTER to calculate the transformation.

The calculations should show an RMS (root-mean-square) error very close to 0. RMS measures how close AutoCAD LT has come to finding a perfect fit. The goal is to find the smallest RMS error. The affine transformation should be exact.

Tablets vary in accuracy. Accuracy specifications are listed in the manual provided with each tablet model. AutoCAD LT can compensate for a paper-to-drawing scaling inaccuracy by using the three-point affine transformation during tablet calibration. If the inaccuracy in the orthogonal transformation is greater than the inaccuracy of manually positioning the tablet's cursor, use the affine transformation.

ONLINE HELP See "TABLET" in Listing of Commands.

Reinitializing the Tablet

For AutoCAD LT to run properly, all the input and output ports, the *aclt.pgp* file, and peripheral devices attached to your computer—pointing device, video display, and plotter—must be properly initialized. If your tablet hardware settings change and need to be reinitialized for any reason, use REINIT to reinitialize the tablet parameters with AutoCAD LT.

To reinitialize the tablet

1 At the Command prompt, enter **reinit**.

2 In the Reinitialization dialog box under I/O Port Initialization, select Digitizer.

3 Under Device and File Initialization, select Digitizer.

Some Wintab drivers require a computer shutdown and start-up sequence in order to reinitialize properly. If reinitializing the tablet does not correct your problem, shut down your computer and restart it. Then verify that the Wintab driver is functional in Windows before running AutoCAD LT.

Glossary

Commands associated with definitions are shown in parentheses at the end of the definition.

absolute coordinates Coordinate values measured from a coordinate system's origin point. *See also* **origin**, **relative coordinates**, **user coordinate system** (UCS), **world coordinates**, *and* **World Coordinate System** (WCS).

ADI For *Autodesk Device Interface*. An interface specification for developing device drivers that are required for peripherals to work with AutoCAD LT and other Autodesk products.

affine transformation A tablet calibration method that provides an arbitrary linear transformation in two-dimensional space. Affine transformation requires three calibration points to allow a tablet transformation that combines translation, independent *X* and *Y* scaling, rotation, and some skewing. Use affine transformation if a drawing has been stretched differently in the horizontal or vertical direction. (TABLET)

alias A shortcut for an AutoCAD LT command. For example, *CP* is an alias for COPY, and *Z* is an alias for ZOOM. You define aliases in the *aclt.pgp* file.

aligned dimension A dimension that measures the distance between two points at any angle. The dimension line is parallel to the line connecting the dimension's definition points. (DIMALIGNED)

angular dimension A dimension that measures angles or arc segments and consists of text, extension lines, and dimension line arcs. (DIMANGULAR)

angular unit The unit of measurement for an angle. Angular units can be measured in decimal degrees, degrees/minutes/seconds, grads, and radians. (DDUNITS, UNITS)

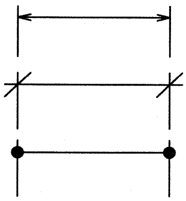
annotations Text, dimensions, tolerances, symbols, or notes.

anonymous block An unnamed block which AutoCAD LT creates to support hatch patterns and associative dimensions.

ANSI For *American National Standards Institute*. Coordinator of voluntary standards development for both private and public sectors in the United States. Standards include programming languages, Electronic Data Interchange (EDI), telecommunications, and the physical properties of diskettes, cartridges, and magnetic tapes.

approximation points Point locations that a B-spline must pass near, within a fit tolerance. *See also* **fit points** *and* **interpolation points**.

array Multiple copies of selected AutoCAD LT objects in a rectangular or polar (radial) pattern. (ARRAY)



sample arrowheads

arrowhead A terminator, such as an arrowhead, slash, or dot, at the end of a dimension line showing where a dimension begins and ends.

ASCII For *American Standard Code for Information Interchange*. A common numeric code used in computer data communications. The code assigns meaning to 128 numbers, using seven bits per character with the eighth bit used for parity checking. Nonstandard versions of ASCII assign meaning to 255 numbers.

aspect ratio Ratio of display width to height.

associative dimension A dimension that adapts as the associated geometry is modified.

associative hatching Hatching that conforms to its bounding objects such that modifying the bounding objects automatically adjusts the hatch. (BHATCH)

attribute definition An AutoCAD LT object that is included in a block definition to store alphanumeric data. Attribute values can be predefined or specified when the block is inserted. Attribute data can be extracted from a drawing and inserted into external files. (DDATTDEF, ATTDEF)

attribute extraction file An ASCII text file to which extracted attribute data is written. The contents and format are determined by the attribute extraction template file.

attribute extraction template file An ASCII text file that determines which attributes are extracted and how they are formatted when written to an attribute extraction file.

attribute prompt The text string displayed when you insert a block with an attribute whose value is undefined.

attribute tag A text string associated with an attribute that identifies a particular attribute during extraction from the drawing database.

attribute value The alphanumeric information associated with an **attribute tag**.

AutoCAD LT library search path The order in which AutoCAD LT looks for a support file: current directory, drawing directory, directory specified in the support path, and directory containing the AutoCAD LT executable file, *aclt.exe*.

Autodesk Device Interface *See* **ADI**.

baseline An imaginary line on which text characters appear to rest. Individual characters can have descenders that drop below the baseline.

baseline dimension Multiple dimensions measured from the same baseline. Also called *parallel dimensions*. (DIMBASELINE)

base point 1. In the context of editing grips, the grip that changes to a solid color when selected to specify the focus of the subsequent editing operation. 2. A point for relative distance and angle when copying, moving, and rotating objects. 3. The insertion base point of the current drawing. (BASE) 4. The insertion base point for a block definition. (BLOCK)

Bezier curve A polynomial curve defined by a set of control points, representing an equation of an order one less than the number of points being considered. A Bezier curve is a special case of a B-spline curve.

bitmap The digital representation of an image having bits referenced to pixels. In color graphics, a different value represents each red, green, and blue component of a pixel.

blip marks Temporary screen markers displayed in the AutoCAD LT graphics area when you specify a point or select objects. (BLIPMODE)

block A generic term for one or more AutoCAD LT objects that are combined to create a single object. Commonly used for either block definition or block reference. *See* **block definition** *and* **block reference**. (BLOCK)

block definition The name, base point, and set of objects that are combined and stored in the symbol table of a drawing. (BLOCK) *See* **block**. *See also* **block reference**.

block reference A compound object that is inserted in a drawing and displays the data stored in a block definition. Also called *instance*. *See* **block**. *See also* **block definition**. (INSERT)

block table The nongraphic data area of a drawing file that stores block definitions. *See also* **symbol table**.

B-spline curve A blended piecewise polynomial curve passing near a given set of control points. (SPLINE)

button menu The menu for a pointing device with multiple buttons. Each button on the pointing device (except the pick button) can be defined in the AutoCAD LT menu file *aclt.mnu* in the BUTTONS*n* and AUX*n* sections.

BYBLOCK A special object property used to specify that the object inherits the color or linetype of any block containing it. *See also* **BYLAYER**.

BYLAYER A special object property used to specify that the object inherits the color or linetype associated with its layer. *See also* **BYBLOCK**.

circular external reference An externally referenced drawing (xref) that references itself directly or indirectly. AutoCAD LT ignores the xref that creates the circular condition.

clipping planes The boundaries that define or clip the field of view. (DVIEW)

CMYK For *cyan, magenta, yellow, and key color.* A system of defining colors by specifying the percentages of cyan, magenta, yellow, and the key color, which is typically black.

color map A table defining the intensity of red, green, and blue (RGB) for each displayed color.

command line A text area reserved for keyboard input, prompts, and messages.

construction plane A plane on which planar geometry is constructed. The *XY* plane of the current user coordinate system (UCS) represents the construction plane. *See also* **elevation** *and* **user coordinate system**.

continued dimension A type of linear dimension that uses the second extension line origin of a selected dimension as its first extension line origin, breaking one long dimension into shorter segments that add up to the total measurement. Also called *chain dimension*. (DIMCONTINUE)

control frame A series of point locations used as a mechanism to control the shape of a B-spline. These points are connected by a series of line segments for visual clarity and to distinguish the control frame from fit points. The system variable SPLFRAME must be turned on to display control frames.

control point *See* **control frame**.

crosshairs

crosshairs A type of cursor consisting of two lines that intersect. Also called *graphics cursor.*

crossing polygon A multisided area specified to select objects fully or partially within its borders. *See also* **crossing rectangle** *and* **enclosing polygon**.

crossing rectangle A rectangular selection area drawn from right to left to select objects fully or partly within its borders. *See also* **crossing polygon** *and* **enclosing rectangle**.

cursor A pointer on a video display screen that can be moved around to place textual or graphical information. Also called *graphics cursor. See also* **crosshairs.**

cursor menu The menu that is displayed in the graphics area at the cursor location when you hold down SHIFT and right-click the pointing device. The cursor menu is defined in the POP0 section of *aclt.mnu.*

default A predefined value for a program input or parameter. Default values and options for AutoCAD LT commands are denoted by angle braces <>.

default drawing *See* **initial environment**.

definition points Points created as part of an associative dimension. AutoCAD LT uses the points to modify the appearance and value of an associative dimension when the associated object is modified. Also called *defpoints* and stored on the special layer DEFPOINTS.

dependent symbols Symbol table definitions that are brought into a drawing by an external reference. *See also* **symbol table**.

DIESEL For *Direct Interpretively Evaluated String Expression Language*. A macro language for altering the AutoCAD LT status line with the MODEMACRO system variable and for customizing menu items.

dimension line arc An arc (usually with arrows at each end) spanning the angle formed by the extension lines of an angle being measured. The dimension text near this arc sometimes divides it into two arcs. *See also* **angular dimension**.

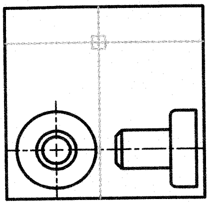
dimension style A named group of dimension settings that determines the appearance of the dimension and simplifies setting dimension system variables. (DDIM)

dimension text The measurement value of dimensioned objects.

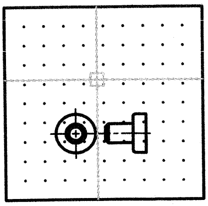
dimensioning system variables A set of numeric values, text strings, and settings that control AutoCAD LT dimensioning features. (DDIM)

direct distance entry A method to specify a second point by first moving the cursor to indicate direction and then entering a distance.

dithering Combining color dots to give the impression of displaying more colors than are actually available.



drawing extents



drawing limits

drawing extents The smallest rectangle that contains all objects in a drawing, positioned on the screen to display the largest possible view of all objects. (ZOOM)

drawing limits The user-defined rectangular boundary of the drawing area covered by dots when the grid is turned on. Also called *grid limits*. (LIMITS)

DWF For *drawing Web format*. A highly compressed file format that is created from a DWG file. DWF files are easy to publish and view on the Web. (EXPORT)

DWG Standard file format for saving vector graphics from within AutoCAD LT.

DXF For *drawing interchange format*. An ASCII or binary file format of an AutoCAD LT drawing file for exporting AutoCAD LT drawings to other applications or for importing drawings from other applications. (EXPORT, IMPORT)

elevation The default *Z* value above or below the *XY* plane of the current user coordinate system (UCS). Elevation is used for entering coordinates and digitizing locations. (ELEVATION)

embed To use object linking and embedding (OLE) information from a source document in a destination document. An embedded object is a copy of the information from a source document that is placed in the destination document and has no link to the source document. *See also* **link**. (PASTESPEC)

enclosing polygon A multisided polygon area specified to select objects contained completely within its borders. *See also* **crossing polygon** *and* **enclosing rectangle**.

enclosing rectangle A rectangular selection area drawn from left to right in the AutoCAD LT graphics area to select objects contained completely within its borders. *See also* **crossing polygon**, **crossing rectangle**, *and* **enclosing polygon.**

explode To disassemble a complex object, such as a block or polyline, into simpler objects. In the case of a block, the block definition is unchanged. The block reference is replaced by the components of the block. *See also* **block**, **block definition**, *and* **block reference**. (EXPLODE)

extents *See* **drawing extents**.

external reference (xref) A drawing file linked (or attached) to another drawing. (XREF)

fence A multisegmented line specified to select objects it passes through.

fill A solid color covering an area bounded by lines or curves. (FILL)

filters *See* **coordinate filters.**

fit points Locations that a B-spline must pass through exactly or within a fit tolerance. *See also* **interpolation points** *and* **approximation points**.

fit tolerance The setting for the maximum distance that a B-spline can pass for each of the fit points that define it.

floating viewports Rectangular objects that display views in paper space. *See also* **paper space**. (MVIEW)

font A character set, comprising letters, numbers, punctuation marks, and symbols, of a distinctive proportion and design.

freeze A setting that suppresses the display of objects on selected layers. Objects on frozen layers are not displayed, regenerated, or plotted. Freezing layers shortens regenerating time. *See also* **thaw**. (LAYER, DDLMODES)

graphics area The area of the AutoCAD LT screen for creating and editing a drawing.

graphics cursor *See* **crosshairs** *and* **cursor**.

graphics screen *See* **graphics window**.

graphics window The graphics area, its surrounding menus, and the command line.

grid An area on the graphics display covered with regularly spaced dots to aid drawing. The spacing between grid dots is adjustable. Grid dots are not plotted. *See also* **drawing limits**. (GRID)

grip commands The editing capabilities activated when grips are displayed on an object: stretching, moving, rotating, scaling, and mirroring.

grips Small squares that appear on objects you select. After selecting the grip, you edit the object by dragging it with the pointing device in combination with the grip commands.

handle A unique alphanumeric tag for an object in the AutoCAD LT database.

home page The main navigating screen for a Web site.

IGES For *initial graphics exchange specification*. An ANSI-standard format for digital representation and exchange of information between CAD/CAM systems.

initial environment The variables and settings for new drawings as defined by the default template drawing, such as *aclt.dwt* or *acltiso.dwt*. *See also* **template drawing**.

instance *See* **block reference**.

Intellimouse The Intellimouse is a two-button mouse with a small wheel between the buttons. The Intellimouse lets you zoom and pan your drawing using the wheel, without using any commands from AutoCAD LT. Clicking the wheel is equivalent to clicking the middle button on a three-button mouse.

interpolation points Defining points that a B-spline passes through. *See also* **approximation points** *and* **fit points**.

island An enclosed area within a hatched area.

ISO For *International Standards Organization*. The organization that sets international standards in all fields except electrical and electronics. Headquarters are in Geneva, Switzerland.

isometric snap style An AutoCAD LT drafting option that aligns the cursor with two of three isometric axes and displays grid points, making isometric drawings easier to create. (DDRMODES)

layer A logical grouping of data similar to transparent acetate overlays on a drawing. You can view and plot layers individually or in combination. (DDLMODES, LAYER)

limits *See* **drawing limits**.

line font *See* **linetype**.

linetype How a line or type of curve is displayed. For example, a continuous line has a different linetype than a dashed line. Also called *line font*. (LINETYPE)

link To use object linking and embedding (OLE) to reference data in another file. When data is linked, any changes to it in the source document are automatically updated in any destination document. *See also* **embed**. (COPYLINK)

mirror To create a new version of an existing object by reflecting it symmetrically with respect to a prescribed line or plane. (MIRROR)

model A two- or three-dimensional representation of an object.

model space One of the two primary spaces in which AutoCAD LT objects reside. Typically, a geometric model is placed in a three-dimensional coordinate space called model space. A final layout of specific views and annotations of this model is placed in paper space. *See also* **paper space**. (MSPACE)

named view A view saved for restoration later. (VIEW)

node An object snap specification to locate points, dimension definition points, and dimension text origins.

noun-verb selection Selecting an object first and then performing an operation on it rather than entering a command first and then selecting the object.

NURBS For *nonuniform rational B-spline curve*. A B-spline curve or surface defined by a series of weighted control points and one or more knot vectors. *See* **B-spline curve**.

object One or more AutoCAD LT graphical elements, such as text, dimensions, lines, circles, or polylines, treated as a single element for creation, manipulation, and modification. Also called *entity*.

object snaps Methods for selecting commonly needed points on an object while you create or edit an AutoCAD LT drawing. *See also* **running object snap** *and* **object snap override**. (DDOSNAP, OSNAP)

object snap override Turning off or changing a running object snap setting for input of a single point. *See also* **object snap** *and* **running object snap**.

OLE For *object linking and embedding*. An information-sharing method in which data from a source document can be linked to or embedded in a destination document. Selecting the data in the destination document opens the source application so that the data can be edited. *See also* **embed** *and* **link**.

origin The point where coordinate axes intersect. For example, the origin of a Cartesian coordinate system is where the *X*, *Y*, and *Z* axes meet at 0,0,0.

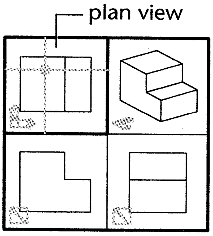
orthogonal calibration A tablet calibration method that specifies translation, uniform scaling, and rotation with two calibration points. Use Orthogonal for dimensionally accurate paper drawings and paper drawings in which the portion to be digitized is long and narrow, with most points confined to single lines. (TABLET)

Ortho An AutoCAD LT setting that limits pointing device input to horizontal or vertical (relative to the current snap angle and the user coordinate system). *See also* **snap angle** *and* **user coordinate system**. (ORTHO)

pan To shift the view of a drawing without changing magnification. *See also* **zoom**. (PAN)

paper space One of two primary spaces in which AutoCAD LT objects reside. Paper space is used for creating a finished layout for printing or plotting, as opposed to doing drafting or design work. Model space is used for creating the drawing. *See also* **model space, viewport.** (PSPACE)

pick button The button on a pointing device that is used to select objects or specify points on the screen. For example, on a two-button mouse, it is the left button.



plan view A view orientation from a point on the positive Z axis toward the origin (0,0,0). (PLAN)

pline *See* **polyline.**

point 1. A location in three-dimensional space specified by *X, Y,* and *Z* coordinate values. 2. An AutoCAD LT object consisting of a single coordinate location. (POINT)

pointing device A pointing device, such as a mouse or a digitizing puck, can be used to interact with AutoCAD LT's interface and create and edit drawing objects in the graphics area. A pointing device usually has several buttons, some of which may be customized to perform commands you specify.

point filters Functions that extract individual *X, Y,* and *Z* coordinate values from different points to create a new, composite point. Also called *X,Y,Z coordinate filters.*

polar array Objects copied around a specified center point a specified number of times. (ARRAY)

PolarSnap An AutoCAD LT control to specify distance and angle for pointing device input. *See also* **Ortho.** (POLAR)

polygon window A multisided area specified to select objects in groups. *See also* **CPolygon, crossing window,** *and* **WPolygon.**

polyline An AutoCAD LT object composed of one or more connected line segments or circular arcs treated as a single object. Also called *pline.* (PLINE, PEDIT)

projective calibration A tablet calibration method that specifies a transformation equivalent to a perspective projection of one plane in space onto another plane with four calibration points. A projective transformation provides a limited form of what cartographers call rubber sheeting, in which different portions of the tablet surface are stretched by varying amounts. Straight lines map into straight lines. Parallel lines do not necessarily stay parallel. (TABLET)

Projective transformation corrects parallel lines that appear to converge.

prompt A message on the command line that asks for information or requests action such as specifying a point.

redraw To quickly refresh or clean up the current viewport without updating the drawing's database. *See also* **regenerate.** (REDRAW)

regenerate To update a drawing's screen display by recomputing the screen coordinates from the database. *See also* **redraw.** (REGEN)

relative coordinates Coordinates specified in relation to previous coordinates.

RGB For *red, green, and blue.* A system of defining colors by specifying percentages of red, green, and blue.

rubber-band line A line that stretches dynamically on the screen with the movement of the cursor. One endpoint of the line is attached to a point in your drawing, and the other is attached to the moving cursor.

running object snap Setting an object snap so it continues for subsequent selections. (DDOSNAP, OSNAP**)** *See also* **object snap** *and* **object snap override**.

script file A set of AutoCAD LT commands executed sequentially with a single SCRIPT command. Script files are created outside AutoCAD LT using a text editor, saved in text format, and stored in an external file with the extension *.scr.*

selection set One or more AutoCAD LT objects specified for processing as a unit. (DDSELECT)

shortcut menu The menu that is displayed in the command window, text window, or graphics area at the cursor location when you right-click the pointing device. The shortcut menu commands vary depending upon the screen context and whether an object is selected. The shortcut menu provides easy access to editing commands when editing with grips.

snap angle The angle that the snap grid is rotated. (SNAP)

snap grid The invisible grid that locks the graphics cursor into alignment with the grid points according to the spacing set by SNAP. Snap grid does not necessarily correspond to the visible grid, which is controlled separately by GRID. (SNAP)

Snap A drawing aid for locking a pointing device into alignment with an invisible rectangular grid. When Snap is on, the screen crosshairs and all input coordinates are snapped to the nearest point on the grid. The snap resolution defines the spacing of this grid. (SNAP)

snap resolution The spacing between points of the snap grid. (SNAP)

symbol table A nongraphic AutoCAD LT object definition that is stored in the drawing, also known as a *named object*. Symbols can include definitions of blocks, dimensioning styles, layers, linetypes, and text styles.

system variable A name that AutoCAD LT recognizes as a setting, size, or limit. Read-only system variables, such as DWGNAME, cannot be modified directly by the user.

template drawing A drawing file with preestablished settings for new drawings such as *aclt.dwt* and *acltiso.dwt*; however, any drawing can be saved and used as a template. *See also* **initial environment**.

temporary files Data files created during an AutoCAD LT session. AutoCAD LT deletes the files by the time you end the session. If the session ends abnormally, such as during a power outage, temporary files might be left on the disk.

text style A named, saved collection of settings that determines the appearance of text characters—for example, stretched, compressed, oblique, mirrored, or set in a vertical column. (STYLE)

thaw A setting that displays previously frozen layers. *See also* **freeze**. (LAYER)

thickness The distance certain objects are extruded to give them a 3D appearance. (CHPROP, DDCHPROP, ELEV, THICKNESS)

tiled viewports A type of display that splits the AutoCAD LT graphics area into one or more adjacent rectangular viewing areas. *See also* **floating viewports**, **TILEMODE**, *and* **viewport**. (VPORTS)

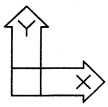
TILEMODE A system variable that controls whether viewports can be created as movable, resizable objects (floating), or as nonoverlapping display elements that appear side-by-side (tiled). *See also* **viewport**.

toolbar Part of the AutoCAD LT interface containing icons that represent commands.

tracking A way to locate a point relative to other points on the drawing. (TRACKING)

transparent command A command started while another is in progress. Precede transparent commands with an apostrophe.

UCS *See* **user coordinate system**.



UCS icon

UCS icon An icon that indicates the orientation of the UCS axes. (UCSICON)

user coordinate system (UCS) A user-defined coordinate system that defines the origin and orientation of the *X*, *Y*, and *Z* axes in 3D space. The UCS determines the default placement of geometry in a drawing. *See also* **World Coordinate System** (UCS).

vector A mathematical object with precise direction and length but without specific location.

vertex A location where edges or polyline segments meet.

view A graphical representation of a model from a specific location (viewpoint) in space. (VPOINT, DVIEW, VIEW)

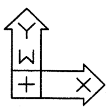
viewpoint The location in 3D model space from which you are viewing a model. (DVIEW, VPOINT)

viewport A bounded area that displays some portion of the model space of a drawing. The TILEMODE system variable determines the type of viewport created. 1. When TILEMODE is off (0), viewports are objects that can be moved and resized. (MVIEW) 2. When TILEMODE is on (1), viewports are noneditable, nonoverlapping screen displays. *See also* **floating viewport** *and* **TILEMODE.** (VPORTS)

viewport configuration A named collection of tiled viewports that can be saved and restored. (VPORTS)

WCS *See* **World Coordinate System**.

working drawing A drawing for manufacturing or building purposes, displayed in the graphics area.



WCS icon

world coordinates Coordinates expressed in relation to the World Coordinate System.

World Coordinate System (WCS) A coordinate system used as the basis for defining all objects and other coordinate systems.

xref *See* **external reference**.

***X,Y,Z* point filters** *See* **coordinate filters.**

zoom To reduce or increase the apparent magnification of the graphics display. (ZOOM)

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Hard Disk Space

Graphics Card(s)

Digitizer/Mouse

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Plotter	☐ Serial ☐ Parallel
Printer	☐ Serial ☐ Parallel

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