Chameleon™ ATX for All Windows® Platforms

Standards Based IntraNet Solutions

User's Guide

Version 6.0

NetManage, Inc.
10725 North De Anza Blvd.
Cupertino, CA 95014, USA
Fax: (408) 973-8272
Internet: sales@netmanage.com, intl_sales@netmanage.com
support@netmanage.com, intl_support@netmanage.com

West Coast	East Coast	International
Sales: (408) 973-7171	Sales: (603) 888-2800	Phone: +972-4-8550234
Support: (408) 973-8181	Support: (603) 888-3500	Fax: +972-4-8550122

Chameleon™ ATX for All Windows® Platforms User's Guide, Version 6.0

Part number 3000-60-0696

June 1996

Notice

Disclaimer

NetManage makes no representation or warranties other than the standard warranty with regard to the contents of this manual or the suitability of this software product for any particular purpose. Therefore, NetManage assumes no liability for damages incurred directly or indirectly from errors, omissions or discrepancies between the software and this manual.

Important: You should carefully read the license terms and conditions included in this package before you open or use the product. If you do not agree with the terms and conditions, you should promptly return the product unopened.

Disclosure: Permission to use, copy modify, and distribute portions of this program are granted by M.I.T. and Carnegie Mellon. Such rights are granted provided that the names M.I.T. and Carnegie Mellon are not used in any advertising or publicity pertaining to the distribution of this program or portions thereof. M.I.T. and Carnegie Mellon make no representations about the suitability of this software, or portions thereof for any purpose. It is provided "AS IS" without expressed or implied warranty.

Trademarks

NetManage and the NetManage Logo are registered trademarks of NetManage, Inc. Automatic Internet, Chameleon, and NEWT are trademarks of NetManage, Inc.

Adobe and Acrobat are registered trademarks of Adobe Systems Incorporated.

Contents

Chapter 1. Overview

Chameleon ATX for Windows (including the Windows 3.1 and Windows 95 operating systems) combines an NFS client/server and a powerful set of network applications, all built on top of TCP/IP for Microsoft Windows. Network drives appear and act as local drives through the use of the NFS (Network File System) protocol.

Through the exclusive use of TCP/IP for all network communication, the user is guaranteed transparent access to a wide variety of systems that support these same protocols. Your Windows-based PCs can seamlessly integrate into existing network environments, or expand small networks into large ones, without the painful changes commonly associated with these transitions.

Most applications have both client and server capabilities. This permits building of peer-to-peer networks of PCs without the use of a centralized workstation or mainframe. This also eliminates the need for a proprietary protocol for PC-to-PC communication; it also means that TCP/IP no longer needs to be used only to provide host access.

NetManage products are not adaptations of previous DOS products. Instead they are unique applications that take full advantage of Windows-based systems.

Product Features

Chameleon ATX for Windows incorporates a number of features not normally found in other networking products such as:

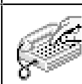
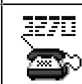
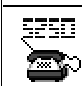
- NFS client and server
- Mounting and unmounting of network drives from within the Windows File Manager
- Multiple windows running multiple TCP/IP sessions
- Applications that include both client and server functions
- Implementation as Windows DLL (not a TSR)
- Use of a Windows point-and-click user interface
- Context-sensitive online help
- TCP and UDP protocol support
- Integrated serial dialing and gateway access
- Policy Management Architecture Enforcer lets you manage the amount of bandwidth available to specific application traffic. For more information, refer to the NetManage Configuration Guide.

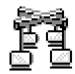
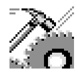
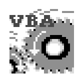
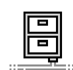
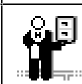
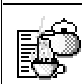
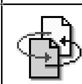
- Policy Management Architecture, a feature provided by some major router manufacturers, can guarantee bandwidth to specified IP traffic. For more information, refer to the NetManage Configuration Guide.
- NetWare IP, a feature offered by Novell to allow Windows clients to access Novell servers using the Internet Protocol (IP). For more information, refer to the NetManage Configuration Guide.
- Go IntraNet! feature, which lets you start many NetManage applications with destinations from a single location.
- Tech Support Wizard, provides the means to automatically report problems and ask questions of the NetManage Technical Support Department. More information on this feature and other support-related feature is provided in a later chapter of this manual.
- Optional TCP/IP development kits

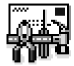
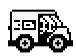
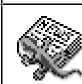
Chameleon ATX Applications

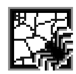
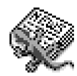
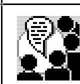
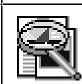
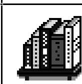
The Chameleon ATX product includes a wide variety of network applications. Application icons, listed in the following table, are arranged by type.

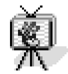
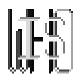
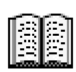
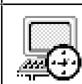
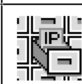
Note: Not all the applications listed are available for the three operating systems. The differences among applications for each operating system are noted in each chapter for your information.

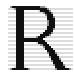
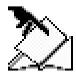
Tool	Icon	Description
NetManage TCP/IP		
Custom		Lets you set up and configure your TCP/IP client. You can create LAN or dial-up interface profiles.
Dialer		Lets you establish dial-up connections to the Internet and modify interfaces
Host Access		
Telnet		Lets you connect your PC with a UNIX or VMS host and many types of BBSes using Telnet or asynchronous connections
TN3270		Emulates 3270 terminals for connecting to IBM hosts
TN5250		Emulates 5250 terminals for connecting to IBM hosts

Tool	Icon	Description
PR3270 Printer		Emulates an IBM 3287 system printer and enables you to print IBM S/370 and S/390 Sessions to various printers. The 3270 Printer application makes the printer appear to the host as if it were an IBM system printer.
Session Manager		Lets you manage multiple terminal emulation sessions from within a single program
VBA Script Editor Visual Script Player VBA Script Player	  	Lets you dynamically reconfigure the user interface, perform random access file operations, automate tasks, modify current functionality, and handle incoming events with the Telnet and ASCII terminal, TN3270, TN5250, Z-Mail® Pro, and FTP applications
File and Print		
FTP		Lets you connect to FTP servers and transfer files to and from wide variety of remote computers
FTP Server		Lets other Internet users directly access files on your PC
TFTP		Lets you transfer one file at a time to and from remote host computers that do not support FTP
NFS Utilities		Converts text files and displays file name mapping between UNIX and Windows
LPR LPD	 	Lets files be printed remotely over the network **Note:** Icon displays only in Windows 95 and Windows 3.1.
E-Mail and Messaging		
Z-Mail® Pro		Lets you send and receive electronic mail

Tool	Icon	Description
Z-Mail® Pro Administration		Provides additional functions for managing Z-Mail® Pro
Mail Alias Viewer		Lets you expand a mail alias for a particular server, which allows you to see the individual e-mail addresses that are included in the aliases
Mailbox Converter		Converts messages stored in various mail systems to the Z-Mail® Pro message base
Outbox		Delivers sent messages as soon as the Outbox receives them. Includes all messages sent using Z-Mail® Pro, Tag-It!, Phone Tag, Forum, and NEWTCal
Directory Server		Lets you keep a company-wide address book, or any other address book, available for other users
Phone Tag		Provides a handy method of recording and sending phone memos
Rapid Search		Automatically tracks and finds all text indexed by the Indexing Agent
Indexing Agent		Provides full text indexing for mail messages and Web pages
Tag-It!		Provides a method of sending informal messages to other people
Group Collaboration		
NEWTCal		Lets you schedule appointments in an online calendar
Forum (formerly NEWTNews)		Lets you subscribe to news groups on the Usenet, and to read and post articles
InPerson™		Lets you have audio and video online working sessions with people in other locations

Tool	Icon	Description
Internet Access		
WebSurfer		Lets you browse hypertext documents published on World Wide Web (WWW) servers
Personal Web Server		Lets you publish multimedia hypertext documents on the WWW
WebSpider		Lets you create and design documents over the WWW
NEWTNews (now called Forum)		Lets you subscribe to news groups on the Usenet, and to read and post articles. Also considered part of Group Collaboration.
Archie™		Lets you view and retrieve public-domain files from anonymous FTP hosts
Talk		Lets you participate in real-time conversations with remote users Acts as both a client and server, and responds to remote talk requests with a pop-up dialog box and with or without sound
Internet Relay Chat (IRC)		Lets Internet users chat with one another in real-time
NEWTView		Lets you display, copy, and print bit-mapped graphics files produced by various applications
Gopher		Lets you browse thousands of public Gopher information servers
NEWTSound		Lets you play a sound file

Tool	Icon	Description
TrueSpeech		Third-party embedded application that lets you bring real-time, streaming audio playback over the Internet using WebSurfer
RealAudio		Third-party embedded application that lets you bring live and on-demand audio to the desktop over the Internet using WebSurfer
Internet Viewer		Lets you view several different features of the Internet from a single, integrated application, including a News client, a Weather viewer, a Stock quote service, and a Web browser
WhoIs		Provides information about other network users including phone numbers and e-mail addresses
Desktop Management		
NEWTToolbar		Maintains a list of applications that you can start without having to switch to another window
Virtual Desktop Manager		Lets you use a large virtual screen
Ping		Checks your connection to a specified computer
Finger		Displays information about users on a specified computer
Bind		Provides name server functions as a simplified Domain Name Server (DNS)
PC Net-Time		Lets you synchronize the time between your PC and any time server
Name Resolver		Lets you easily retrieve the IP address of a known host or the hostname of an IP address

Tool	Icon	Description
R-Commands		Lets you enter and execute the rcp (remote copy) and rsh (remote shell) UNIX commands in a Windows environment
File & Disk Replicator		Creates an up-to-date local copy of files in a specified directory of a remote host
Modem Server		Lets you use one of your PCs as a modem server for other PCs on the same LAN
NISLookup		Lets you display information retrieved from NIS including map entries, domain and server names
Look@Me		Third-party embedded application and plug-in that lets you view another Look@Me user's screen anywhere in real-time

Other Products

NetManage is committed to expanding network capabilities in line with existing products, such as ChameleonNFS for Windows NT. We will maintain our high standards for ease of use and integration with Windows.

Our current set of products includes:

- Chameleon: TCP/IP applications package for Windows
- ChameleonNFS for Windows: TCP/IP applications package for Windows, including NFS client and server for Windows.
- Chameleon for Windows 95: TCP/IP applications package for Windows 95
- ChameleonNFS for Windows 95: TCP/IP applications package for Windows 95, including NFS client and server
- Chameleon/X: Chameleon bundled with Xoftware
- ChameleonNFS/X: ChameleonNFS bundled with Xoftware
- Internet Chameleon: Internet dial-up access
- NEWTWatch: SNMP-based desktop management
- NEWT-SDK: Development kit for TCP/IP DLLs and ONC RPC/XDR
- NEWT IntraNet ActiveX: Development kit for ActiveX Controls
- JetMail Client and Server: Mail services applications for small businesses

- NetManage Intranet Server: Suite of server applications for Windows NT systems
- ECCO Lite™, ECCO Professional™: Personal information management and workgroup communications applications
- NetManage IntraNet HostLink™ (formerly Swift™): Full-featured terminal emulation solution for Windows PC users who need access to AS/400™ and IBM mainframe systems
- Xoftware for Windows 3.1, Windows NT, and Windows 95: X Windows System display server for Windows products
- Chameleon/X for Mac™ OS: Chameleon bundled with Xoftware for the Mac OS
- ChameleonNFS/X for Mac™ OS: ChameleonNFS bundled with Xoftware for the Mac OS
- WinSock SDK for Mac™ OS: Developer kit using WinSock API for the Mac OS
- Chameleon for Mac™ OS with PacerTerm: TCP/IP package applications package for the Mac OS

We also have native-language versions of some of our products in:

- Dutch
- French
- German
- Japanese
- Spanish

Using This Manual

There are now two manuals for the Chameleon ATX for Windows product:

- The first manual, NetManage Configuration Guide, contains the configuration and setup information for NEWT for Windows 3.1 and Windows 95. You do not need to read the first manual if you are using the Microsoft TCP/IP stack instead of NEWT.

The following table indicates how to use the NetManage Configuration Guide for the different environments in which you can run NetManage products.

Environment	Guide Reference
Windows NT	Chapter 2. Then refer to the User’s Guide for your NetManage product.
Windows 95 and using NEWT	Chapters 1 through 9. Then refer to the User’s Guide for your NetManage product.
Windows 95 and using the Microsoft TCP/IP stack	Chapter 2. Then refer to the User’s Guide for your NetManage product.
Windows 3.1 and using NEWT	Chapters 1 through 9 and Appendixes A and B. Then refer to the User’s Guide for your NetManage product.
Windows 3.1 and using the Microsoft TCP/IP stack	Chapter 2. Then refer to the User’s Guide for your NetManage product.

- The second manual, Chameleon ATX for Windows 95 and Windows 3.1 User's Guide, contains information about each application included for all three Windows operating systems.

To use these documents, you are assumed to be familiar with Microsoft Windows 95, or Windows 3.1, DOS, and basic TCP/IP terminology. You will find troubleshooting sections at the end of most chapters, which provide a simple way to quickly diagnose any problems you may encounter. The Glossary at the end of the manual also defines basic terminology.

Finding Help

Most applications include online documentation in the form of standard Windows help. This is available at all times from within the application, and includes a glossary of terms, limiting the need to refer to this documentation. The help files for all applications are included within the \netmanag (for Windows 3.1) directory or \netmanag.95 (for Windows 95) directory, and are named with the file extension *.hlp. The Additional Online Information sections in this manual list many of the help topics available for an application.

To access online help from any application, select Contents from the Help menu. You can navigate through a variety of topics by choosing the buttons along the top of the window, as well as by clicking on underlined text. Hypertext links are provided to cross-reference information.

Note that a context-sensitive help button is included within many dialog boxes. When you choose the Help button, specific help information relating to that particular dialog box is displayed.

You may obtain printed information by selecting Print Topic from the File menu; you may also copy text using the Edit menu.

Some application help files contain hypertext links to other help files, called embedded help files. Embedded help files are supporting help files that describe additional features available in Chameleon ATX. These features often apply to more than one application, so you can access these embedded help files from within more than one application help file. If you access an embedded help file within an application help file and you want to return to the original application help file, click the Back button on the toolbar the number of screens you have read to return to that help file.

Starting Applications

How you start NEWT applications depends on whether your NetManage product is installed in a Windows 3.1 or Windows 95 environment.

There is no need to quit a particular application to start another. Each application is totally independent of the others. You may minimize an application to help conserve screen space, if the desktop becomes too cluttered.

Starting Applications in Windows 3.1

To start an application in Windows 3.1, choose (double-click) the corresponding icon. To start another copy of the same application, double-click the icon again. Some applications may be opened multiple times.

Starting Applications in Windows 95

To start an application in Windows 95, do the following:

1. Choose the Start button to open the Start menu, then choose the Programs command.
2. From the menu that appears, choose the program group that contains the application you want to start.
3. From the menu that appears, choose the application you want to start.

Configuration Files

Every application uses a configuration file to maintain its settings. This file is automatically loaded every time the application is started. Use the File menu to create a new default configuration (New), save the current settings (Save or Save As...), open another configuration file (Open...), or exit the application (Exit). If you quit an application without explicitly saving changes, the application displays a dialog box asking if changes should be saved.

Dialog Box Fields

A down arrow button next to a dialog box field indicates that additional predefined selections are available for that value. Click on the button and a drop-down list box appears, containing these selections. Some dialog box fields may also allow you to type in an alternate selection.

Error checking is performed on values entered into dialog boxes. Invalid values will result in a beep and display the insertion point at that field. Other dialog boxes may not permit an action until all required fields have been supplied: for example, you cannot send mail until you specify the destination address.

Fields in which IP addresses are entered have a red button to the left of the first field. You can use this to clear the entire address entry.

Status Bar

All applications provide you the option of showing or hiding the application's status bar. The status bar is displayed at the bottom of the application window, where it shows messages and provides status and statistics about the current application.

Toolbar

The Toolbar menu item allows you to display or hide an application's toolbar. The toolbar gives you quick mouse access to several of the application's common functions.

Smart Buttons

The option to display toolbar icons, including each icon's description, is available in all applications.

Changing Column Width and Order

In some applications you can change the width of each column or field. You can also rearrange fields if you want to view them in a different order. To change the width of a column:

1. Position the mouse pointer at the right side of the column border for the column you want to resize.

2. Drag the column border to the desired size.

To move a column:

Select the column you want to move and drag it to its new position.

License Violation Detection

A license violation occurs when multiple copies of a single licensed copy of the product are present on a network. Under the license agreement, each copy of the product is for a single user only and must not be shared.

Chameleon ATX is not copy-protected and may be freely installed on any type of disk. Although you can make as many copies of the product as you like, the software notifies all illegal copies of a license violation.

Each copy of the software must have a unique serial number and key code. When the software detects a duplicate serial number on the network, a message is displayed on all duplicate serial numbered copies. This message interrupts all work in progress on the computers involved.

Note: All network activity is disabled after the license violation has been detected. The Chameleon ATX applications appear to run normally, but communication over the network is disabled.

Customer Support

Before you use Chameleon ATX, complete the registration card enclosed and mail it to NetManage. This will ensure that you will be provided with customer support. It will also allow us to keep you up-to-date on new products as they are introduced.

See the chapter about the Tech Support Wizard for more information.

NetManage provides world-wide support for its customers. We can be reached at the following:

West Coast

- ❑ Sales Phone (408) 973-7171
- ❑ Sales Fax (408) 257-6405
- ❑ Support Phone (408) 973-8181
- ❑ Support Fax (408) 973-8272
- ❑ NetManage, Inc.
  10725 De Anza Blvd.
  Cupertino, CA 95014 USA

East Coast

- ❑ Sales Phone (603) 888-2800
- ❑ Support Phone (603) 888-3500
- ❑ Fax (603) 888-0304

Internet

- ❑ Sales sales@netmanage.com
- ❑ Support support@netmanage.com

Compuserve 70640,1074

BBS (408) 257-3794, 8-N-1

International

Internet intl_sales@netmanage.com
intl_support@netmanage.com

France

- ❑ Phone +33-1-47720808
- ❑ Fax +33-1-42046599

Germany

- ❑ Phone +49-(0)8165-9470-0
- ❑ Fax +49-(0)8165-9470-147

Israel

- ❑ Phone +972-(0)4-8550234
- ❑ Fax +972-(0)4-8550122

Japan

- ❑ Phone +81-(0)332218400
- ❑ Fax +81-(0)332218484

UK

- ❑ Phone +44-(0)1483-881800
- ❑ Fax +44-(0)1483-881818

Chapter 2. Support Wizard

The Support Wizard provides the means to automatically report problems and ask questions of the NetManage Technical Support Department. It also lets you launch WebSurfer to get additional online information. By using this function you will get answers to many of your questions without calling Technical Support. The Support Wizard makes it possible to collect files, screen shots and additional information Technical Support engineers need to analyze your problem or to handle a request. Please use the Support Wizard only after reviewing the online documentation and Technical Support's Web resources.

You are welcome to submit any type of question using the Support Wizard. It can be a feature request, a question about a particular problem or any other issues relating to the NetManage product line.

Submitting a Request

You can submit a request by selecting the Support Wizard option in the Help menu. A series of dialog screens will lead you through a step by step procedure for registering yourself, reporting the problem, creating a log file (if applicable to the application), and sending the data.

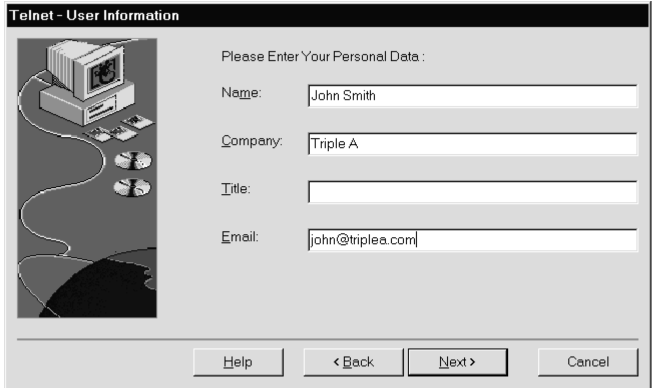
The following sections describe the procedure.

Understanding the Submittal Process

The Guide screen explains the Support Wizard and the submittal process. It also lists important numbers and addresses for contacting NetManage. After you have read about the process, choose the Next button to continue.

Registering

Complete the form on the User Information screen to register yourself. Your name and e-mail address are required. If you do not enter data in a field you will receive a message telling you what is missing. In this case, choose the OK button and enter the missing information.

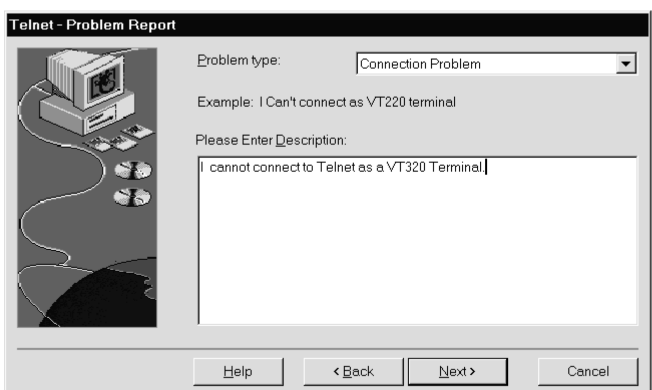


Field	Description	Required
Name	Enter your name.	Yes
Company	Enter the name of the company you represent. If you are a private user, enter your name.	No
Title	Enter your job title.	No
e-mail	Enter the e-mail address you would like Technical Support to use to contact you. In most cases, all contact will be by e-mail.	Yes

Choose the Next button to continue to the next screen where you can enter additional, optional user information

Describing the Request

Use the Problem Report screen to describe the problem.



1. Select a problem category from the list in the Problem Type drop-down menu. If one of the predefined options does not describe your problem, choose the Other option.

2. Add a short explanation in the Description area. You must enter a description to continue. Try to make it concise but clear, to enable the support team to understand your request.

3. Choose the Next button to continue.

Capturing Data

The Snap and Log screen for capturing data is application dependent. You will only see this screen if the application supports the submittal of log files and/or screen shots. Applications that do not support log file submittal will display the Files to Send Screen.

To submit a log file and/or screen capture:

1. Select Run Diagnostic Session.

2. Choose the Next button to continue to the Log Screen.

If you do not need to submit a log file or screen shots even though the application supports it:

1. Select Skip Diagnostic Session.

2. Choose the Next button to continue to the Files to Send Screen.

Sending a Log File

1. Select the Always on Top option to keep the log file on top. This will let you see exactly what is being entered in the log file as the session progresses.

2. Choose the Start button. This will open a log file of your entire session and change the Start button to a Pause button. If there are parts of your session you do not want to record, such as your password, choose the Pause button. When recording pauses, the button changes to Resume. No data is recorded in the log file until you choose the Resume button.

Note: If you want to open a log file but are already connected to a host, it is recommended that you close the connection. Then restart the Support Wizard and connect again. This will ensure that the log file includes only data relevant to the specific problem.

Sending a Screen Shot

You can also submit a screen capture. The screen capture can be in addition to or instead of a log file.

To send a screen shot:

1. Select the Screen Shot option on the Log Screen. A Snapshot Description screen will appear. You must enter a short description of the screen you have captured or you will not be able to continue. If you want to send the entire desktop, select the All Screen option before choosing the Screen Shot button.

2. Choose the OK button to return to the Log Screen.

Selecting Files to Send

The Files To Send screen lists files related to the session. You can then decide which files to send. These files are compressed and zipped into one file that is automatically sent to NetManage. Your choices are:

- Add File. Choose this option to see the Add File Screen where you can select an additional file you want to send, if it is not in the existing list. Choose the OK button to continue. The next screen will help you send the data.
- Select All. Automatically selects all files in the list.
- Deselect All. Automatically deselects all files in the list.

Choose the OK button to continue.

Sending Data

The final step in the submittal process is to send the data. Indicate the way you want to send the data by selecting one of these options.

- Send via the Internet

  Select this option to automatically send your files to the Technical Support team. Files are sent by e-mail or FTP depending on their size.

  Choose the Send button. If Z-Mail® Pro is running, the Finish screen is displayed. This screen notifies you that your request has been submitted to the Support department. Choose the Finish button to return to the main application.

  Note: Z-Mail® Pro must be running to enable the files to be sent. If you are not logged into Z-Mail® Pro, the Z-Mail® Pro Login Screen will be displayed. Enter your user name and password and choose the OK button. Z-Mail® Pro will automatically be activated.

  To enter an e-mail or FTP address other than the default:

  - Choose the Advanced Button. This displays a screen requesting the alternate address to which you want to send files. Enter the address.
  - Choose the Use Default button to cancel the alternate address and return to the How to Send Data screen.
  - Choose the Use Alternate button to return to the How to Send Data screen with the alternate addresses.

- Save to Directory

  This option lets you save your file to a specific directory and send it later from a Internet connected computer, or by surface mail, if necessary.

  Note: You may want to use this option if your computer is not connected to the Internet, and you cannot send the files directly. You can save the files to a directory, copy them to diskette and take them to a machine that is connected.

  - Enter the directory name or choose the Browse button to see a list of your available directories. You will then receive a dialog stating the name of the file.
  - Use a MIME compatible Internet Mail application to send the file as a MIME attachment. Do not send this data in uuencoded or any other format. For information on how to send this file via the Internet, choose the Help button on this screen.

  Note: If you are using surface mail, please use the NetManage mailing address closest to your location, and send it to the attention of Technical Support. Refer to any NetManage manual for a listing of NetManage offices.

- Choose the Save button to continue. The Finish screen is displayed. This screen notifies you that the file was saved to disk, the file name and how to submit it at a later date.
- Choose the Finish button to return to the main application.

Launching WebSurfer

In addition to submitting a request, the Technical Support function lets you launch WebSurfer by selecting one of the following options in the Help menu to get more information. Although these are separate options in the Help menu, they are also part of the Support function. You should review the online material before contacting the Support department.

Help Option	Web Site Launched
About NetManage	Takes you to the NetManage home page (http://www.netmanage.com) where you can learn more about the company and its products.
Register With Us	Takes you to the NetManage Product Registration Web page. Fill in the form to register and ensure that you will be provided with the latest updates on new products as they are introduced.
Technical Support Web Site Home Page Application FAQ	Technical Support Web Site opens a drop-down menu with Home Page and Application FAQ options. Selecting Home Page takes you to the Technical Support Web page where you can obtain additional technical information. Application FAQ brings you to a Web page containing answers to frequently asked questions.
Online Manual	Takes you to the NetManage Online Manual Web page. You will see an index of chapters that you can select from and review online.

Chapter 3. Go IntraNet!

Go IntraNet! is a feature that lets you easily start other NetManage applications. The menu includes specific configuration data, allowing you to select sessions, destinations, or programs.

This feature can be activated either as a menu in a NetManage application or by selecting the Go IntraNet! icon on the NEWTToolbar. When a command is selected, the appropriate application is started with the requested destination or from a list of previously defined .CFG files, connection profiles, history lists and hotlists. You can also use Go IntraNet! to copy selected text into Z-Mail® Pro and WebSurfer applications. Go IntraNet! is simply a faster and more efficient way to perform these operations.

Activating NetManage Applications

The following sections describe the procedures for using Go IntraNet! to activate NetManage applications.

General Information

The first command of Go IntraNet! changes every time you open the menu. However, in all cases, selecting this command displays general information about NetManage from a local Web page which means you do not have be connected to the Internet to access this page.

Shoot Selected Text As

Select the Shoot Select Text As command to copy selected text into Z-Mail® Pro and WebSurfer applications. This command is displayed only if the application allows text selection. NEWTToolbar allows copying selected text from any Windows application. Copying information with this command in Go IntraNet! produces the same results you would get if you copied from the Windows clipboard except you do not have to leave your current application.

The Shoot Select Text As command lets you place selected text in the following locations:

- address or body of Z-Mail® Pro message
- URL destination of WebSurfer

Copying Selected Text Using NEWTToolbar

To use Go IntraNet! to copy selected text into the URL destination of WebSurfer, mail address or mail text of Z-Mail® Pro using NEWTToolbar, do the following:

1. Select the text you want to copy from your current application.
2. Choose Go IntraNet! from NEWTToolbar.
3. Choose the WebSurfer URL command from the drop-down menu of the Shoot Select Text As command. The WebSurfer main window appears with the selected text in the URL destination.

   OR: Choose the Mail Address option from the drop-down menu of the Shoot Select Text As command A new mail message appears with the selected text in the Address field.

   OR: Choose the Mail Text option from the drop-down menu of the Shoot Select Text As command A new mail message appears with the selected text in the body of the mail message.

Copying Selected Text From an Application

To copy selected text into the URL destination of WebSurfer, mail address or mail text of Z-Mail® Pro from an application, do the following:

1. Select the text you want to copy from your current application.
2. Choose the Go IntraNet! menu of the current window.
3. Choose the WebSurfer URL command from the drop-down menu of the Shoot Select Text As command. The WebSurfer main window appears with the selected text in the URL destination.

   OR: Choose the Mail Address option from the drop-down menu of the Shoot Select Text As command A new mail message appears with the selected text in the Address field.

   OR: Choose the Mail Text option from the drop-down menu of the Shoot Select Text As command A new mail message appears with the selected text in the body of the mail message.

Note: If you are not logged into Z-Mail® Pro, the Z-Mail® Pro Login Screen is displayed. Enter your user name and password and choose the OK button. Z-Mail® Pro is automatically activated.

FTP

To activate FTP select the FTP command. By default, you can activate the NetManage FTP Server or an anonymous FTP. If you have created your own connection profiles, they will also be listed.

1. To activate an anonymous FTP, choose the FTP To command. You will be prompted for the host name. Either type in the host name or select one from the drop-down list. This list contains a history of the last 20 sites you have FTPed through Go IntraNet!.

2. To activate the NetManage server or one of your connection profiles, select the server name from the option list and the relevant FTP window is displayed. Fill in the fields as you would in any FTP application.

Terminals

To activate one of the Host connectivity applications: Telnet, TN3270, TN5250, Rlogin, or Asynch select the Terminals command. Go IntraNet! will prompt you for the host name and display the Connection Dialog.

Telnet, Async, TN3270 and TN5250 display all the connection profiles and let you enter a new host name. The history list contains the last 20 hosts entered. Rlogin displays all the connection profiles defined.

WebSurfer

To activate a specific URL on the WebSurfer, select the WebSurfer command. To enter a URL:

1. Choose the Goto URL option.

2. Enter the URL in the text box or select a URL from the drop-down list. This list contains a history of the last 20 URLs entered.

Upon installation you will be able to select the NetManage home page or a list of default URLs in addition to the Goto URL option. As you create folders with your favorite URLs (hotlists), they will also be displayed hierarchically.

To go to a URL within a folder just choose the item.

Forum

To connect to an online news group and activate the Forum application, select the Forum command. Type a newsgroup name in the dialog box or select a server from the list provided. For every server you will see a list of subscribed news groups.

To open a particular news group:

1. Choose the Open Newsgroup option from the drop-down menu.

2. Type the newsgroup in the text box or select a newsgroup from the drop-down list. This list contains a history of the last 20 newsgroups entered.

Note: If you are not logged into Forum, the Forum Login dialog is displayed. Fill in the required information and you will be logged in automatically.

Z-Mail® Pro

To activate Z-Mail® Pro or related applications from the E-Mail and Messaging group, select the Z-Mail® Pro command. You can send Tag-it! or Phone Tag memos or start Outbox.

To send mail:

1. Choose the Send Mail To command from the drop-down menu.

2. Enter any e-mail address or select an address from the drop-down list. If you are not logged into Z-Mail® Pro, you will see the Z-Mail® Pro Login Screen. Enter your user name and password and choose the OK button. Z-Mail® Pro will automatically be activated.

To send mail to NetManage Sales or Support departments, choose the corresponding option from the option list.

To retrieve mail, choose the Retrieve Now command.

InPerson

To activate InPerson, select the InPerson command. Choosing the Invite: option prompts you for the name of the person you want to conference with and makes the connection.

Choose Invite to see a list of predefined members in the phone book. Either choose a name from the list or add any other name.

Talk

To activate Talk, select the Talk command. Choose Talk With: and you will be prompted for the remote user name. Users with an existing conversation profile will also be listed in the list box. For more information on the Talk address format, refer to the Talk chapter in this manual.

Gopher

To activate the Gopher application using settings in the gopher.cfg file, select the Gopher command. You can then browse thousands of public Gopher information servers.

Search

To search the Internet, select the Search command. This activates Archie, WWW or Rapid Search. In the dialog box, enter the terms you want to search for. Otherwise, select one of the terms from the drop-down list box. This list is a history of the last 20 terms you searched.

Servers

Select the Start Servers command to activate FTP Server, WWW Server, Directory Server, or Modem Server.

Chapter 4. Telnet and ASCII Terminal Emulation

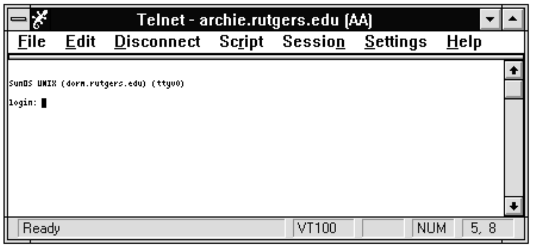
The Telnet and ASCII Terminal application is a terminal communications program that allows you to connect your PC with a UNIX or VMS host computer and various types of BBS (bulletin board service), using the following connection types:

- Telnet TCP/IP protocol
- Rlogin
- Async communications protocol

Telnet and ASCII Terminal can be invoked in several ways as described in online help. The typical connection is over an Ethernet or Token Ring LAN. The connection may also be established using a modem on both the local PC and the remote host computer, or an RS232-C cable can be used to connect the PC to the remote host computer. (RS232-C is the physical-layer protocol standard used for asynchronous communications.)

Note: An animated icon of a telephone indicates that a connection is established when the Telnet and ASCII Terminal application is minimized.

The current preferences and host name can be saved to a configuration file (default is TELNET.CFG). The configuration file is loaded when the Telnet and ASCII Terminal application is started.



Supported Terminal Types

The Telnet and ASCII Terminal application allows you to emulate the following terminal types: TTY, ANSI, ANSI-BBS, ANSI-SCO, AT386, VT52, VT100, VT220, VT320, VT340, VT420, TVI950, TVI955, WYSE50, and WYSE60.

Starting Telnet and ASCII Terminal

As with all the terminal emulator applications, there are several methods for starting the application. One way is to use the command line options. See online help for a list of other methods.

Command Line Options

You can use command line options to automatically connect to a host when you invoke Telnet and ASCII Terminal.

To do this for a Telnet Connect Type, select the Program Manager File/Run option (Start/Run in Windows 95) and type:

```
Telnet.exe [<host>] [<port>] ] [options]
```

where <host> is the IP address or host name to which you want to be connected.

where <port> is the TCP/IP port you want to use for the connection (usually 23)

Some options are:

Option	Meaning
-f <config>	uses specified configuration file
-k <keyboard>	uses specified keyboard file
-i	opens the Telnet and ASCII Terminal application minimized (iconized)
-s <script>	automatically executes the specified script file when starting Telnet and ASCII Terminal
-c<description>	connects to the host identified by the specified connection profile description. Connection profile names that include a space character must be enclosed in double quotes (").

For example,

```
TELNET.EXE SUNNY 23 -F  C:\NETMANAG\SUNNY.CFG  -C TEST1
```

invokes Telnet and ASCII Terminal, connects to the host Sunny on port 23, specifies the configuration file for this terminal session, and invokes the connection profile named Test1.

A complete list of options is given in the online help.

Connecting to a Host

The connection details that you must specify for the Telnet and ASCII Terminal application depend upon whether you are using the Telnet connection type, the Rlogin Connection Type or the Async Connection Type. The Telnet and ASCII Terminal Connect menu contains three tab dialogs allowing you to specify connection-type options for Telnet, for Rlogin, and/or for Async.

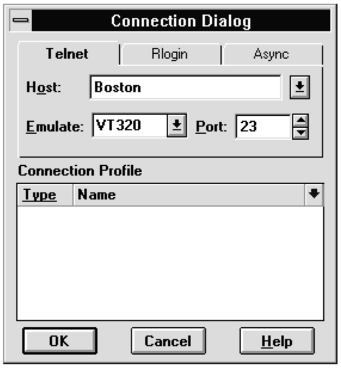
If you have already defined one or more connection profiles, you may highlight and select a connection profile from the list box. If additional details are required for the connection, the appropriate connection type dialog (Telnet, Rlogin, or Async) will be displayed automatically.

Session Menu

The Session Menu is displayed only after you make a connection to a host. The specific Session Menu differs, depending on the selected connection type. See online help for more information about Session Menu commands.

Telnet Connection Type

The Telnet connection type provides virtual terminal services for a wide variety of remote systems over a TCP/IP network, using the Telnet protocol. You can use the Telnet connection type to connect to any computer that is connected to the Internet. The Telnet connection type operates over a TCP/IP stack as a full WinSock-compliant application.



Telnet connection settings are defined using the Settings/Preferences command (Terminal, History and Printer sub-options). If you are not using a predefined connection profile, you must add the following information to connect to a host:

Host: You must provide the application with a remote host name (or IP address) to which you wish to connect.

Note: You can specify a shortened version of the host name (such as "rainbow") if the name is listed on the local domain name server or host table. Otherwise, you must specify the IP address or the full Internet host name.

Emulate: The default terminal emulation type is VT320. You can select a different emulation type from the scrolling list.

Port: Telnet's default port number is port 23. Connection to a specific port (other than port 23) is provided for additional devices with terminal login capabilities (such as network routing hubs). You may have multiple Telnet and ASCII Terminal sessions active concurrently.

If you have already defined a connection profile for a Telnet connection type, its name will be displayed in the list box.

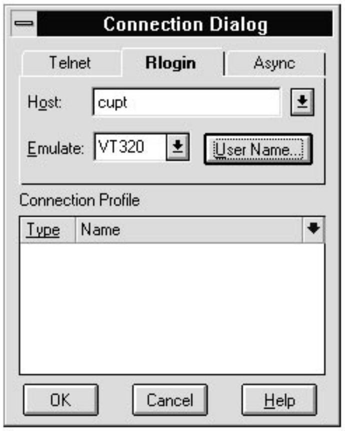
NVT Mode

When you first connect to a host, before any Telnet negotiation takes place, the connection is in NVT mode (line mode). In NVT mode, anything you type is echoed locally to your screen and is sent to the host only when you type ENTER or RETURN. This mode is indicated in the status bar, which displays "NVT" instead of the emulation type selected.

For most hosts, Telnet negotiation procedures take place as soon as you are connected to the host, at which point Telnet is automatically set to the emulation type selected. (You will see the selected emulation type displayed in the status bar.) However, a few hosts do not negotiate these options, in which case, the session remains in NVT mode. To force the session into an emulation mode, you can deselect the Session Menu/NVT command. To resume NVT mode, reselect the Session Menu/NVT command.

Rlogin Connection Type

The Rlogin Connection Type establishes a remote login session from your terminal to the remote machine (user directory), specified as host.



If you are not using a predefined Connection Profile, you must specify the following information in the Rlogin Connect dialog box:

Host: Specify the remote host to which you wish to connect.

Username: Specify both the local user name and the remote user name as follows:

- **Local User Name**: This is the username that gives you permission to access the remote machine or remote user directory.

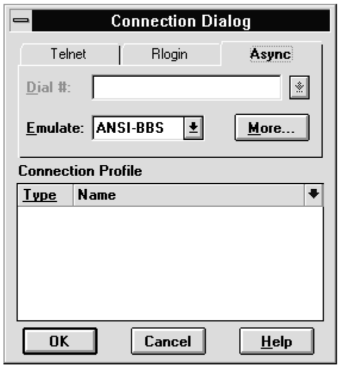
  The remote machine may have files named /etc/hosts.equiv and .rhosts, containing a list of trusted host names with which it shares user names.

  You may log in without entering a password—even if the name of your local machine is not in the /etc/hosts.equiv database—if there is a line in the .rhosts file in the home directory of the login on the remote machine with either of the following:

  - your local machine's name (if the login name is the same as yours)
  - your local machine's name and your user name.
- **Remote User Name**: This is the name of the remote user on the host, who has given your Local User Name permission to access that user's directory.

Asynchronous Connection Type

The Asynchronous (Async) connection type allows you to connect to a remote host using the serial COM port of your PC. When you select the Async connection type, the Connection dialog box does not contain a host field; rather, it contains a Dial field.

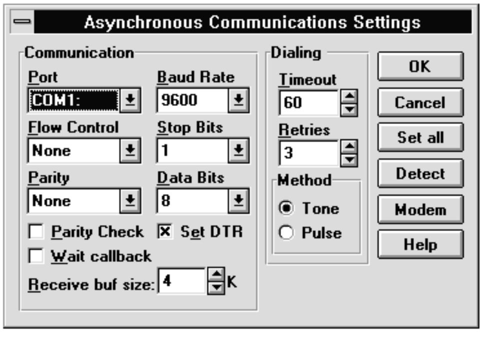


Dial: Specify the number that you wish the modem to dial. You may leave this field blank if you wish to connect directly to the COM port.

If you do not specify a phone number, the Async connection type connects to the COM port by default. When you set up this type of connection, a blank screen is displayed. You can then specify additional commands to connect and log in to a remote host, or to communicate directly with the modem. When connecting to the COM port, you can manually specify modem commands (such as the AT Command) or specify commands relevant to any other equipment connected to that COM port (such as a remote host via an RS232-C cable).

Emulate: You can use the default terminal emulation, ANSI-BBS, or select another terminal emulation from the list. For example, when the ANSI-BBS emulation is selected, the default terminal setting is 25 lines, the default font is NewtPC, and the default background is black.

More: You can choose the More button to display the Asynchronous Communication Settings dialog to define or modify your settings before you connect. Alternatively, you can also define these settings in Settings/Preferences/Async menu option.



Using a Connection Profile

If you have already defined a Connection Profile that meets your current needs, you can select the connection profile description (name) from the list box and choose the OK button, or double-click the description name.

The list box contains two column headings, as follows:

- **Type**: Indicates whether the connection profile is for an Async, Telnet or Rlogin connection type.
- **Name**: Indicates the name specified for the connection profile description via the Settings/Connection Profile... command.

You can control the order of the connection profile displays in either or both of the following ways:

- Double-click on the Type or Name column heading to sort all connection profiles according to that characteristic.
- Double-click on the arrow in the upper right corner of the list box to sort the connection profiles in ascending or descending order.

Working in a Session

When a connection has been made and you have logged in to the remote host, your PC performs all functions offered by an actual terminal connected to a host.

In addition to the terminal session features, you will also have access to other features provided by using a PC with MS-Windows and Chameleon ATX installed, such as:

- Fonts
- Log file
- Capture to file
- Cut and paste
- History buffer
- Clear and reset
- Colors
- Recording scripts

For general information about fonts, colors, cut and paste, scripts, and the log file, see online help. The following sections of this chapter describe the Capture to File, History Buffer, Clear and Reset features.

Capture to File

Using this function, you may store the whole session as it is displayed on the screen in a text file (with the extension .cap) for later use (such as to review, edit, print, or insert the file into an e-mail message or another text file).

To use this feature, you can either choose the Capture button or choose the File/Capture to file menu command. You are then prompted to select Text Only or All data.

Text Only: Captures and stores only data displayed in the session window on a line-by-line basis.

All data: Captures and stores all communications between the host and the terminal, including both data and escape sequences.

When you have selected whether Text Only or All data are to be captured, you are prompted with a Capture incoming data to file dialog box.

In the Capture Incoming data to file dialog box, enter the file name to which data will be captured and its location. You may choose to Append the contents of this session to an already existing file by choosing the Append check box.

When the session is ended or you wish to stop the capture, you can either choose the Capture button or choose the File/Capture to File command.

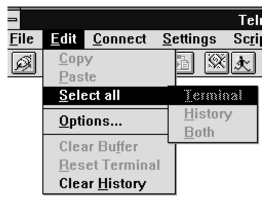
You are then prompted with a Stop Capture YES/NO dialog box.

After you have completed capturing the data to a file, you may use the file as you would use any plain ASCII text file.

Copy and Paste

The Edit menu provides Copy and Paste functions. The Edit/Select all menu option allows you to choose which part of the terminal session is to be selected, as follows:

- All of the currently displayed data (the current terminal session)
- Only the history buffer data
- Both the current terminal session and the history buffer



Note: You can also select a portion of the terminal session and/or the history buffer. To do that, drag the mouse pointer to select any part of the data on the screen.

For information on copy and paste options, see online help.

History Buffer

The history serves as a continuation of the terminal page. When a history buffer exists, the top portion of the window displays the history buffer and the bottom portion of the window displays the current terminal page.

When you have reached the terminal page size limit, a vertical scroll bar appears and you may scroll both the history and terminal page windows. If you change the window to a size that does not match the terminal page size, a scroll bar will appear as soon as the terminal window limit is reached.

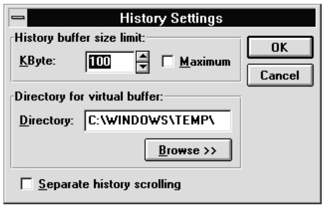
The history buffer provides virtual storage of the terminal session. Data from the terminal session is stored in the history buffer when the terminal page limit has been reached.

For example, assume that the terminal page is set to 24 lines (page size). When the 25th line of the terminal session is echoed from the host, the first line of data from the terminal session moves to the history buffer. The original first line of the terminal session will become the first line of the history buffer. The second line of the terminal session will move to the history buffer as soon as the 26th line is echoed from the host, and so on.

This means that the information from the terminal session is not lost when the terminal page limit is reached. Instead the data moves to the history buffer on a "first-in, first-out" basis.

History Settings: You set the size of the history buffer by choosing the Preferences command from the Settings menu and then choosing the History... command from the menu that appears.

The history buffer limit can be set to a specific number of kilobytes (1 KB of buffer contains approximately 8 lines of history) or to the Maximum option. The maximum size of the history is 4000 KB (which is approximately equivalent to 32000 lines). The minimum history buffer size is 0 KB. Select 0 KB if you prefer to work without a history buffer. You can select which directory will serve as the history virtual buffer. When selecting the size of the history and choosing the directory, check to be sure that there is enough free disk space.



Separate history scrolling. With this option, the terminal session window and the history window are separated from each other by a split bar. Each window has its own vertical scroll bar, allowing you to scroll each window separately.

When the Separate history scrolling option is not selected, the terminal session window and the history window share a common vertical scroll bar that allows you to scroll them as a single window.

Choose the mode that is more convenient for you.

Copying or Capturing the History: The history content is not flushed when you end the session. All information stored in the history virtual buffer may be copied to a permanent ASCII file using the copy and paste commands. If you specify the Capture to file function before the session begins, you can capture the history buffer data to a file during the session.

Clear and Reset

The Telnet and ASCII Terminal application provides the following options for clearing data:

Clear Terminal Page: Clears the current terminal page and puts the cursor in the home position at the beginning of the terminal page window. The lines cleared from the terminal page are copied to the history buffer to avoid any loss of data.

Reset Terminal: In addition to clearing the terminal page, this option resets all terminal settings to their original settings when you started the connection. Some applications change the value of these settings during the session. Use this option to clear the buffer and to reset the terminal to its initial state.

Clear History: Prompts you with a YES/NO dialog box to clear the history buffer. You can use this option to begin a new session with a clean history buffer.

The Settings/Preferences menu option displays a pop-up menu that lets you select terminal preferences for Telnet and Asynchronous connection types, and Async communications settings for an Async Connection.

Using the Settings Menu

The Settings menu allows you to specify a variety of options, including various terminal preferences, mapping your keyboard, and more. These settings are saved to a Telnet configuration file, if you select the option for saving your configuration file. Refer to Telnet online help for detailed information about using the Settings menu.

Logging a Session

The Settings menu includes the Log command, which allows you to debug programs, report problems, and record demos. See online help for a list of steps in logging a session.

Mapping Your Keyboard

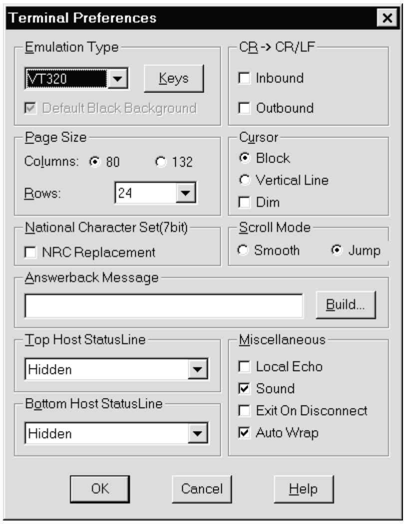
Choose the Keyboard... command from the Settings menu remap and view your keyboard. The PC keyboard displayed in the Keyboard Remap window is based on the keyboard setting specified in the International dialog of the Windows Control Panel.

For methods of mapping keys, see online help.

Terminal Preferences

Terminal preferences for Telnet and Asynchronous sessions include the following:

- Emulation type
- Keys
- CR -> CR/LF
- National character replacement
- Scroll mode
- Page size
- Miscellaneous
- Host Status Line
- Cursor
- Answerback Message



Emulation Type: Specify the emulation type to be used as a default setting when a connection is made without using a connection profile. This default emulation will appear as the selected emulation type when you choose the Connect button or the Connect command.

Keys: Allows you to select the function of some commonly used keys.

- BACKSPACE may be set to either BACKSPACE or DELETE.
- Keypad mode may be set to either Numeric or Application-dependent
- Cursor may be displayed either as Normal (programmed by NetManage) or Application-dependent.

CR -> CR/LF: The UNIX operating system usually ends each line of a text file with a carriage return (CR), whereas DOS usually ends each line of a text file with a carriage return / line feed (CR/LF). This option specifies how the end of lines are to be treated when data is transferred between a host (which uses UNIX) and a terminal (which uses DOS):

- Select Inbound to convert each CR to CR/LF when data is transferred from the host to the terminal.
- Select Outbound to convert each CR to CR/LF at the end of a line when data is being transferred from the terminal to the host.

National Character Set (7 bit): Telnet and ASCII Terminal application supports 8-bit character sets. Some computers or applications have only 7-bit character sets. If you would like to copy and use the National Character Set's special characters on a 7-bit system, select the NRC replacement check box option.

For example, you may wish to use the UK pound sign on a system that does not support the 8-bit character set. The UK pound sign is not included with the standard 7-bit character set. Use the NRC replacement option to copy it onto the 7-bit character set as a replacement for a character that you do not use.

Scroll Mode: Set whether the cursor movements will be Smooth or Jump. Jump mode uses less resources, so that sessions in Jump mode will be displayed faster than sessions using Smooth mode.

Page Size: Specifies how the display will appear when a connection is made. You may specify 132 columns (if this format is requested by the application running on the host), or you can use the default 80 columns page. You may also define the number of rows per terminal page. This parameter also sets the buffer size (as discussed previously in the "History Buffer" section of this chapter).

Miscellaneous Terminal Settings: This section of the Terminal Preferences dialog box includes the following settings:

Local Echo: Any key entered in a terminal session is passed to the remote host, and the host’s response is echoed onto the terminal display. Thus, the character that you type is not the one you see immediately reflected on the PC display: what you see is the remote host echo to your typing. If you wish to see both the terminal and the remote host responses to your entry, use the Local Echo option.

You may toggle on/off the following additional settings within the Miscellaneous section of the dialog box:

- Sound
- Exit on Disconnect
- Black Background (option is available only when ANSI-BBS is the selected emulation type)

Cursor. Set the cursor shape to one of the following options:

- Block
- Vertical Line
- Dim

Answerback Message: Specifies a sequence of characters that are sent to the host when ^E (Control E) char is received. You can choose the Build button to create a sequence of characters which has the same attribute as a keyboard macro, that is, it may contain <ESCAPE>, <ENTER>, \xHH, and so forth.

Host Status Line: Support the status line commands of certain terminal emulation types. The status lines can be set to the following values:

- Hidden: This is the default value.
- Always Visible: If supported by the selected terminal type, this value indicates that the status line should be displayed just after the connection and remain displayed until the session is disconnected.
- Host Initiated: This value allows the status line to be hidden or displayed by commands sent from the host.

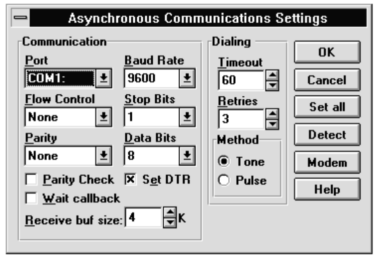
Asynchronous Communications Settings

You can use the Settings/Preferences Async menu suboption to specify the communications settings for an Async connection type.

The Asynchronous Communications Settings dialog box has two main portions: Communications Parameters and Dialing Parameters. It also has buttons on the side that provide access to additional options.

Communications Parameters

Communications parameters specify how the COM port is used for data transfer. The application can automatically select the appropriate communications parameters for your system. This is the recommended method. To enable this automatic configuration, choose the Detect button.



Note: The Detect option finds and automatically inserts the appropriate value for port, baud rate, flow control, stop bits, parity, and data bits..

Port: Select the default COM port to be used for the async session. This is the physical port to which either the modem or a RS232-C cable is connected.

Baud Rate: Select the baud rate of your local modem or the baud rate of the remote modem. The range is 300-115200 bits per second (BPS).

Flow Control: Select the way flow control is handled between the sending and receiving devices. The Flow Control options are None, CTS/RTS and XON/XOFF.

- **None**: Select this option if the remote system uses no overflow method.
- **CTS/RTS** (clear to send / request to send). Using CTS/RTS flow control, the sending device activates the RTS signal when it is ready to send data. When the receiving device is ready to receive data, it activates the CTS signal. The receiving device actively controls the receipt of data, thereby preventing an overrun.
- **XON/XOFF**. XON/XOFF causes your system to pause when the buffer fills. When a fast device is connected to a slower device, the data may be sent to the slower device faster than it can be processed. Even though devices often use buffers, sometimes the slower device cannot keep up with the rate of receipt. XON/XOFF flow control may be used to prevent the faster device from overflowing the slower device’s buffer and causing data loss. Select XON/XOFF if you don't know what flow control method is used.

In data communications environments, one device sends a message to another device to establish a communications channel. The two devices then send messages back and forth that enable them to agree on data flow speed. (The Detect option finds and inserts the appropriate value automatically.)

Stop Bits, Parity, and Data Bits: These settings have to be identical on both ends of the communications. Check with the remote end so that you can match these settings to have a usable session between your two modems.

Set DTR: Set DTR (data terminal ready) if you wish the terminal to indicate that it is powered up and ready to send and receive data.

Wait Call Back: If you are connecting to a host that provides a call-back service (dials back to your PC), you can use this option. When this option is selected, the Telnet and ASCII Terminal application dials to the host, then disconnects and waits for the host to call back.

Rx buf size: You can specify the size of the receive buffer used for the Async Communications. You may experience loss of data if the program does not process data received from the other device fast enough. The larger the receive buffer, the less data you may lose.

Dialing Parameters

These parameters specify the way the local modem dials to the modem of the remote host. Specify one of the following values, or use the default values:

Timeout. Specify the length of time in milliseconds (ms) for a timeout in case the remote end does not respond. The default is 60 ms.

Retries. Set number of retries in case a timeout occurs. The default is 3 retries.

Method. Specify Tone or Pulse to indicate which method is being used for dialing. The default is Tone.

Modem

The modem button lets you select a modem type to be used for your async connection. If you want the Telnet and ASCII Terminal application to Dial, you must first select a modem type and then type the number to be dialed in the Connect menu (or include it in a Connection Profile)

1. Select your modem model (or the most compatible one) from the scrolling list box.
2. Choose the Install button to have it installed.

Note: If you are familiar with configuring modems, you can continue to specify the details of the modem configuration when the appropriate modem name is displayed in the Modem Installed field. To specify a detailed modem configuration, choose the Setup button.

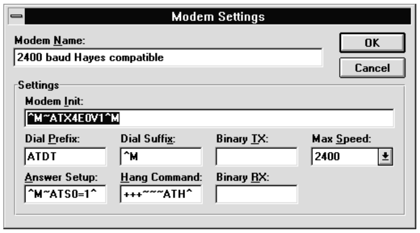
Do not change the specific settings unless you are familiar with the technical issues of setting modem parameters: consult your modem user's guide for the correct settings. Otherwise, simply leave the default settings.

The default settings will work for most configurations. Incorrect settings of the modem may cause it to miscarry the connection or to be reinitialized.

Note: NetManage supplied strings in the Modem Settings dialog are as follows:

`^M (CTRL+M): ASCII#13, ENTER`

`~ (Tilde):` time delay, each '~' is a one second delay



Detect

The Serial Port Diagnostic pop-up box is displayed when you choose the Detect button in the Asynchronous Communications Parameters dialog box. Use this option to properly configure and confirm the connection for the asynchronous connect type.

Note: Ensure that your modem is plugged in and powered on before using this option.

When you choose the Detect option, the application automatically detects the communications parameters in your PC and inserts the appropriate values (for COM port, baud rate, and so forth) in the Asynchronous Communications Settings dialog box.

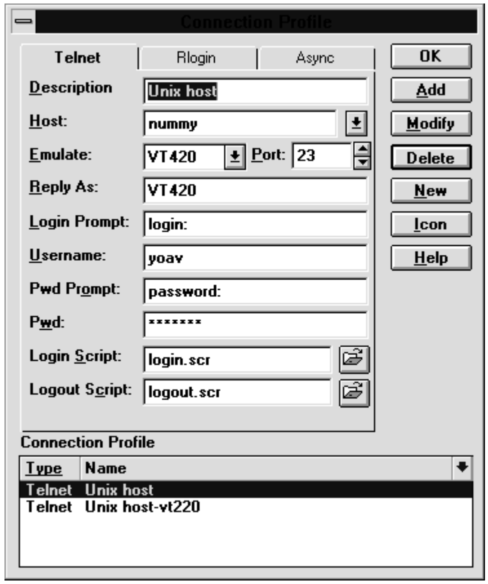
You can also use the Detect option to confirm or diagnose the status of the serial ports.

Connection Profile

Defining a connection profile is the most efficient way to establish a connection. The Connection Profile dialog box lets you specify all the necessary information for connecting to a remote host. Telnet sessions and asynchronous sessions use different connection profiles, because they differ from one another in some of their connection parameters.

Telnet Connection Profile

The Telnet Connection Profile dialog contains the following fields:



Connection Profile Field	**Meaning**
Description	Name of the connection that appears in the list box in the Connect dialog box. Must be specified.
Host	Name or IP address of the remote host to which you wish to connect. Must be specified.
Emulate	Desired emulation type. Choose from the scrolling list box. The default terminal type is VT320.
Port	The default port is 23, the Telnet communications port. This port is appropriate for most configurations. You can specify an alternate port if so desired.
Reply As	Some Telnet hosts require that the terminal identify itself differently than the standard Telnet name (for example, DEC-VT1 instead of the usual VT100). If your host needs such identification, input the appropriate emulator type into this field. You may type any character string into this field. The default reply is the same as the emulator type.

Connection Profile Field	Meaning
Login Prompt	The login prompt that you would see on your screen. Should be used if automatic login is desired. Case sensitive. Optional.
Username	The user name that you use to login. Should be used if automatic login is desired. Optional.
Pwd Prompt	The host password prompt that you would see on your screen. Should be used if automatic login includes password. Case sensitive. Optional.
Password	The password that you normally enter after you have entered your username. Component of automatic login with password. Optional.
Icon	You can save the current Connect Profile description as an icon. The icon will be displayed in a window group on your desktop, consisting of customized Chameleon ATX Terminal Emulations. By selecting this icon, you can connect from your desktop to a terminal emulation session using the options specified for this Connection Profile.
Login Script	You can type or browse to select the name of a script file that automatically logs into the host when you connect to a host by using this profile. Optional.
Logout Script	You can type or browse to select the name of an existing script file that automatically logs out of the host specified by this profile when you close the application. Optional.
Connection profile list	The list box contains the connection profiles that you have already defined.

Note: If you want to include the Login Prompt and the Password Prompt as part of your Connection Profile, you should connect and log in to the host and write down the prompts that are displayed for username and password. The specified text string must match the prompts (or a portion of the prompts) as they are displayed.

Itelnet Security Connection Profile

Itelnet is a specific type of Telnet connection. It is a security mechanism provided by Sun for the Sun/OS platform and allows users to gain access through a security gateway to other corporate internal users or out to the Internet.

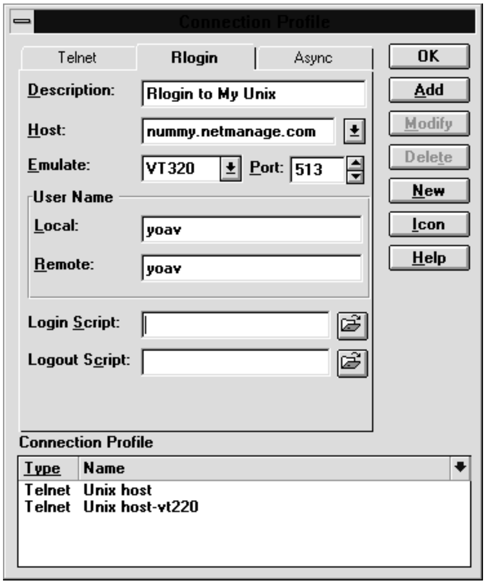
For an Itelnet connection, you need to fill out your Connection Profile dialog box as follows:

Connection Profile Field	**Field Entry**
Description:	Itelnet
Host Name:	Name or IP address of the host to which you are connecting.
Port:	Must be 4514
Emulated Terminal:	Name of the terminal you want to emulate.

Note: Fill out the remaining fields just as you would for any other connection profile.

Rlogin Connection Profile

The Rlogin Connection Profile dialog contains the following fields:

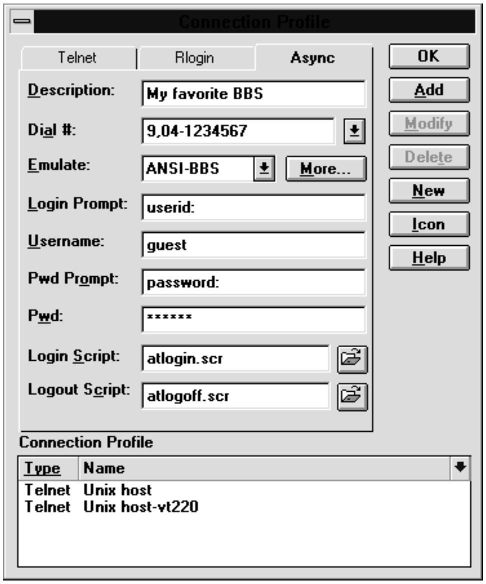


Connection Profile Field	**Meaning**
Description:	Name of the connection that appears on the List field in the Connect option dialog box. Must be specified.
Host Name:	Name or IP address of the host to which you are connecting. Must be specified.

Connection Profile Field	Meaning
Port:	The default (513) Telnet communications port.
Emulated Terminal:	Name of the terminal you want to emulate. The default is VT320.
Local Username	Specify the username that gives you permission to access the remote machine or remote user directory. The remote machine may have files named /etc/hosts.equiv and .rhosts containing a list of trusted host names with which it shares user names. Users with the same user name on both the local and remote machine may rlogin from the machines listed in the remote machine's /etc/hosts.equiv file or .rhosts file without supplying a password.
Remote Username	Specify the name of the remote user on the host, who has given your Local User Name permission to access that user's directory.
Icon	You can save the current Connect Profile description as an icon. The icon will be displayed in a window group on your desktop, consisting of customized Chameleon ATX Terminal Emulations. By selecting this icon, you can connect from your desktop to a terminal emulation session using the options specified for this Connection Profile.
Login Script	You can type or browse to select the name of a script file that automatically logs into the host when you connect to a host by using this profile. Optional.
Logout Script	You can type or browse to select the name of an existing script file that automatically logs out of the host specified by this profile when you close the application. Optional.
Connection profile list	This list contains the connection profiles that you have defined.

Asynchronous Connection Profile

The Asynchronous Connection Profile dialog contains the following fields:



The Asynchronous Connection Profile dialog box contains the following fields specific to asynchronous connection types. (See the descriptions of the Telnet Connection Profile fields for information on the fields that are the same.)

Connection Profile Field	**Meaning**
Dial	Type in the number you wish the modem to dial to when utilizing this connection profile. If you do not specify a telephone number, the async connection type connects to the COM port by default. When connecting to the COM port, you can manually type in modem commands (such as the AT command) or commands that are relevant to any other equipment connected to that COM port (such as, a remote host via a RS232-C cable).
More	Choose this button to specify communication settings to be associated with the current connection profile. See the section "Asynchronous Communication Settings" for detailed information on these settings.

File Transfer

When a connection has been made, you can transfer files using any of the following telecommunication protocols:

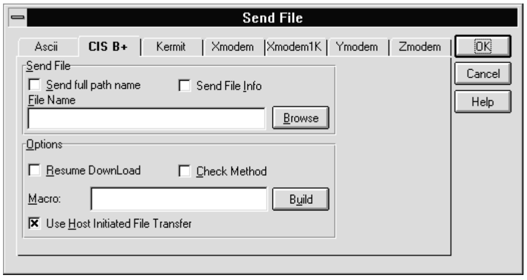
- ASCII
- CIS B+
- KERMIT (default)
- XMODEM
- XMODEM-1K
- YMODEM/Batch
- ZMODEM

Files may be transferred as binary (text and control codes) or ASCII (text only).

To transfer files between computers, both computers must use the same file-transfer protocols. The file-transfer protocols are responsible for error detection and ensuring that extraneous data is not stored in the transferred files.

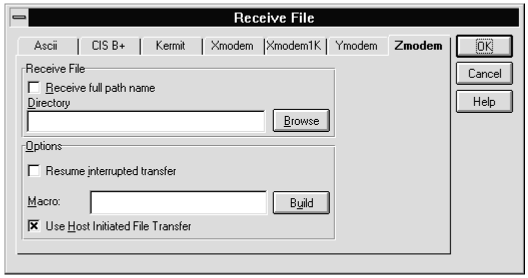
The choice of a file-transfer protocol depends on your configuration. Before transferring files, verify that the file-transfer protocol on the local computer is the same as the file-transfer protocol being used by the remote host. You may also set a default file-transfer configuration using the Settings/File Transfer menu command.

File Transfer can be used for sending files (File/Send File) or receiving files (File/Receive File).



The transfer steps are similar for sending and receiving files:

1. Select the file-transfer protocol.
2. Input the necessary information in the protocol-specific dialog box that appears.
3. Type or select the file name and its directory.



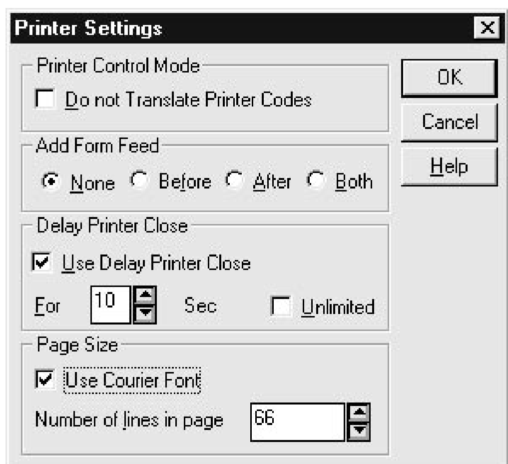
Note: The Drag and Drop feature can also be used to automatically send files from your PC to the host. Before using this feature, it is recommended that you fill in the Macro field in the Send File dialog box. Refer to Telnet online help for detailed information about the Drag and Drop feature.

Printing with Telnet and ASCII Terminal

There are several ways to print using the Telnet and ASCII Terminal application. You must set up the printer (whether it is an attached or network printer) in the Windows Control Panel. As with any other Windows application, you may select to print to any installed printer.

Printer Settings

Printer settings may be defined in the Settings/Preferences/Printer menu option.



Printer Control Mode: All print commands and data are communicated from the remote host to the printer via the Telnet and ASCII Terminal application. Sometimes the application running on the remote host may send escape sequences (such as, skip to a new page, switch to a different font, and so forth) directly to the printer. This is sometimes referred to as a printer "passthru" state. If you wish to use this mode, select the option "Do not translate printer codes" before sending print jobs to the printer.

Add Form Feed: You may select whether each print job should be separated from the next by adding a blank page before the job, after the job, or both before and after. Alternatively, you may disable this option by selecting None. There are some devices that require a Form Feed at the end of the print job to enable a printout. In such cases, select the After option.

Delay Printer Close: You can use this option to allow a specific or unlimited time delay for the printer to complete its job before the host issues a CLOSE command. This option can be used to prevent extraneous form feeds from occurring when using host printer Controlled Mode.

Some hosts send a single print job as multiple sequences of lines. Each sequence of lines is followed by a Stop Mode command, causing a brief pause. When the stop interval expires (timeout), the printing resumes with the next sequence. If you enable the Delay Printer Close command for the specified timeout, the printing will resume on the same page. Otherwise, the next sequence of lines will be treated as a separate print job.

Page Size: If the Use Courier Font option is selected, you may also specify the number of lines that will be printed on each page. The default number of lines per page is 66. You can specify from 1 through 200 lines per page. If you specify a number of lines that is out of range, the default value of 66 lines is used instead. If you do not select the Use Courier Font option, you cannot specify the number of lines in a page (the option is disabled), and the printer uses the default printer font. To print special characters as line drawings or international characters, you must select Courier Font.

Print Mode

Use the File/Print menu option to select the Print Mode. The Telnet and ASCII Terminal application supports three print modes: Screen, Auto, and Control.

Screen Mode: In Screen mode, you may print all or a selected part of the data currently displayed on the session screen. This option is also available by choosing the screen button from the toolbar.

The range options for Screen Mode are as follows:

- **All**: prints the number of lines specified as the buffer size. This may include blank lines if the buffer size is larger than the number of lines that has been used.
- **Selection**: prints only a selected number or block of lines.
- **Terminal Page**: prints only any actual data in the terminal page.

Auto Mode: In Auto Mode, incoming data is echoed to both the printer and the terminal display. The data is kept in the printer buffer until this mode is disabled, at which time it is printed.

Control Mode: In Control Mode, all incoming data is directed to the printer. Nothing is displayed on the terminal while this mode is enabled.

Additional Online Information

The online help file for Telnet and ASCII Terminal includes a description of each menu command, plus further information in the "How to Use" section:

- Command Line Options
- Setting up Telnet and ASCII Terminal
- Defining Key Mapping
- Defining a Button Pad
- Using Scripts
- Printing with Telnet and ASCII Terminal
- Transferring Files
- Dragging and Dropping Files from PC to Host
- Setting Colors

Troubleshooting

If you experience difficulties using the Telnet and ASCII Terminal application, refer to the following items:

❑ Use the Diagnose Connection command in the Help menu to make sure that the connection is operational. (If your last connection was a Telnet connection type, this option invokes the Ping application. If your last connection was the async connection type, this option invokes the Detect function.)

❑ If the window display appears incorrect, verify that emulation was specified correctly.

❑ Disconnect and change the history setting in the Preferences dialog box to scroll further back.

❑ Select Reset Size from the Settings menu if full-screen applications appear to not fit correctly within the terminal session window.

❑ Check to see that the appropriate font and size is installed if graphic characters appear incorrectly.

❑ Check to see that the appropriate font and size is installed, or scroll horizontally, if 132-column mode does not fit within the terminal session window.

Tips for Telnet Connections

- ❑ Make sure that the remote system provides a Telnet server.
- ❑ Ensure that the port is specified correctly; it should be the default (23) in almost all cases.

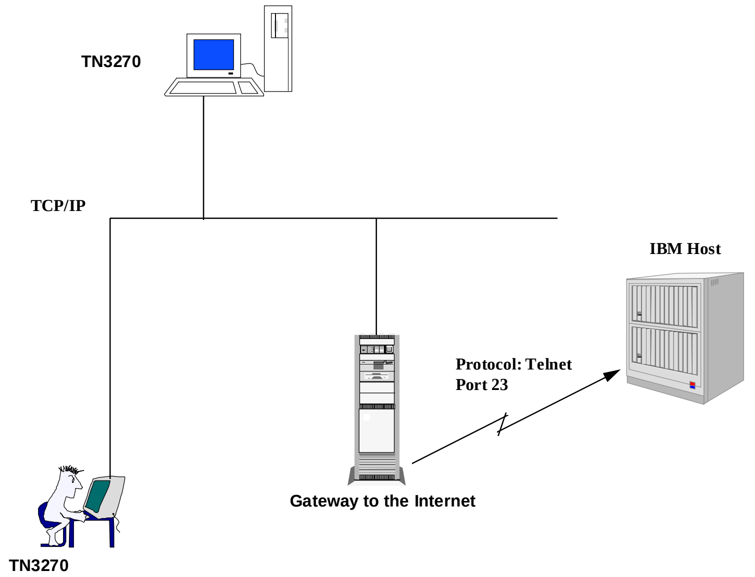
Tips for Asynchronous Connections

- ❑ You can use the Diagnose Connection Option under the Help menu or the Detect option in the Asynchronous Communications dialog box (under Settings/Preferences/Async) to check your Async connection.
- ❑ Check to see that the modem is connected..
- ❑ Check to see if the line is available. If there is only one available phone line and someone is talking on it, you will not be able to connect to the host.
- ❑ Check to see that you are using the correct COM port (the one to which the modem is connected).
- ❑ Check to see that the modem was reset.
- ❑ Check to see that the communications parameters on the local end match the communications parameters on the remote end.

Chapter 5. TN3270 Terminal Emulation

TN3270 is a terminal emulation communication program for IBM mainframe systems. This application offers 3270 protocol support for IBM hosts with IBM's TCP/IP via supported TCP/IP attachment devices.

Note: Support for other Connection types is available with the NetManage HostLink product.



Features

The TN3270 application includes the following features:

- TN3270 emulates any IBM 3278 or 3279 terminal (models 2, 3, 4, or 5 with full SAA compatibility and RFC 1041).
- The TN3270E standard is now supported according to RFC1647. You can print TN3270 sessions to the IBM 3287 printer using the 3270 Printer application provided with this release.
- International character sets are supported. The PC keyboard displayed in the Keyboard Remap window is based on the keyboard setting specified in the International Dialog of the Windows Control Panel.

- Other windows applications programmers can use the EHLLAPI interface for sending and receiving data with the IBM host using terminal emulation services.
- You may have multiple instances (up to the number of sockets) of the TN3270 application active concurrently.
- To log onto a host other than 3270, use the Network Virtual Terminal (NVT) mode.
- An animated icon signals whether the connection is active or inactive.

3270 Terminal can be invoked either with arguments {TN3270 host [port] [options]} or without arguments. A list of 3270 Terminal command line options is provided in the online help.

Note: To start 3270 Terminal from a command line in Windows NT and Windows 3.1, you can use the Windows Program Manager/File Menu/Run... command. In Windows 95 use Start/Run.

The current preferences and host name can be saved to a configuration file (default is TN3270.CFG). The configuration file is loaded when the TN3270 application is started. You can have different configuration files for each session. The default configuration file the next time you open a connection will be the last configuration file that you saved.

Note: An animated icon of a telephone labeled "3270" indicates that a connection is established when the TN3270 application is minimized.

Starting 3270 Terminal

As with all the terminal emulator applications, there are several methods for starting the application. See online help for detailed information.

TN3270 Command Line Options

General Information on starting emulators from a command line and a list of common command line options is included in the online help.

Connecting to a Host

To connect to a host system on the network, choose the Connect command from the menu bar. If a connection profile has already been defined, you can select a name from the Connection Profiles list box.

Note: The Connect menu contains five tab dialogs: TN3270 (Telnet), NW4SAA, Coax, NT Server Connection, and 802.2. However, only the TN3270 tab dialog is usable; the other connection types are available only with the HostLink product.

Defining a Connection Profile

A connection profile allows you to set and store a list of frequently used connections. The connection profile names will be listed in the Connect dialog box, and allow you to connect automatically to a host. To create a connection profile, you must define a description and host name (or IP address). You may also specify additional options, as desired (see "TN3270 Connection Options").

TN3270 Connection Type

You can use the TN3270 Connection Type to connect to any 3270 IBM host that has a TCP/IP interface, connecting that host with the Internet. The TN3270 Session is performed over a TCP/IP stack as a full WinSock compliant application.

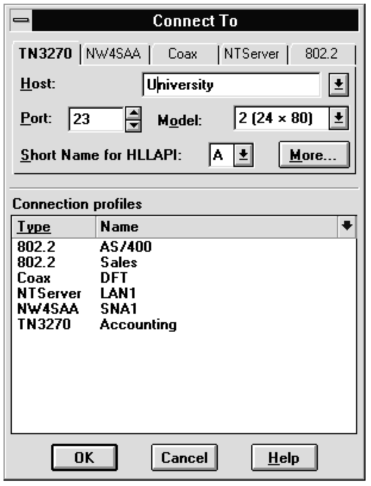
TN3270 Connect Options

If you are not using an existing Connection Profile from the list box, you can add the following information to complete the connection details:

Connect Option	Meaning
Host	Specify the TCP/IP name or IP address of the host that you want to connect to. If you have not specified a host in a Connection Profile, you must specify a host in the Connect dialog. There is no default value for this option. **Note:** You can specify a shortened version of the host name (such as "rainbow") if the name is listed on the local Domain Name Server or Host Table. Otherwise, you must specify the IP Address or the full Internet host name.
Port	Specify the TCP/IP port that your host uses for Telnet. In almost all hosts, the port number is 23 (default).
Model	Defines the maximum screen size to be used. The default model is Model 2. The appropriate Model depends on the application, which is running on the remote Host, and the user's display capabilities. 3270 Terminal supports the following models: • Model 2: 24 Lines, 80 Columns (default) • Model 3: 32 Lines, 80 Columns. • Model 4: 43 Lines, 80 Columns. • Model 5: 27 Lines, 132 Columns. **Note:** Model 2 is always used for the initial login phase.

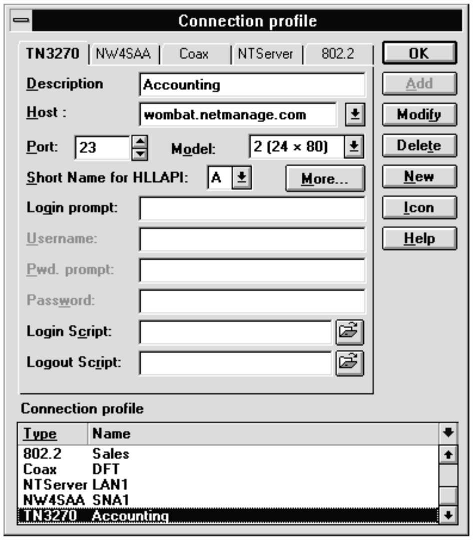
Connect Option	Meaning
Short Name for HLLAPI	HLLAPI short name that will be assigned to the session if available. When you create the first session, the default name is A. The default name for the second session is B, and so forth. You can choose any letter from the list (A through Z).

The More button allows you to specify advanced settings. See "Advanced Options for TN3270 Connections."



TN3270 Connection Profile

To create a Connection Profile choose the Connection Profile command from the Settings menu.



The TN3270 Connection Profile dialog box contains the following fields:

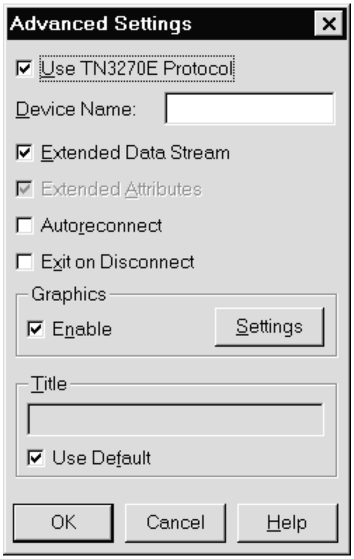
Connection Profile Field	**Meaning**
Description	Name of the connection that appears in the list box in the Connect dialog box. Must be specified.
Host	Name or IP address of the host to which you are connecting. Must be specified.
Port	The default port is 23, the Telnet communications port. This port is appropriate for most configurations. You can specify an alternate port number if so desired.
Model	Terminal model (defines the maximum screen size used). You can accept the default or select another screen size: Model 2: 24 x 80 (default) Model 3: 32 x 80 Model 4: 43 x 80 Model 5: 27 x 132
Short Name for HLLAPI	HLLAPI short name that will be assigned to the session if available. When you create the first session, the default name is A. The default name for the second session is B, and so forth. You can choose any letter from the list (A through Z).

Connection Profile Field	**Meaning**
Login Prompt	The login prompt that you would see on your screen. Should be used if automatic login is desired. Case sensitive. Optional.
Username	The username that you use to login. Should be used if automatic login is desired. Optional.
Pwd Prompt	The host password prompt that you would see on your screen. Should be used if automatic login includes password. Case sensitive. Optional.
Password	The password that you normally enter after you have entered your username. Component of automatic login with password. Optional.
Icon	You can save the current Connect Profile description as an icon. The icon will be displayed in a window group on your desktop, consisting of customized Chameleon ATX Terminal Emulations. By selecting this icon, you can connect from your desktop to a terminal emulation session using the options specified for this Connection Profile.
Login Script	This is optional. You can type or browse to select the name of a script file that automatically logs into the host when you connect to a host by using this profile.
Logout Script	You can type or browse to select the name of an existing script file that automatically logs out of the host specified by this profile when you close the application. Optional.
Connection profile list	The list box contains the connection profiles that you have already defined.

Note: If you want to include the Login Prompt and the Password Prompt as part of your Connection Profile, you should connect and log in to the host and write down the prompts that are displayed for username and password. The specified text string must match the prompts (or a portion of the prompts) as they are displayed.

Advanced Options for TN3270 Connections

To specify additional options, choose the More button in the Connect To or Connection Profile dialog box. An Advanced Settings dialog box appears with the following options. Select the desired options, and choose the OK button.

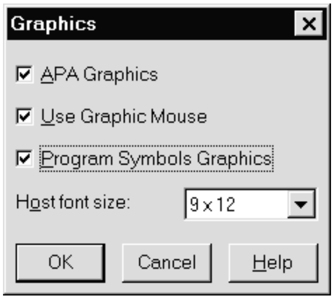


Advanced Options	**Meaning**
Use TN3270E Protocol	Select this option if you want to be able to print this terminal session to an IBM 3287 printer using the 3270 Printer application..
Device Name	Device name is similar to an "LU name." This option allows the TN3270 session to be assigned to a particular device name. The TN3270E protocol allows the TN3270 server to organize terminal and printer device pools. The server can assign these device pools for use by specific applications and/or users.
Extended Data Stream	Select Extended data stream if your Display terminal can support it, and if the application running on the Host requires it. The Extended Data Stream option emulates an IBM terminal that supports structured field commands. If you select the Extended data stream feature, the Extended Attributes option is automatically selected. If you do not select this option, you will not be able to use structured field support.
Extended Attributes	This option enables individual characters within a field to be assigned different Attributes (such as highlighting, colors, reverse video, underline and blinking modes). Otherwise, Attributes affect the entire field. You may select to use Extended Attributes as a separate option, even if you do not select to use the Extended Data Stream option.

Advanced Options	Meaning
Autoreconnect	Automatically reconnects the terminal to the host when the host disconnects you.
Exit on Disconnect	Exits from the TN3270 application (and closes it) when you are disconnected from the Host.
Graphics	Check the Enable option to enable graphics support. Choose the Settings button within this section of the dialog box to display another dialog box with specific graphics options. (see the section "Graphics Settings").
Title	This is the name that appears in the title bar of the terminal emulation window when you are connected to the specified terminal emulation session. If you do not specify a title name, the host name appears in the title bar by default.

Graphics Settings

You can set graphics options by choosing the Settings button within the Graphics section of the Advanced Settings dialog box (To access the Advanced Settings dialog box, you must choose the More button within the Connect or Connection Profile dialogs.)



The Graphics Settings are as follows:

Graphics Options	Meaning
APA Graphics	Provides IBM3179G and IBM3192G terminal emulation, including APA graphic support This option lets you show graphs on your Session display and use the mouse as a graphic pointer. These are vector graphics.

Graphics Options	Meaning
Program Symbol Graphics	Lets the host program the fonts to be displayed on the terminal. The default font size is 9X12.
Host font size	You may select a different font size from the pop down list in the dialog box. (The font size selected by the Fonts command from the Settings menu must match the font size selected for the Program Symbol Graphics, if the Program Symbol Graphics option is enabled.)

Capturing Files

TN3270 allows you to capture a session to a printer or a file. The Capture command is in the File menu. The parameters for the capture may be set by using the Capture command from the Settings menu. See online help for a description of these settings.

Scripts

A script is a file of commands that automate one or more procedures that can be done with the terminal emulation application. You can choose the language for recording the script and the run mode for playing it. The recording language can be: VBA or Simplified. The run mode can be: VBA or Cham4.5.

- **Simplified**: Provides a limited set of commands that can be used to create a script for automated login. This type of script can be run from Telnet, TN3270 and TN5250 Terminal Emulation applications, either by specifying the script name in the connection profile or by running the script after a connection has been made. If you edit a script using the Simplified Script Type, you must run the script as Cham 4.5.
- **VBA**: Refers to NetManage Visual Basic for Applications. This Script Type is compatible with the standard Microsoft VBA and supports Visual Basic™ syntax. It also supports OLE automation and function calls for external DLLs. If you edit a script using VBA, you must also run it using VBA.

For additional information, see the Visual Script Editor/Visual Script Player chapter of this manual and the online help.

Using the Edit Menu

The Edit menu provides copy and paste functions for text and graphics. (The graphics options are available only if enabled through the Advanced Settings of the Connect or Connection Profile dialog.) You can use these functions to extract text (or graphics) to use in other applications. All text pasted to the emulation window is transmitted directly to the host system. For additional information, see online help.

Using the Settings Menu

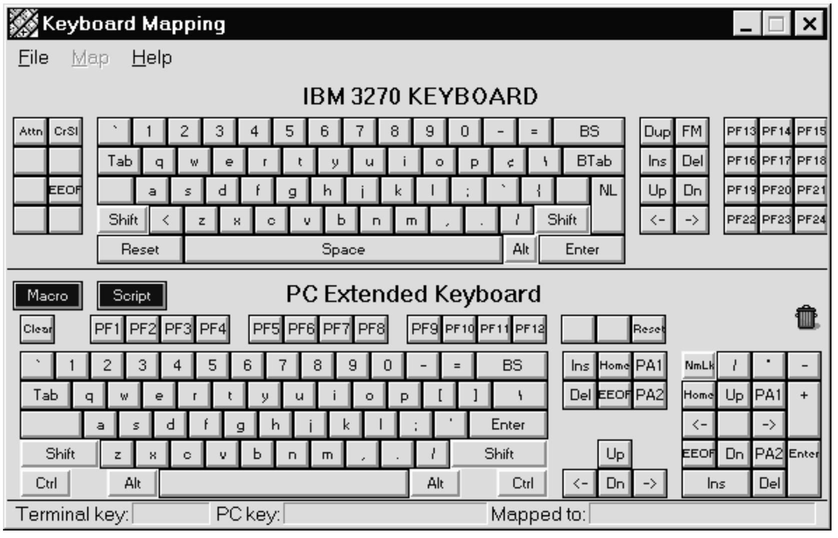
The Settings menu allows you to specify a variety of options, including various terminal preferences, such as cursor type; settings options for file transfer and file capture; and more. These settings are saved to a TN3270 configuration file, if you select the option for saving your configuration file. Refer to TN3270 online help for detailed information about using the Settings menu.

Logging a Session

The Settings menu includes the Log command, which allows you to debug programs, report problems, and record demos. See online help for a list of steps in logging a session.

Mapping Your Keyboard

Choose the Keyboard... command from the Settings menu remap and view your keyboard. A display of a PC extended keyboard and a IBM 3270 keyboard appears. The PC keyboard displayed in the Keyboard Remap window is based on the keyboard setting specified in the International dialog of the Windows Control Panel.



Setting Hotspots

Hotspots allows you to enable word selection and define PF templates. After a hotspot has been defined, you can activate the hotspot during a session by selecting a hotspot (using the right mouse button) displayed on the screen. Selecting a predefined hotspot invokes its associated function-key call or terminal command. A hotspot can also be used to invoke a Visual Script. Refer to TN3270 online help for detailed information about setting Hotspots.

Transferring Files between a PC and a Host

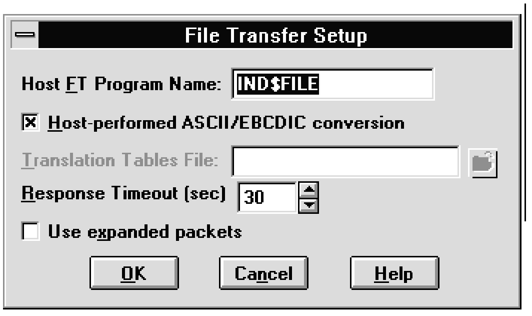
To transfer files between a PC and a host, perform the following steps:

1. You may choose the File Transfer Setup command from the Settings menu to specify general file-transfer settings (or you may access the File Transfer Settings dialog box by using the More button in the Send File or Receive File dialogs).

2. Choose the Send File or Receive File command from the File menu to perform the actual transfer.

File Transfer Setup Options

Choose the File Transfer Setup command from the Settings menu to specify general file-transfer settings or use the More button in the Send File or Receive File dialogs to select these options.



Host File Transfer Program Name: Specifies the name of the host program that executes file transfer operations. The default name is IND$FILE. If the operating system on the host uses a different file-transfer program, you must type in the appropriate file-transfer program name.

Translation Table File: If nonstandard ASCII-EBCDIC translation is required, you can provide a file that contains the conversion table. A sample translation file, EBASCII.TAB, is automatically copied into your "`netmanag`" directory.

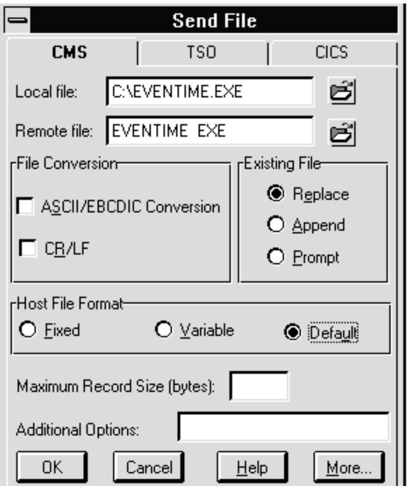
Response Timeout: Specifies maximum time, in seconds, that the terminal emulation waits for a host response to a PC request. The default is 30 seconds.

Use Expanded Packets: The default packet size is 2K. If you know that the remote host supports packets larger than 2K, you may choose the Use expanded packets option. This option takes effect only if the extended data stream option (in the Connect To dialog box) has also been configured.

Sending Files

Choose the Send File command from the File menu to transfer a local PC file to the remote IBM host. The Send File options differ somewhat for the different host operating systems. Select the appropriate tab dialog for your host operating system: CMS, TSO, or CICS. For detailed information on the options for each of these operating systems, see online help.

For example, the CMS tab displays the following Send File dialog box:



The following fields need to be completed:

Local File. Type the local file name (with the full path) or use the Browse button to select the file.

Remote File: If you do not select or type a remote file name, the file is copied to the default disk of the remote host with the same name as the local file.

Note: Remote file names do not contain a period before the file extension. IBM hosts use a space instead.

If you wish to select a different location, use the Browse button. The TN3270 then displays a message box announcing that it is "Filling host disk list."

When the file list is ready, you are prompted with the CMS File List dialog box.

Disk: Select A through Z disk name from the Disk list. The asterisk ("*") indicates the default disk. TN3270 will display file lists only for the disks that you are permitted to access. Choose the Browse button to get the list of files currently stored in the selected disk.

File Conversion. The default transfer is in binary mode. You may select to transfer the file using the ASCII/EBCDIC Conversion option. This converts PC ASCII files into IBM EBCDIC format. Each ASCII file line ends with CR/LF. You may choose to treat each CR/LF as a end of data in a field by selecting the CR/LF check box.

Existing File: You may select the following options for transferring to an existing file:

- **Replace:** Overwrites an existing host file, or creates a new file if the host file does not exist. If the host file exists, its attributes are preserved.
- **Append:** Adds the file to the end of the host file, or creates a new file if that file does not exist.
- **Prompt:** Checks if the host file exists. If it exists, you are prompted to replace the existing file or abort the transfer.

Host File Format. The host file may be formatted with fixed or variable field length. Select Fixed, Variable, or Default for the transferred and converted file. A line of data in a PC file is converted into one field of the host file.

Note: If the data in a line exceeds the field length, the extra data is lost. It is important to remember this before selecting a fixed field length.

Additional Options: Valid commands are as follows:

- **Quiet -** The transfer proceeds without displaying any message boxes during the operation.
- **No Progress -** The transfer proceeds without displaying the progress box during the operation.

Use the More button if you wish to display "File Transfer Setup" dialog box.

When you have completed specifying the file-transfer settings. choose the OK button to start the Send File operation. A dialog box is displayed with all the transfer details.

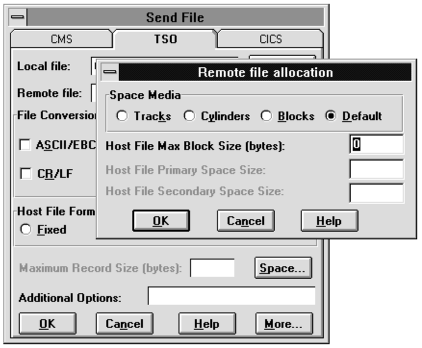
Fixed details are:

- Local file name
- Remote file name
- Status of operation
- Total size of transferring file

Dynamically changing details are:

- Bytes transferred
- % transferred
- Actual FT (file transfer) time elapsed
- Estimated completion time

The TSO and the CICS tab dialog boxes for File Transfer contain similar fields. The TSO tab dialog box has an additional Space button, which displays a Remote File Allocation dialog box for specifying additional information regarding the file being transferred.



Note: The CICS tab dialog box adds an option to send a clear signal to the host prior to beginning of the transfer operation. Also, the CICS tab dialog box omits the record size options.

Receiving Files

Choose the Receive File command from the File menu to transfer a file from a remote IBM host file to the local PC. The Receive File options differ somewhat for the different host operating systems. Select the appropriate tab dialog for your host operating system: CMS, TSO, or CICS. For detailed information on the options for each of these operating systems, see online help.

Using HLLAPI

HLLAPI (High-Level Language Application Program Interface), which is compatible with IBM EHLLAPI (except for a few unsupported functions) is available when you connect to a host. You can use HLLAPI to program in high-level languages such as COBOL, Pascal, BASIC, or C. Refer to TN3270 online help for more detailed HLLAPI information.

Additional Online Information

The online help file for TN3270 includes a description of each menu command, plus the following information in the "How to Use" section.

- Connecting to a Host
- Defining a Connection Profile
- Defining Key Mapping
- Defining a Button Pad
- Command Line Options
- Setting Up TN3270
- Setting Colors
- Setting Hotspots
- Transferring Files between a PC and a Host
- Using the Script Interface
- Using HLLAPI
- OfficeVision
- Tips on Using TN3270

Troubleshooting

If you experience difficulties using the TN3270 application, refer to the following items:

❑ Ensure that the Connect port is correct: it should be the default (23) in almost all cases.

- ❑ Choose the Diagnose Connection command from the Help menu to verify connectivity to any remote system to which you want access if a failure to connect has been reported.
- ❑ If the window display appears incorrect, verify that the extended data stream and extended attribute was specified correctly when you connected.
- ❑ Verify that you are using the correct model (commonly 2 or 3).
- ❑ If you cannot continue typing, press the RESET key.
- ❑ If you have disconnected from your application without logging off and you cannot log in again, use the reconnect option (in TSO) or ask your system administrator to cancel your login (in CMS).
- ❑ If you cannot use IND$FILE file transfer, make sure that the IBM host command is correct. To test it, type the command (IND$FILE, or another command that replaces it) from the ready prompt of TSO or from the running state of CMS. The host will respond with the version number of your IND$FILE program.

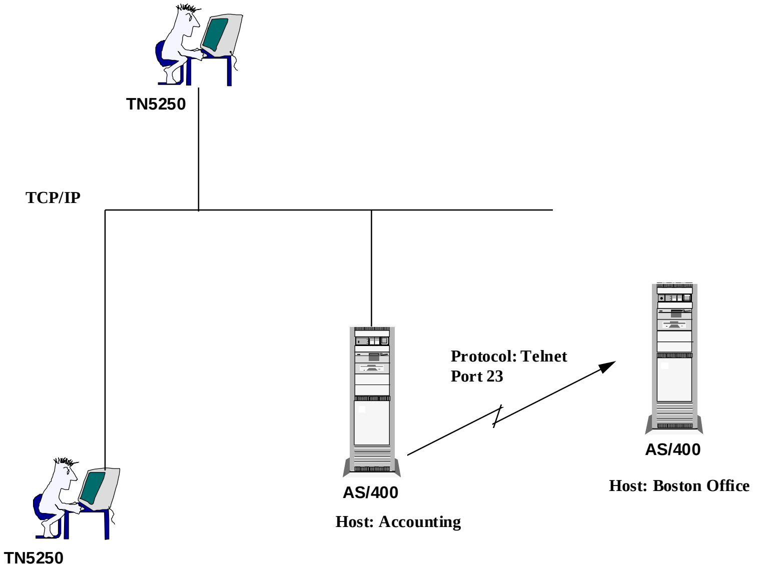
Note: In some early versions of IND$FILE, you will see a blank screen and the cursor will move to the lower right corner of the screen. If this occurs, press PF2 twice to return to normal operations.

- ❑ Make sure to start file transfer from the ready prompt of TSO or when CMS is in the running state.

Chapter 6. TN5250 Terminal Emulation

TN5250 is a terminal communications program connecting your PC and an IBM AS/400 host. This application offers 5250 protocol support for IBM hosts with IBM's TCP/IP via supported TCP/IP attachment devices.

Note: Support for the PCS APPC connection type is available with the NetManage HostLink product.



Features

The TN5250 application includes the following features:

- Monochrome terminal emulation
- IBM Office Vision commands
- International character sets are supported. The PC keyboard displayed in the Keyboard Remap window is based on the keyboard setting specified in the International Dialog of the Windows Control Panel.
- Other windows applications programmers can use the EHLLAPI interface for sending and receiving data with the IBM host using terminal emulation services.

- You may have multiple instances (up to the number of sockets) of the TN5250 application active concurrently.
- An animated icon signals whether the connection is active or inactive.

TN5250 can be invoked in several ways, as described in online help. TN5250 can be invoked either with arguments {TN5250 host [port] [options] .}or without arguments. A list of 5250 Terminal Emulation command line options is provided in the online help.

Note: To start TN5250 from a command line in Windows NT and Windows 3.1, you can use the Windows Program Manager/File Menu/Run... command. In Windows 95 use Start/Run.

The current preferences and host name can be saved to a configuration file (default is TN5250.CFG). The configuration file is loaded when the TN5250 application is started. You can have different configuration files for each session. The default configuration file the next time you open a connection will be the last configuration file that you saved.

Note: An animated icon of a telephone labeled "5250" indicates that a connection is established when the TN5250 application is minimized.

Starting 5250 Terminal Emulation

As with all the terminal emulator applications, there are several methods for starting the application. See online help for detailed information.

TN5250 Command Line Options

General Information on starting emulators from a command line and a list of common command line options is included in the online help.

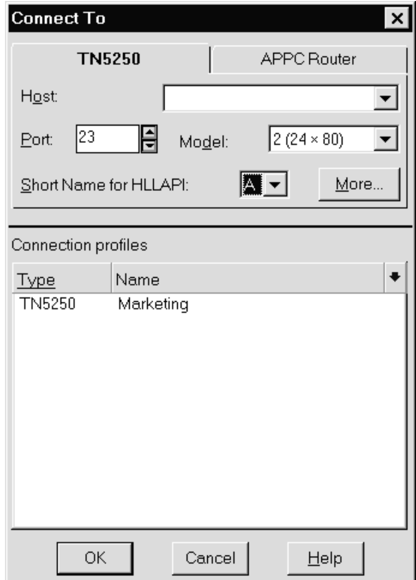
Connecting to a Host

To connect to a host system on the network, select the Connect command from the menu bar. If a connection profile has already been defined, you can select a name from the Connection Profiles list box.

Note: The Connect menu contains two tab dialogs: TN5250 (Telnet) and APPC Router. However, only the TN5250 tab dialog is enabled; APPC Router is available only with the HostLink product.

TN5250 Connect Type

You can use the TN5250 Connection Type to connect to any AS/400 IBM host that has a TCP/IP interface, connecting that host with the Internet. The TN5250 Session is performed over a TCP/IP stack as a full WinSock-compliant application.



If you are not using an existing connection profile from the list box, you can add the following information to complete the connection details:

<u>Connect Option</u>	**<u>Meaning</u>**
Host	Specifies the TCP/IP name or IP address .of the host that you want to connect to. If you have not specified a host in a Connection Profile, you must specify a host in the Connect dialog. There is no default value for this option. You can either type a name or choose one from the list. **Note:** You can specify a shortened version of the host name (such as "sunny") if the name is listed on the local Domain Name Server or Host Table. Otherwise, you must specify the IP Address or the full Internet host name.
Port	Specifies the TCP/IP port that your host uses for Telnet. In almost all hosts, the port number is 23. Use the default port (port 23) or select a different port.
Model	Defines the maximum screen size of the terminal emulation. You can choose one of the following screen sizes: • Model 2: 24 lines, 80 columns (default) • Model 5: 27 lines, 132 columns

Connect Option	Meaning
Short Name for HLLAPI	Specifies session name. Choose a session short name for HLLAPI or use the default value. The default session name is A for your first session, B for the second session, and so forth. If you want to choose a different session name, select a letter from A-Z from the pull-down menu to identify your session. The only valid session names are from A-Z, therefore you can only have 26 sessions at a time.

For information on additional settings, see "Advanced Options for TN5250".

Advanced Options for TN5250 Connections

Advanced options for TN5250 Connections are as follows:

Advanced Option	Meaning
Autoreconnect	Reconnects the 5250 session if you are disconnected by the host.
Exit on Disconnect	Closes the 5250 Terminal application when you disconnect the session.
Monochrome	Enables the AS/400 monochrome terminal emulation.
Title	This is the name that appears in the title bar of the terminal emulation window when you are connected to the specified terminal emulation session. If you do not specify a title name, the host name appears in the title bar by default. (You can specify a Title in the Connect to or Connection Parameters dialogs, but not in the Connection Profile dialog.)

Defining a Connection Profile

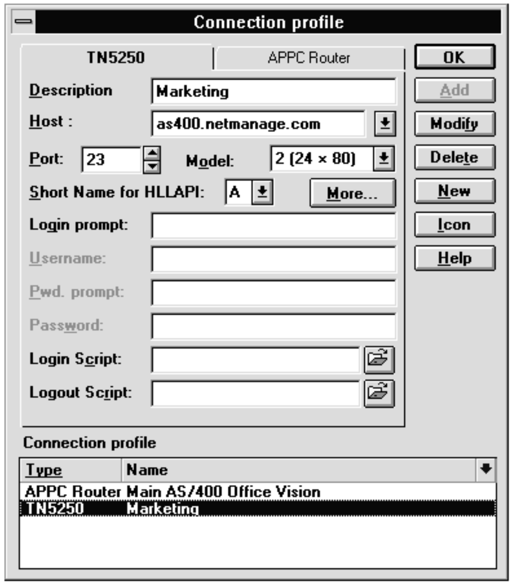
A connection profile allows you to set and store a list of frequently used connections. The connection profile names will be listed in the Connect dialog box, and allow you to connect automatically to a host. To create a connection profile, you must define a description and host name (or IP address). You may also specify additional options, as desired (see "TN5250 Connection Options"). For general information on creating, modifying, and deleting connection profiles, see online help.

Connection Profile

To create a Connection Profile, choose the Connection Profile command from the Settings menu. A connection profile allows you to set and store a list of frequently used connections. The connection profiles names will be listed in the Connect dialog box, and allow you to connect automatically to a host.

TN5250 Connection Profile

To create a Connection Profile for the TN5250 Connect Type, choose the Connection Profile command from the Settings menu and select the TN5250 tab.



The TN5250 Connection Profile dialog box contains the following fields:

Connection Profile Field	Meaning
Description	Name of the connection that appears in the list box in the Connect dialog box. Must be specified.
Host	Name or IP address of the host to which you are connecting. Must be specified.
Port	The default (23) Telnet communications port.
Model	Specifies maximum rows and columns on the screen. Relevant values are: model 2 (24 x 80) model 5 (27 x 132)

Connection Profile Field	Meaning
Session Short Name for HLLAPI	(High-Level Language Application Program Interface) enables other applications to communicate with your terminal emulation session. Each session should be defined with a different name. When you create the first session, the default name is A. The default name for the second session is B, and so forth. You can choose any letter from the list (A through Z).
Monochrome Terminal	Select if you want monochrome terminal emulation.
Login Prompt	The host login prompt that you would see on your screen when you login. Optional. Use for automatic login. Case insensitive; accepts partial matching.
Username	The username that you use to log in. Optional. Use for automatic login.
Pwd Prompt	The password prompt that you would see on your screen. Optional. Use for automatic login. Case insensitive; accepts partial matching.
Password	The password that you normally enter after you have entered your username. Optional component of automatic login.
Login Script	This is optional. You can type or browse to select the name of a script file that automatically logs into the host when you connect to a host by using this profile.
Logout Script	This is optional. You can type or browse to select the name of an existing script file that automatically logs out of the host specified by this profile when you close the application.
Connection Profile List	This list box contains the connection profiles that you have defined.

Note: If you want to include the Login Prompt and the Password Prompt as part of your Connection Profile, you should connect and log in to the host and write down the prompts that are displayed for username and password. The specified text string must match the prompts (or a portion of the prompts) as they are displayed.

You may include additional options in your Connection Profile by choosing the More button. See "Advanced Settings for TN5250" for details.

Capturing Files

TN5250 allows you to capture a session to a printer or a file. The Capture command is in the File menu. The parameters for the capture may be set by using the Capture command from the Settings menu. See online help for a description of these settings.

Scripts

A script is a file of commands that automate one or more procedures that can be done with the terminal emulation application. You can choose the language for recording the script and the run mode for playing it. The recording language can be: VBA or Simplified. The run mode can be: VBA or Cham4.5.

- **Simplified**: Provides a limited set of commands that can be used to create a script for automated login. This type of script can be run from Telnet, TN3270 and TN5250 Terminal Emulation applications, either by specifying the script name in the connection profile or by running the script after a connection has been made. If you edit a script using the Simplified Script Type, you must run the script as Cham 4.5.
- **VBA**: Refers to NetManage Visual Basic for Applications. This Script Type is compatible with the standard Microsoft VBA. and supports Visual Basic™ syntax. It also supports OLE automation and function calls for external DLLs. If you edit a script using VBA, you must also run it using VBA.

For additional information, see the Visual Script Editor/Visual Script Player chapter of this manual and the online help.

Using the Edit Menu

The Edit menu provides copy and paste functions for text and graphics. All text pasted to the emulation window is transmitted directly to the host system. For additional information, see online help.

Using the Settings Menu

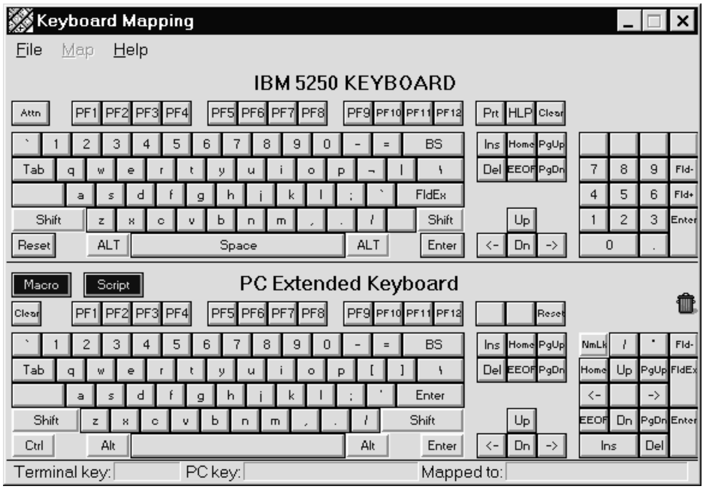
The Settings menu allows you to specify a variety of options, including various terminal preferences, such as cursor type; settings options for file capture; and more. These settings are saved to a TN5250 configuration file, if you select the option for saving your configuration file. Refer to TN5250 online help for detailed information about using the Settings menu.

Logging a Session

The Settings menu includes the Log command, which allows you to debug programs, report problems, and record demos. See online help for a list of steps in logging a session.

Mapping Your Keyboard

You can remap and view your keyboard by selecting the Keyboard command from the Settings menu. A display of a PC extended keyboard and an IBM 5250 keyboard appears. Note that the PC keyboard displayed in the Keyboard Remap window is based on the keyboard setting specified in the International dialog of the Windows Control Panel.



Setting Hotspots

Hotspots allows you to enable word selection and define PF templates. After a hotspot has been defined, you can activate the hotspot during a session by selecting a hotspot (using the right mouse button) displayed on the screen. Selecting a predefined hotspot invokes its associated function-key call or terminal command. A hotspot can also be used to invoke a Visual Script. Refer to TN5250 online help for detailed information about setting Hotspots.

Using HLLAPI

HLLAPI (High-Level Language Application Program Interface), which is compatible with IBM EHLLAPI (except for a few unsupported functions) is available when you connect to a host. You can use HLLAPI to program in high-level languages such as COBOL, Pascal, BASIC, or C. Refer to TN5250 online help for more detailed HLLAPI information.

Additional Online Information

The online help file for TN5250 includes a description of each menu command, plus the following information in the "How to Use" section:

- Connecting to a Host
- Defining a Connection Profile
- Defining Keyboard Mapping
- Defining A Button Pad
- Command Line Options
- Setting Preference
- Defining Keyboard Mapping
- Screen Size and Automatic Font Adjustment
- Setting Colors
- Setting Hotspots
- Using the Script Interface
- Using HLLAPI

Troubleshooting

If you experience difficulties using the TN5250 application, refer to the following items:

❑ Ensure that the Connect port is correct: it should be the default (23) in almost all cases.

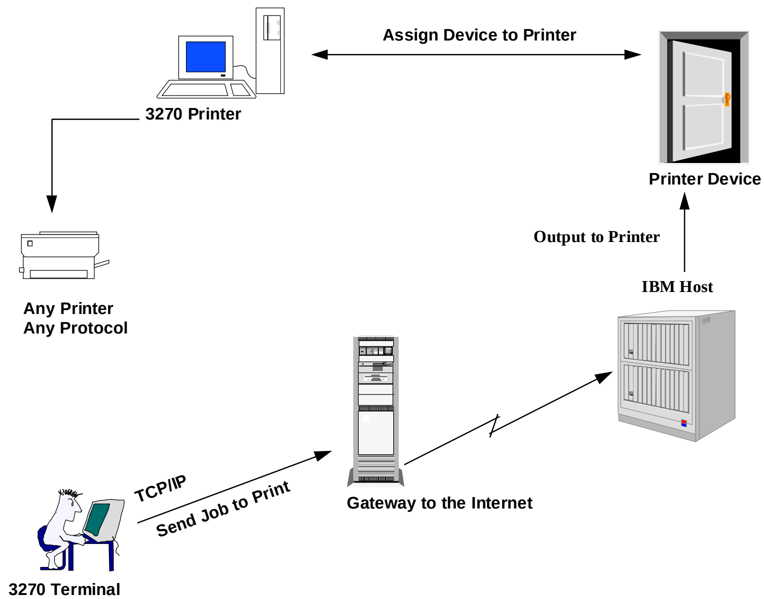
❑ Use the Diagnose Connection command from the Help menu to verify connectivity to any remote system to which you want access if a failure to connect is reported.

❑ If you cannot continue typing (for example, the keyboard is locked), press the RESET key or enable the Auto-Reset option in the Settings/Preference menu.

Chapter 7. 3270 Printer Emulator

The 3270 Printer application allows you to print IBM S/370 and S/390 sessions to various printers. While the 3270 Printer application makes the printer appear to the IBM host as if it were an IBM system printer, it also processes and translates the print jobs received from the host into a protocol that can be handled by a regular Windows printer.

Note: If you are using a 3270 terminal (either real or emulated), you must print via a real IBM system printer or an emulated IBM system printer.



The 3270 Printer application can be used in any of the following ways:

- By any user of a real or emulated 3270 terminal connected to the system host
- For printing to a local attached printer, a network printer, or a printer connected to the host by coaxial cable
- For printing a session from the TN3270 application

  Note: You must choose the "Use TN3270E protocol" option within TN3270 for that terminal emulation session if you want to print using the 3270 Printer application.

Printing with the 3270 Printer Emulator

To print using the 3270 Printer application, you should use the following procedure:

1. Start the printer application.
2. Select connection settings for the session to which you wish to connect.
3. Establish a connection to the host.
4. Minimize the associated printer application. (This is optional. You can leave the application in its non-iconized form, if you prefer.)
5. Select any additional print options and start the print job from within your terminal session.

Note: The 3270 Printer application can be invoked in several ways, as described in online help. 3270 Printer can be invoked either with arguments {PR3270 host [port] [options] }or without arguments. A list of command line options is provided in the online help.

Supported Printer Configurations

A print job processed through the 3270 Printer application can be directed to:

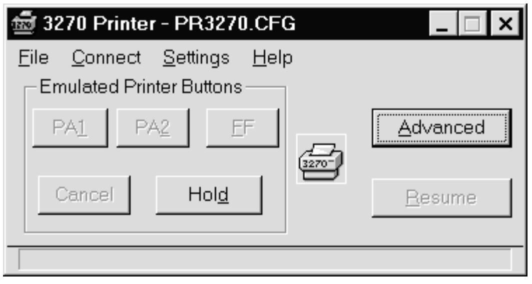
- A designated printer attached to the PC workstation
- A print file
- A network printer

Note: A network printer can be connected using any valid type of connection (such as, a Novell printer, TCP/IP LPR/LPD printer, and so forth.)

The printer settings options available within the 3270 Printer application depend upon the printer settings specified in the Windows Control Panel.

Connecting to a Session

When you first open the 3270 Printer application, the Basic dialog box is displayed.



To print a terminal session, you must connect the printer emulator to the host. You can connect the printer emulator to the host in any of the following ways:

- Non-preconfigured Connection.
- Preconfigured connection in a connection profile list
- Customized icon
- Connection via the Session Manager application

The Connect and Connection Profile dialog boxes contain tab dialogs for various connection types. However, only the TN3270 tab is accessible for this product. If you want to use other connection types, you must use our HostLink product.

Non-Preconfigured Connection

A non-preconfigured connection allows you to connect the printer emulator to a single session. When the session is disconnected, the settings are lost. The next time that you want to connect to the host, you must re-enter the connection details.

To connect to a host via the printer emulator application dialog box, do the following:

1. Choose the Connect menu command from the basic 3270 Printer application dialog box.
2. A specific connection-type Connect To dialog box will be displayed and you can enter the necessary connection details.
3. When you have specified the desired connection details, choose the OK button.

Preconfigured Connection in a Connection Profile List

A preconfigured connection profile appears in the list box of the Connect To dialog box. When you choose the profile name, the application connects to the session described by that profile.

To create a Connection Profile, choose the Connection Profile command from the Settings menu, and specify the connection details according to the desired connection type.

Customized Icon

If you wish, you can save a connection profile as an icon. The customized icon appears in a window on your desktop, allowing you to open the printer session from there.

Connection via Session Manager

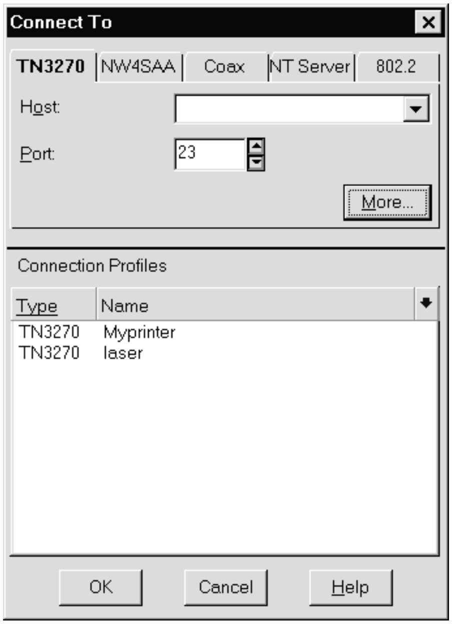
You may also connect to a Printer Emulator session by using the Session Manager. See the Session Manager chapter or online help for additional information.

TCP/IP Connection Type Options

If you wish to print a TN3270 terminal session, either choose the Connect menu item or choose the Connection Profile... command from the Settings menu. Then specify the following connection details in the dialog box that appears:

Host Name: Specify the host name or the IP address of the terminal session.

Port: The default port is the standard Telnet port, 23.



More/Advanced Settings

If you do not specify any particular setting in the Advanced Settings dialog, and select TN3270E protocol, the 3270 Printer application will request the TN3270E server to assign any available device name from its generic pool of printer devices. The TN3270E server also maintains separate pools of printer devices that can only be used to satisfy specific terminal session requests. Specify desired options, as follows:

- **Use TN3270E Protocol**: Printing with TN3270 protocol can be done using one of 2 standards: TN3287 (as defined in RFC 1646) and TN3270E (as defined in RFC 1647). You should know if the Telnet server supports these extensions to the standard TN3270 protocol. If the TN3270E option is selected, 3270 Printer uses TN3270E for the printing protocol. If the option is not selected, it uses protocol TN3287.
- **Device Name**: The Device Name command requests that the TN3270E server assign the specified device name from the pool of devices reserved for use with the

Device Name command. The Device Name command may be used when requesting either a terminal or printer session. If the specified device name is not available, the server denies the request.

- **Associate**: The Associate command requests that TN3270E server assign the specified device name from the pool of devices reserved for use with the Associate command. These are printer device names, which are paired with the specific terminal device names. If the specified device name is not available, the server denies the request.

Note: To request a specific device name, you must specify the name in either the Device Name text box or the Associate text box. You cannot specify device names in both text boxes.

Controlling the Print Session

The basic dialog box includes a section of emulated printer buttons. These buttons represent buttons that appear on actual IBM system printers. The printer buttons provide the following functions:

- **Hold**: If you choose the Hold button, it stops the host from sending more data to the printer. This provides you with the ability to manually halt additional input from the host: for example, to add paper to the printer or to correct a paper jam. In cases where the print queue is too long, the host will generally detect the problem itself, and stop sending additional data until the print queue has begun to clear. When the printer is in a hold state, the Hold button is replaced with a Release button. Choose the Release button, when you are ready to have the print job continue.
- **Cancel**: This button becomes available only when a print job is being processed. It allows you to cancel (stop) the current print job.
- **PA1** and **PA2** (**Program Access**): The use of these buttons depends upon the host application.
- **FF** (**Form Feed**): In order to use this button, you must first choose the Hold button. FF directs the printer to start printing at the beginning of a new page. The printer will resume printing at the beginning of a new page when you release the printer by reselecting the Hold button.
- **Resume**: This button allows you to resume a print job. If you are using an LU1 printer, you must choose this button to resume printing after having used the Cancel button. If you are using an LU3 printer, the host can resume printing itself, whether or not you choose the Resume button.

Checking the Print Session

You can check the status of your print sessions by choosing the Advanced button from the basic dialog box. This buttons displays another dialog box, which displays the real time status of the host, the PC, and the printer.

Using the File Menu

The 3270 Printer application has the standard HostLink application File menu options (New Configuration, Open Configuration, Save Configuration, and Save Configuration As), relating to the printer emulator's configuration file. The Printer Emulator's configuration file saves settings such as connection profiles and preferences that you have specified during a printer emulator session. The Exit command from the File menu closes the printer emulator application.

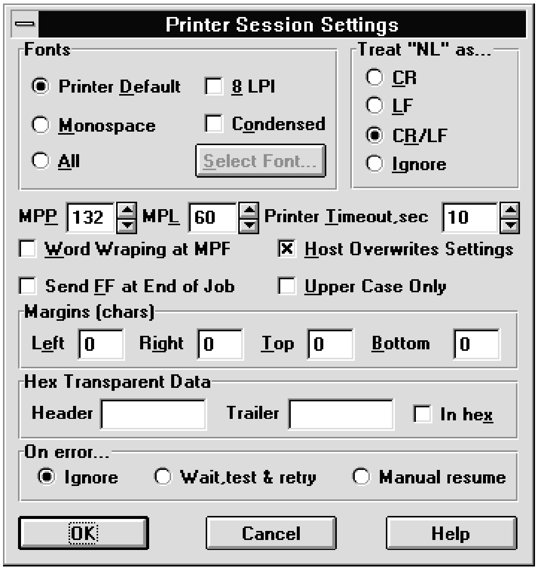
Using the Settings Menu

The 3270 Printer application allows you to select the following settings:

Preferences

The Preferences menu options displays a dialog box that allows you to select detailed options (such a fonts, use of carriage returns and line feeds, etc.) regarding the format of the printer output.

Note: You cannot change Preferences during a print session. Also, some options cannot be selected concurrently with other options. In that case, options that cannot be selected are grayed out.



Fonts: Allows you to specify details regarding the printer font.

- **Printer default font**: You can use this option only if your printer supports IBM/EPSON printer escape sequences. Some dot matrix printers support this standard. Most laser printers do not support this standard.
- **8LPI**: Prints 8 lines per inch (smaller font). This option is available only when the Printer default font is selected. Otherwise, it is replaced by the Select font button.
- **Monospace**: Uses fonts, where each character has the same width. If this option is not selected, the printer uses proportional fonts (if the printer, itself, supports these fonts).
- **Condensed**: Condenses the space between the characters. This option is available only when the Printer default font is selected. Otherwise, it is replaced by the Select font button.
- **All**: Uses both proportional and monospace fonts.
- **Select Font**: Displays a font dialog box, allowing you to select the desired print font from a list of possible fonts.

Treat "NL" as: Allows you to specify how the printer behaves when a new line (NL) escape sequence is received from the host.

- **CR**: Prints from the beginning of a line.
- **LF**: Advances the paper one line.
- **CR/LF**: Prints from the beginning of a line and advances the paper one line
- **Ignore**: Disregards the NL escape code. Prints as many characters on the line as can be fit into the line using the specified font and page setup.

MPP: Specifies "Maximum Presentation Position." Indicates to the maximum number of characters that can be printed on a line. The default is 132 characters.

MPL: Specifies "Maximum Presentation Lines." Indicates the maximum number of lines that can be printed on a page. The default is 60 lines.

Word Wrapping at MPP: When this option is selected, the printer moves a word to the next line if there is not sufficient space to fit the word in the current line.

Send FF at end of job: Advances the printer one page length or to the top of the next page when the print job is completed.

Host overwrites settings: This option is set by default. It indicates that the print settings specified on the host will override any settings that have been specified in the Printer Emulator application.

Upper Case only: Prints all characters in upper case.

Margins: Allows you to specify margins of the printed document

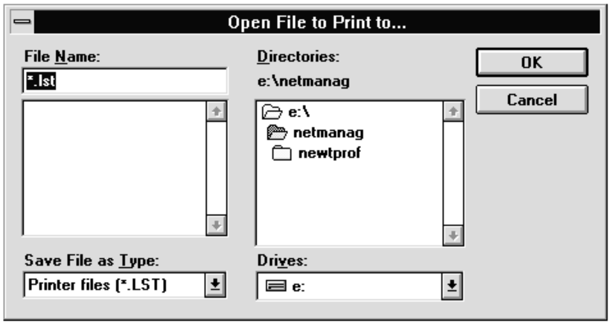
Hex Transparent Data: Allows you to specify a header and trailer (of printer escape sequences) that can be passed transparently by the Printer Emulator to the printer. The header specify the beginning of the transparent data and the trailer specifies the end of the transparent data. The Hex Transparent Data option can be used if you want to preserve control sequences (from another application or device) that determine the printing format. This option is available only when the printer default font is selected.

In hex: Distinguishes the header and trailer from regular data by embedding an escape character (a character that would never occur in normal data) in these fields. To do this, highlight one or more of the characters in the header or trailer fields and select the **In hex** option..

On Error: Specifies options for error recovery in cases where the printer issues its own error messages (such as, out of paper, paper jam).

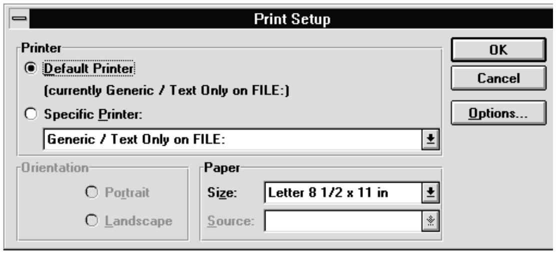
Print to File

Allows you to redirect output from the real Windows printer to a file. You can either type the filename or choose the Browse button. The file will be produced by the printer's driver rather than by the printer application; therefore, to use this option, you must have specified the Print to File option in your Windows Control Panel. This option is available only when the printer session is disconnected or on hold.



Printer Setup

Allows you to change the printer setup options that have been specified in the Windows Control Panel. In this example, the default printer has been specified in the Windows Control Panel as Printing to a File. If you want to change the printer setup for this session, you can select a different printer by choosing the Specific Printer option. This option is available only when the printer session is disconnected or on hold.



Additional Online Information

The online help file for PR3270 includes a description of each menu command, plus the following information in the "How to Use" section:

- Using 3270 Printer
- Starting 3270 Printer
- Command Line Options
- Printing with the 3270 Printer
- Connecting to a Host
- Defining a Connection Profile

Troubleshooting

If you experience difficulties using the 3270 Printer application, refer to the following items:

❑ Ensure that the Connect information is correct.

❑ Ensure that you have connected a virtual printer in the Windows Control Panel..

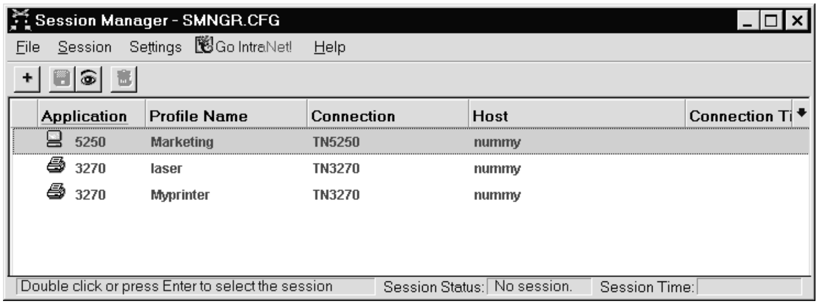
Chapter 8. Session Manager

Session Manager is an application that allows you to manage multiple terminal emulation and printer emulation sessions from within a single program.

With Session Manager you can perform the following functions:

- Open the terminal emulator and printer emulator applications
- View existing terminal sessions
- Disconnect and exit from a selected emulation session
- Show status of existing emulation sessions
- Invoke an existing connection profile

You can open, close or view multiple sessions at the same time.



Viewing Session Status

The Session Manager window displays the status of terminal and printer emulation sessions and the names of any existing connection profiles created within the emulator applications.

The Session Manager window includes the following columns:

Icons: The icon column can contain one or more icons.

A terminal icon identifies list box entries for terminal emulation sessions.

A printer icon identifies list box entries for printer emulation sessions.

A lightbulb icon identifies each emulation session previously saved to the Session Manager configuration file or currently connected. When a session is connected, the lightbulb icon is illuminated. A profile icon represents existing emulator connection profiles.

Application: The name of the emulator application is displayed for currently connected or previously saved emulator sessions. For 3270 and 5250 emulators, this column also includes the HLLAPI session name.

Profile Name: A connection profile name is listed for existing connection profiles. Connection profiles are not listed in the Session Manager window if they are created while the Session Manager is running: they will be listed when the Session Manager is closed and restarted.

Note: You cannot create a connection profile within the Session Manager application. You create connection profiles within the emulator applications. Session Manager lets you view and activate existing connection profiles from the various emulator applications.

Connection: Indicates the connection type (such as VT100) being used for a communications session.

Host: Indicates the name of the remote host for the terminal emulation session.

Connect Time: When there is a connection, this column contains a timestamp, indicating the start time of a currently connected session. The start time is indicated in the format of hours:minutes:seconds. This column also indicates the amount of elapsed time (within parentheses, in minutes or hours:minutes) since the beginning of the session. The elapsed time is not indicated until at least one minute has elapsed.

Additional Online Information

The online help file for Session Manager includes a description of each menu command.

Troubleshooting

If you experience difficulties using the Session Manager application, refer to the following items:

- ❏ Ensure that the terminal emulator's Connect port is correct: it should be the default (23) in almost all cases.
- ❏ Use the Diagnose connection item from the Help menu to verify connectivity to any remote system to which you want access if a failure to connect has been reported.

Chapter 9. Visual Script Editor/Visual Script Player

Visual Script Editor is an editor used to write or modify scripts. The editor is user-friendly, helps you to learn the script language, and assists you in building the commands.

What is a Script?

A script is a sequence of ASCII text lines stored in a file. You can create scripts to automate a variety of tasks, such as:

- Log in to a host
- Perform file operations
- Log data communications to a file
- Synchronize transmissions between host and local computers
- Print, copy and delete files.

The Script utilities allow you to dynamically reconfigure the user interface, perform random access file operations, automate tasks, modify current functionality, and handle incoming events. For example, you could write a script to customize an end-user interface that would allow non-technical users to easily access various programs on a variety of hosts.

Note: One of the easiest and most effective ways to familiarize yourself with Visual Script Editor/Player capabilities is to run the demo scripts provided with this application. Run one of the script files (such as, demo.vba or demo1.vba) to see how a script works. You can copy a sample script file and edit it to suit your needs.

Components of the Script Utility

There are three main components of the script utility:

- Visual Script Editor
- Script language
- Visual Script Player (VSP and VBAVSP)

The Visual Script Editor is used to create or modify script files. The Visual Script Editor application allows you to use any of two Script Types for editing your script:

- **Simplified**: Provides a limited set of commands that can be used to create a script for automated login. This type of script can be run from Telnet, TN3270 and TN5250 Terminal Emulation applications, either by specifying the script name in the connection profile or by running the script after a connection has been made.
- **VBA**: Refers to NetManage Visual Basic for Applications. This Script Type is compatible with the standard Microsoft VBA, and supports Visual Basic™ syntax. It also supports OLE automation and function calls for external DLLs.

The Script language consists of the sequence of commands that make up the script. The components and rules of the Script language depend on the selected Script Type. For details on language components and rules, see the online help.

Note: If you have scripts that have been created using Cham 4.5, it is recommended that they be converted to VBA scripts. VSP, which must be used to execute Cham 4.5 scripts, is still provided for backward compatibility. However, support for Cham 4.5 script types is limited.

The Visual Script Player executes the commands specified in the script file. There are two Visual Script Players:

- **VSP** runs scripts that have been created using the Cham 4.5 or Simplified Script Type.
- **VBAVSP** runs scripts that have been created using either the Cham 4.5 or VBA Script Type. Using this script player you can test your scripts by setting breakpoints, single stepping through the script, and watching as the variables change.

Note: You can invoke either one of the Visual Script Player applications from within Visual Script Editor by choosing the Run command. The choice of VSP application to be invoked depends upon which option is currently selected for the Run As option under Settings/Script Type.

Help

The Visual Script Editor online help (represented by the question mark icon) provides information on the Visual Script Editor components and on using the menus.

Assist

The Assist menu option (represented by the question mark with a pen icon) invokes a dialog box that contains explanations of the Script commands and actually assists you to write a script. The contents of the Assist changes depending on which Script Type mode you have currently selected.

The Assist dialog contains the following parts:

Command Class: This pop-down box in the upper right corner of the dialog lets you filter the selection of commands listed in the Script Commands list box. The contents of this list changes depending on which Script Type mode you have currently selected. If you have selected Simplified Mode, the only Command class available is Simplified. If you are using standard VBA, you can select to view All the commands or one of the listed categories (such as File, Dialog, Emulation).

Script Commands: This scrolling list box provides an alphabetical list of the commands available for the current Script Type in the selected Command Class. The default list shows all the commands available for the Script Type currently being used.

Command Description: This block contains the syntax, the definition, and an example of the command you have highlighted in the alphabetical list of commands. The Assist Screen contains a Print button that allows you to Print the current Command Description, current class commands, or all commands, so that you can have a hard-copy for reference.

Current Command: This field contains the script command line that you have highlighted.. When you select or enter command parameters for that command, the specified parameters are added to the Current Command line.

To build a script using the Assist menu, do the following:

1. Highlight a command in the Script Commands list box. The command description is displayed in the Command Description box.
2. Choose the Select button. The selected command is displayed in the Current Command line, and text boxes appear prompting you to input any parameters that are part of that command. Open the drop-down menu to see a list of valid parameter types.
3. Specify the requested information for the parameters in the appropriate text boxes. The parameters you specify appear in the Current Command line. If you fail to specify a required parameter, an error message will appear prompting you to specify that parameter.
4. You can move within the script or edit it as you work. To add a new line choose the New Line button. To move up or down within the script use the up or down arrow button. Repeat these steps for any other commands on the Assist list to continue building your script.
5. Choose the Close button when you have completed building the script. Your selected command and selected command parameters now appear in your script template.

Additional Online Information

The online help file includes a description of each menu command, plus further information in the "How to Use" section.

- VBA Script Type Language Components
- Editing a Script
- OLE Automation in VBA
- Using the OLE Browser

Troubleshooting

If you experience difficulties using this application, refer to the following items:

- ❑ Ensure that the terminal emulation can be invoked manually.
- ❑ Ensure that terminal emulation short names are correct.
- ❑ Ensure that the syntax in your commands matches the syntax of samples in the Assist Command Description box.

Chapter 10. FTP - File Transfer

The FTP application provides file transfer services between remote computers and your local computer through the use of the File Transfer Protocol (FTP). FTP displays both the local and remote directory structures in the same way local files and directories are displayed in Windows. You can browse directories as well as select multiple files for transfer in either direction.

NetManage also provides an FTP Server application, which permits other FTP clients access to selected local files and directories. Server access can be restricted for security, and client and server may run concurrently.

FTP

FTP lets you establish a connection to a remote system and then create, remove, rename, or transfer files and directories to and from that system. For ease of use, you can define connection profiles that can be used to connect to frequently used FTP servers. Discussions on establishing connections, transferring files, and creating connection profiles follow.

Establishing Connections

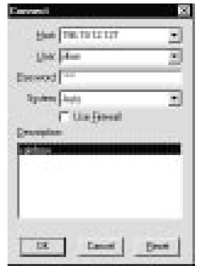
Establish an FTP connection to a remote system if you want to create, remove, rename, or transfer files and directories to and from that system.

To connect to a remote system, do the following:

1. Start FTP.

   The FTP window appears. After you establish a connection, you can use this window to create, remove, rename, or transfer files or directories to and from a remote system as discussed in the Transferring Files or Directories section in this chapter.

2. Choose the Connect menu option to open a Connect dialog box similar to the following:



3. Enter the host name or IP address, username, and password by doing one of the following:

   - Enter the information in the appropriate fields.
   - If you previously defined a connection profile for the server to which you want to connect, select that profile in the Description box. This automatically enters the host, user, and password information defined in the profile.

   If a password is not specified, FTP prompts you for this information when you connect. For systems to which you do not have specific login rights, select anonymous from the User drop-down menu and then use your e-mail address as the password. You can set up FTP to automatically use your e-mail address for anonymous as discussed in the Setting Preferences for FTP Client.

4. Select the remote system type from those listed in the System field.

   In most cases, FTP automatically determines the remote system type. If during this connection, FTP reports that it could not determine the system type, you may need to specify the exact system type.

   If you do not know the system type to which you are connecting, select Other. Selecting Other allows you to use the Info buttons to view the files and directories in the remote system list boxes. Then you must type the directories and filenames when you transfer or change directories. You may also clear the Retrieve detailed file listing option in the Preferences dialog box to get all the directories and filenames in raw (NLST) format.

5. If the connection is established through a firewall system, ensure that the Use Firewall check box is selected. Otherwise, go to the next step.

   A firewall is an intermediate computer that provides security to a network, such as the Internet. As discussed in the Defining a Firewall section, you can define a default firewall that more than one connection can use and you can define firewalls for specific connections.

   Note: If you select the Use Firewall check box and there is no firewall information available for the connection, the Firewall dialog box appears. For details on entering information in this dialog box, see the table included in the Defining a Firewall section.

To monitor a connection to a firewall, choose the Log... command from the Settings menu. If the Start button is available, choose it to start the Log. It will then change to the Stop button.

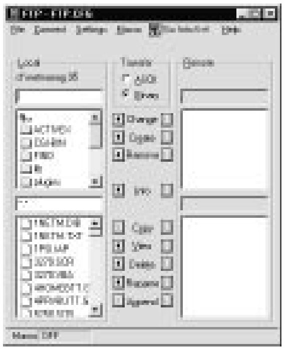
6. Choose the OK button on the Connect dialog box to connect to the designated system and return to the FTP window.

   The host or IP address of the connected system is displayed in the caption of the FTP window, or in the icon title if FTP is minimized. The greeting message from the remote system is displayed along the top of the window. You can use the FTP window to transfer files as discussed in this chapter.

Note: Once you establish a connection, you can store the connection information as a connection profile. The connection profile feature is particularly useful for connections that you need to frequently establish.

Transferring Files or Directories

The FTP window lets you transfer files and directories as well as create, remove, and rename files and directories. This window appears when you start FTP or establish a connection. In this window, folders represent directories, and folders containing an arrow represent symbolic links. FTP treats symbolic links as directories.



To transfer files or directories, do the following:

1. Specify the file or directory you want to transfer and the directory or file to which you want to transfer.

   You can either type the names in the Directory and Files fields or choose the names from the window's list boxes. Local files and directories are specified on the left side of the window, and remote files and directories are specified on the right side.

When specifying file and directory names, you can:

- change to another directory by either double-clicking the directory name or by typing the name in the local or remote Directory field and then choosing the Change button.
- create a file or directory by typing its name and then choosing Create.
- remove or rename a file or directory by selecting it and then choosing the appropriate button (Remove or Rename).
- display additional details about a directory by selecting it and then choosing the Info button.

2. Specify a file transfer mode by selecting either the ASCII or Binary option. If you are unsure as to which mode to use, select Binary.

   All transfers are executed in either ASCII (text) or Binary (image) mode. Transferring a file in the incorrect mode may corrupt that file. ASCII mode performs carriage return/line feed translation and is only needed when transferring text files for use on a non-Windows system. Unless you transfer text files to or from a UNIX FTP server, choose Binary.

3. Transfer the file or directory.

   The left Append, Copy, and View buttons signify append, copy, or view remote-to-local. The right buttons signify append, copy, or view local-to-remote.

   Note: You can also copy files or directories by dragging them to the appropriate list box.

   The View button lets you transfer a file and open it in the appropriate viewer with one click. When you select a file and choose a View button, FTP retrieves the file and gives it to the appropriate viewer application. If FTP reports that it is unable to view a file, associate a viewer with that file's three-letter extension.

   Selected files and directories are transferred with the same name, unless you type a new name in the local or remote edit box. Files or directories with illegal names will be translated and prefixed with the exclamation character (!) if a new name is not specified. For example, a UNIX file named init-logcode.c copied to the local system will be translated to the filename !INIT-LO.C.

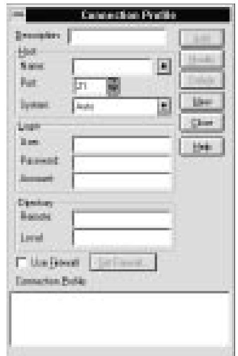
Creating Connection Profiles

Use connection profiles to set and store a list of frequent destinations. You can define a host, port number, user, password, system type, account, remote directory, local directory, and a description field for each destination. FTP saves profiles to a configuration file (FTP.CFG is the default). The configuration file is loaded when you start FTP.

Create a connection profile for a destination if you plan to use that destination frequently. To create a connection profile, do the following:

1. Choose the Connection Profile... command from the Settings menu.

   The Connection Profile dialog box appears.



2. Enter information in the Connection Profile dialog box fields, which are described in the following table.

Field	Description
Description	Name of the connection profile.
Name:	Name or IP address of the host to which you are connecting.
Port:	The number of the port to which you want to connect. The default is 21. The port setting should be 21 unless you need to connect to a specific FTP server that is available on a non-standard port.
System:	FTP server type that the client expects to see.
User:	Username that is to be entered when user logs in to FTP Server.
Password:	Password you want entered at the password prompt. This will be invisible to the user.
Account:	Optional account command (ACCT) to be sent when connecting.
Remote:	Default directory on the remote system. This directory is used when FTP Client establishes a connection to FTP Server.

Field	Description
Local:	Default directory on your system. This directory is used when FTP Client establishes a connection to FTP Server.

3. If this connection is to use a firewall, see the Defining a Firewall section. Otherwise, go to the next step.

   A firewall is an intermediate computer that provides security to a network, such as the Internet.

4. Choose the Add button to add the profile to the Connection Profile list box. Then choose the Close button.

Defining a Firewall

A firewall is an intermediate computer that provides security to a network, such as the Internet. For information on the effect of a firewall, see your online help.

You define a firewall to provide information about a firewall system through which a connection will be established. If more than one connection is to share the same firewall, define a default firewall. If you plan to use a firewall for a specific connection, define the firewall for that connection.

To define a firewall, do the following:

1. Do one of the following:
   - If more than one connection will be going through the same firewall, choose the Preferences... command from the Settings menu. Then select the Use Firewall by Default check box and choose the Set Default Firewall... button.
   - If the firewall is for a specific connection, choose the Connection Profile... command from the Settings menu. Enter information in the Connection Profile dialog box fields as discussed in the Creating Connection Profiles section. Then select the Use Firewall check box and choose the Set Firewall... button.

2. In the Firewall dialog box that appears, enter information in the fields that are described in the following table. Then choose the OK button.

   You must obtain information for the firewall, firewall user, and firewall password, and type of firewall connection from your network administrator.

   Note: For a more detailed description of the firewall options, see your online help.

Field	Description
Firewall:	Enter the IP address or the machine name of the firewall.
User:	Enter the username that lets you access the firewall computer.

Field	Description
Password:	Enter your password to the firewall computer. This is invisible to the user. **Note:** If the firewall password times out, see your network administrator. Some firewalls have passwords that automatically time out after a certain time period. You may have to use Telnet to change the password on the firewall.
Port:	The number of the port to which you want to connect. The default is 21. Some firewalls may use another port number.
Use SITE command	Select this option to use the SITE command to connect your local network to the firewall. The firewall, firewall user, and firewall password fields are required.
Use USER after logon	Select this option to use the USER command to connect your local network to the firewall. The firewall, firewall user, and firewall password fields are required.
Use USER with no logon	Select this option to use the USER command to connect your local network to the firewall. The firewall field is required, but the firewall user and firewall password fields are ignored.
Use proxy OPEN command	Select this option to use the OPEN command to connect your local network to the firewall. The firewall field is required, but the firewall user and firewall password fields are ignored.

3. If you opened the Firewall dialog box from the Connection Profile dialog box, finish adding or modifying the connection profile by choosing the appropriate button. Then choose the Close button.

FTP Server

FTP Server lets you give Internet users access to your PC files and directories. You can restrict access to an FTP server by defining users and assigning passwords as discussed in this chapter. You can also configure the server to define the server's time out value and specify how many users can be connected to the FTP server simultaneously as discussed in the online help.

Restricting Access to an FTP Server

You restrict access to an FTP server by defining a user list. This list specifies who is authorized to access the FTP server.

To define a user list for an FTP server, do the following:

1. Start FTP Server.

2. Choose the Users... menu option to open the Users dialog box.

3. Choose the Add button.

4. Enter the username and password.

   If the username is "anonymous", you do not need to specify a password. This is because the server will expect a password that is a valid Internet mail ID in the form user@company.com. The server will ignore other types of password entries.

5. Specify the user's access by doing one of the following:

   - Grant user's access to all of the server's local drives, directories, and files by selecting the All Drives option.
   - Restrict user's to a specific directory and its subdirectories by selecting the Restricted to option. If necessary, choose the Dir... button and browse until you locate the desired directory.

6. If you want the user's access to be read-only, select the Read Only check box.

   Read-only users can retrieve files. They cannot, however, send the retrieved files to the FTP server or create or rename directories or files.

   If you select the Read Only check box, the following operations are denied: put, append, delete, rmdir, mkdir, and rename.

7. Choose the OK button.

After you add a user to the user list, you can modify the information for that user or delete the user from the list. For details, see the FTP Server online help.

Additional Online Information

The online help file for FTP includes a description of each menu command, plus the following information in the "How to Use" section:

- Automated File Transfers
- Creating a Connection Profile
- Creating Macros
- Defining a Firewall
- Deleting a Connection Profile
- Establishing a Connection
- Modifying a Connection Profile

- Running FTP from the Command Line
- Setting Preferences
- Transferring Files or Directories

The online help file for FTP Server includes a description of each menu command, plus the following information in the "How to Use" section:

- Deleting Users from the User List
- Disconnecting Users
- Modifying the User List
- Restricting Access to an FTP Server
- Setting Preferences

Troubleshooting

If you experience difficulties using FTP, refer to the following items:

❑ Make sure the remote system provides an FTP server and that the server is running. Note that some operating systems (for example, VMS), do not include TCP/IP services with the standard package, and not all TCP/IP hosts run an FTP server.

❑ If you fail to connect, choose the Diagnose Connection option from the Help menu in FTP Client. This option invokes the Ping application, which can show you if the host is accessible.

❑ Use the Log command as discussed in your online help to aid you in diagnosing problems.

❑ Verify that the username and password were entered correctly if the FTP client reports a failure to login.

❑ If during connection, FTP reports that it failed to determine the system type, or if you connect successfully but are unable to see any files, you may need to specify the exact system type. If you are unable to determine the remote system type, you can select Other from the list.

  When you connect as Other, all the file and directory names in the Remote Files list are displayed. You may enter the appropriate name to transfer files or change the current directory. You can also clear the Retrieve Detailed File Listing option in FTP’s Preferences dialog box to retrieve all files and directories in raw (NLST) format.

❑ Make sure that the correct transfer type (ASCII or Binary) is chosen. Transferring a file in the incorrect mode may corrupt that file. Unless you transfer text files to or from a UNIX FTP server, choose Binary.

❑ Make sure the clients have the necessary permission for specific operations (for example, write access to a directory).

Chapter 11. LPR/LPD

The LPR and LPD applications allow files to be printed remotely over the network. The NetManage LPR application uses the Line Printer Remote (LPR) protocol and functions as the client side of the remote printing process; it allows you to print to network printers that are running LPD. The NetManage LPD application uses the Line Printer Daemon (LPD) protocol and functions as the server side of the remote printer process; it makes your local printer accessible as a network printer for remote users that are sending files with LPR.

Using LPR

LPR allows you to print a file to a remote LPD printer.

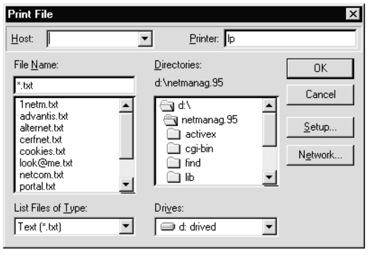
LPR will continuously retry to send your file to the printer. You can view the status of your job by opening the Log file and checking the retry status of the job. Depending on the status, you may choose to remove your job from the LPR queue by highlighting the job listing in the LPR window and choosing the Delete Job command from the LPR menu.

Printing a File Using LPR

When you print a file using LPR, the file does not go through the Windows printer driver defined in the Control Panel's Printer dialog box; instead, it is redirected to a remote printer as specified in LPR.

To print a file to a remote network printer, do the following:

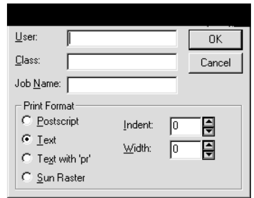
1. Start LPR and choose Print File... command from the LPR menu. The Print File dialog box appears.



2. Enter the remote host name (or IP address) and the remote printer name. Then select your file.

3. If you are printing to any standard LPD printer, you are done. For other implementations of LPD, however, do the following:

   a) Choose the Setup... button to open the LPR Setup dialog box.



   b) Enter the desired information in the LPR Setup dialog box:

      - User Name: your username.
      - Class: the class name, if your print server supports this feature for prioritizing print requests (optional).
      - Job Name: \your job name, which is printed on a banner page if your printer supports this feature (optional).
      - Print Format: the file’s print format. Choose one of the following formats: PostScript, text (raw, unformatted text file), text with pr (unformatted text file with a heading, page number, and pagination), or Sun Raster.

      Note: The remote printer must support the designated format. If you specify PostScript and the printer cannot interpret the PostScript commands, the file will not print. If this occurs, try using the text option.

   c) Choose the OK button to send your file to the printer.

You can print to a file and then send that file to a remote printer. To do so, do the following:

1. In the Control Panel, choose Printers, and then select your default printer.

2. Direct your default printer to print to a file by doing one of the following:

   - If you are using Windows 3.1, choose the Connect button. Scroll down the list of ports until you see File and choose the OK button. You may be prompted for a filename.
   - If you are using Windows 95, choose the Properties... command from the File menu. Then select the Details tab. In the Print to the following port option, select File and choose the OK button.

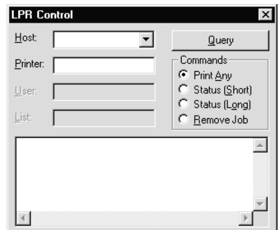
3. Choose the Print File... command from the LPR menu. Then use the Print File dialog box to select the file and send it to the remote printer.

Controlling Print Jobs

The Control menu allows you to perform the following commands: Print Any, Status (Short), Status (Long), and Remove Job. You can use these commands to check if your print job has already printed or is still in the queue. You can also remove print jobs from the queue.

To check the status or control the print job:

1. Choose the Control menu option.



2. Enter the relevant information into the fields of the LPR Control dialog box, as follows:

   - Host: The name of the remote host (must be specified to select any of the commands).
   - Printer: The name of the remote printer (must be specified if you have selected any of the commands).
   - User: Your username (must be specified if you have selected the Remove Job command).
   - List: If you do not enter any print job names in this field and you select Remove Job, LPR will remove all of your print jobs from the queue. If you want to select which of your print jobs will be removed from the queue, you can specify the job names in this field. If you enter one or more job names, only those print jobs will be removed. (You must own those print jobs).

     If you selected the status command (short or long) and you do not enter a print job name in the list field, LPR will display the status for all print jobs for the specified printer. If you enter your username, status for all of your print jobs will be listed. If you enter one or more job numbers, status for that job or jobs will be listed.

   - Query commands are as follows:
     - Print Any: Print all jobs.
     - Status (Short): Display a summarized version of your print job status.
     - Status (Long): Display a more detailed version of your print job status.
     - Remove Job: Delete a print job from the printer queue.

Note: These options are independent of LPD.

LPR Printing from within an Application

In Windows 95, you can configure your system so that you will be able to use LPR to print while working in another application. Before you can use LPR to print from within another application in the Windows 95 environment, you must first start the LPR service from the network and then add a printer. Separate discussions on each task follow.

In the Windows 3.1 environment, you cannot print to a remote printer from within an application. If you want to print a file from another application, you must configure a network printer to print a file, print to a file, and then send that file to a printer. For details, see the Configuring a Network Printer for LPR in Windows 3.1 section.

Starting the LPR Service from the Network

To start the LPR service from the network, do the following:

1. Choose the Start button to display the Start menu.
2. Choose the Settings command and then the Control Panel command.
3. Choose the Network icon.

   The Network dialog box appears.
4. Choose the Add button that is available on the Configuration tab.

   The Select Network Component Type dialog box appears.
5. Select Service from the list of components and then choose the Add button.

   The Select Network Service dialog box appears.
6. Select NetManage from the list of manufacturers and NetManage LPR Printer Sharing from the list of network services.
7. Choose the OK button.

Adding a Printer

To add a printer, do the following:

1. Open the Add Printer Wizard. To do so choose the Start button, choose the Settings command, and then the Printers command. Next, choose the Add Printer icon.

2. Use the Add Printer Wizard as discussed in your Microsoft documentation. When prompted for the network path of the printer that you want to use for LPR printing, use either of the following formats to enter the desired print server, host, or printer:

   ```
   \\server:LPR:printer
   \\server\LPR\printer
   ```

   For example, suppose a host's name is rainbow and a printer's name is iris. You could add iris as an LPR printer using either of the following formats. Printing from within other applications can then be sent to iris using LPR.

   ```
   \\rainbow:LPR:iris
   \\rainbow\LPR\iris
   ```

Print Format when Printing from within an Application

If you are printing from within an application, the default print type is "v" (Sun raster format).

If you need to override the print format, add the following line to your configuration file, which is LPR.CFG by default:

```
format=<options>
```

The following table describes the options you can include in your format statement.

Note: The print format options are all dependent on the LPD implementation at the remote site. The remote LPD and remote printer must be able to interpret these options.

Option	Meaning
c	Plot CIF file
d	Print DVI file
f	Print formatted file
g	Plot file
l	Print file leaving control characters
n	Print ditroff output file
o	Print PostScript output file
p	Print file with "pr" format
r	File to print with FORTRAN carriage control
t	Print troff output file
v	Print raster file

For example, to set the print type to "print formatted file:"

```
format=f
```

Printing PostScript Files

You can print PostScript files on a printer that is capable of interpreting and producing the PostScript format.

To print PostScript files, do the following:

1. Install the appropriate PostScript printer driver for your specific printer in the Printers section in the Windows Control Panel. Then, select that printer from the Installed Printers list.

2. Choose the Setup button. View the setup screen to confirm your selection. Choose the Close button and then exit the Control Panel.

3. If you are using LPR/LPD in the Windows 3.1 environment, edit your WIN.INI file to include the CtrlD=0 statement.

   You enter the CtrlD=0 statement in your WIN.INI file under the section that contains the name of your redirected PostScript printer. (You must enter the CtrlD=0 statement because PostScript drivers add CtrlD to the beginning of a print job, and the UNIX system interpret CtrlD as an end of file indicator.)

   For example, if you installed an Apple LaserWriter printer driver and connected it to the LPT1 printer in the Control Panel, enter the following:

   ```
   [APPLE LASERWRITER , LPT1]
   CtrlD=0
   ```

   If you installed an HP LaserJet 4/4 printer driver and connected it to the LPT1 printer in the Control Panel, enter the following:

   ```
   [HP Laserjet 4/4, LPT1]
   CtrlD=0
   ```

4. Save your changes. Then restart Windows to make the changes effective.

Configuring a Network Printer for LPR in Windows 3.1

If you are in the Windows 3.1 environment and you want to print a file from another application, you must configure a network printer to print a file, print to a file, and then send that file to a printer. To do so, do the following:

1. Direct your default printer to print to a file by doing the following:

   a) Open the Control panel and then select the Printers icon.

      The Printers dialog box appears.

b) Select a printer and choose the Connect button.

c) Select File from the list of ports and then choose the OK button.

d) Choose the Close button to close the Printers dialog box.

2. Choose the print command in the application from which you want to print. Enter a filename when you are prompted to do so. The information you want to print is saved to a file.

3. Open LPR and then choose the Print File... command from the LPR menu.

   The Print File dialog box appears.

4. Select the file specified in step 2. Then send the file to the remote printer as described in the Printing a File Using LPR section of this chapter.

Diagnosing Client and Host Transmission Problems

To diagnose any client/host transmission problems, use the Log... command to verify communication between a client and host and to verify the transfer of print files. The Log command keeps a log of data transmissions between the client and host.

Choose the Log... command from the Settings menu before sending the file to the printer. LPR will automatically start recording the transmissions as soon as you send the file.

Using LPD

LPD accepts print jobs from remote users and responds to LPR queries on status.

To use LPD in Windows 95 or Windows 3.1, simply open the application to make it available. You can then minimize it.

To use LPD, simply open the application to make it available. You can then minimize it.

Other users can print to your LPD server if they specify your machine's IP address or host name.

LPD sends print jobs to the printer specified in LPR. If that printer cannot be found LPD sends the print job to your default printer.

Additional Online Information

The online help file for LPR/LPD includes a description of each menu command, plus the following information in the "How to Use" section:

- Configuring a Network Printer for Non-NFS Environment (Windows 3.1 only)

- Controlling Print Jobs
- Diagnosing Client and Host Transmission Problems
- Printing a File
- Printing PostScript Files

Troubleshooting

If you experience difficulties using the LPR application, refer to the following items:

- ❏ Check the host side of your printing procedure. Check with your system administrator about the printcap file, which contains all the necessary parameters and other important information about the various printers connected to the host, and how the host handles the filtering. The printcap file is used by LPD implementation in a UNIX environment; the NetManage LPD does not use this file.
- ❏ PostScript printing problems may occur if you are using LPR in the Windows 3.1 environment and your print server is running on a UNIX machine. If you experience PostScript printing problems, enter the CtrlD=0 statement in your WIN.INI file under the section that contains the name of your redirected PostScript printer.

  You must enter the CtrlD=0 statement because PostScript drivers add CtrlD to the beginning of a print job, and the UNIX system interpret CtrlD as an end of file indicator.
- ❏ The network button is active if a network is installed in the Networks dialog box in the Control Panel, and when an LPT port is free. If the network button is dimmed, make sure your network is installed and that an LPT port is free.
- ❏ Use the Ping application to verify connectivity to any remote system to which you want access if a failure to connect is reported.
- ❏ Not every LPD implementation supports the PostScript function. If the PostScript function does not work, select the text option.
- ❏ If an LPR print job keeps spooling in the Windows 95 environment, check to see whether the server is up. A spooling problem can occur because the Windows 95 Add Printer Wizard does not check the status of the server when you add an LPR printer.

Chapter 12. TFTP - File Transfer

The TFTP application provides rudimentary file transfer services, using the Trivial File Transfer Protocol (TFTP). The application allows simple file transfer to and from a remote system without any directory or file listing. Normally, the FTP application is recommended for file transfer services, since it is supported across many systems. However, if your remote system does not support FTP, you may need to use TFTP.

Server functions are also provided permitting other Chameleon TFTP clients and non-Chameleon clients access to selected local files and directories. A TFTP server is generally used for checking or testing in a development environment.

The current connect parameters, server parameters, and options can be saved to a configuration file (default is TFTP.CFG). The configuration file is loaded when the TFTP application is started.

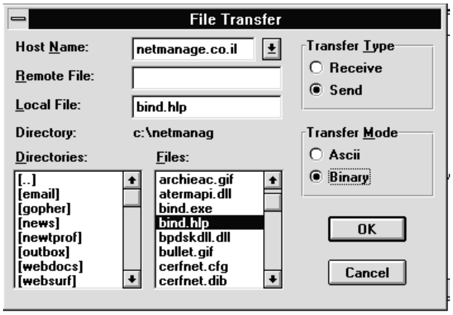
TFTP Client

The TFTP Client transfers files to and from the remote system (Server). Timeout values may be altered and a log produced.

Transferring Files

To transfer a file to or from a remote system, complete the following steps:

1. Select the Transfer menu. The File transfer dialog box appears:



1. In the Host Name drop-down list box, type or select the host name or IP address of the remote system.

2. Select the Transfer Type option button:

   Receive: Specifies that you want to receive a remote file at the local system.

Send: Specifies that you want to transmit a local file to the remote system.

3. Select the Transfer Mode option button (ASCII or binary).

   All transfers are executed in either ASCII (text) or Binary (image) mode. ASCII mode performs carriage return/line feed translation and is only needed when transferring text files for use on a non-Windows system. If you encounter problems or data corruption, it is usually best to transfer in Binary mode.

   If you are unsure as to which mode to use, select Binary.

4. Type or select the Remote File.

   If Receive Transfer Type is selected: a remote filename does not need to be specified.

   If Send Transfer Type is selected: a remote filename specifies how to rename the transferred file on the remote host. If a filename is not specified, the remote filename will be assumed to be the same as the local filename.

5. Type or select the Local File:

   If Receive Transfer Type is selected: specify the same directory as the default directory set through the Server... option of the settings menu.

   If Send Transfer Type is selected: specify the name of the file you want to transfer to the remote Host.

6. Choose the OK button to begin the transfer.

Creating a Log

TFTP lets you create a log (record) of the file transfer operations in your application window and save it into a log file (TFTP.LOG). Select the Preferences... option from the Settings menu and select the appropriate option button. Choose the OK button to start the log. Log entries show transfer start and end summaries, as well as server enabling and disabling.

Setting Timeout

Select the Timeout... option from the Settings menu to modify Timeout values.

The Retransmission value (between 1 and the total transmission seconds) indicates when to send the first retransmission after a failure to receive an acknowledgment.

The total transmission timeout (between 1 and 9999 seconds) indicates the value for the entire transmission; extend this value if you plan to transfer a very large file.

TFTP Server

Set the TFTP Server through the Server... option from the Settings menu. This determines the public directory for TFTP Clients and enables the TFTP Server. The public directory is shared between all TFTP Clients. TFTP Clients cannot access any files or directories outside of this public directory. The Server On check box is used to enable and disable the TFTP Server.

The TFTP Server is activated in two ways:

- On the Server side: select the Server On option from the Settings menu.
- On the Client side: a client can automatically activate the TFTP Server, if the following steps have been completed on the TFTP Server side:

  a) Save the TFTP settings with Server On selected, to the TFTP configuration file.

  b) Through custom, choose the INETD configuration option from the Services menu and create a TFTP Server on port 69 (refer to online help for more details).

  Note: The INETD configuration option does not apply to Windows NT users.

  c) Restart windows.

  Any messages sent to the TFTP Server will automatically activate the Server. After the end of the connection, the TFTP Server closes.

Additional Online Information

The online help file for TFTP includes a description of each menu command, plus the following information in the "How to Use" section:

- Transferring Files
- Setting Timeout
- Creating a Log

- Setting the TFTP Server

Troubleshooting

If you experience difficulties using the TFTP application, refer to the following items:

❑ Make sure the remote system provides a TFTP server and that the server is running. The TFTP server must be explicitly turned on and the application must be running. Some operating systems do not supply TCP/IP services with the standard package (for example, VMS).

❑ Use the Ping application to verify connectivity to any remote system to which you want access if the TFTP application reports a failure to transfer.

❑ Make sure that the transfer type (ASCII/binary) is chosen correctly. Transferring a binary file when the transfer type specifies ASCII, may cause a failure.

❑ Make sure you have permission for specific operations (for example, write access to a directory). The TFTP log can be used to help diagnose access failures.

Chapter 13. File and Disk Replicator

File and Disk Replicator creates an always-up-to-date local copy (replication) of the files from a specified directory of a remote host. The application repeatedly runs an update session that creates a local replication of a directory subtree on a remote host served by an FTP server. As many as 100 different replications of different sites can be handled simultaneously.

Updates are created using a minimal transfer scheme: only files that do not exist in the local site, or files that are different than the ones in the local site, are retrieved from the remote site. You may also specify that the local files which no longer exist on the remote host are deleted.

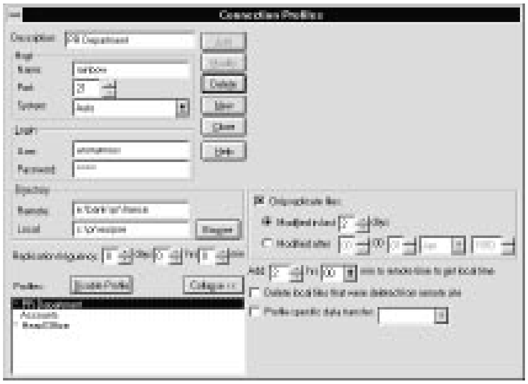
Note: File and Disk Replicator is available only for Windows NT and Windows 95.

Using File and Disk Replicator

To replicate remote hosts you must first create and enable profiles, then select the Connect menu item. The replicating session starts. File and Disk Replicator will run the enabled profiles according to the information specified in the profile’s information and in preferences. File and Disk Replicator should be left running to constantly update local directories according to the time frequencies set in the Connection Profile. File and Disk Replicator updates the local host with files and directories from the remote host(s) - not vice-versa. File and Disk Replicator displays a log in the main window, displaying the progress of file transfer and statistics about the transfer once completed. These messages may also be written to a log.

Creating Connection Profiles

The Connection Profiles command from the Settings menu is used to create, modify or delete File and Disk Replicator profiles, and to enable and disable them.



The dialog window initially appears in a restricted size. Choose the More>> button to extend the dialog and display all the available settings.

The following may be specified for each profile:

- Description: Specify a name for the profile. Existing profile names appear in the profiles list box on the bottom part of the Connection Profiles dialog.
- Host section: Specify the remote Host to replicate:
    - Name: The name or IP address of the Remote host where the directories and files to replicate are located.
    - Port: The number of the port to connect to. The default is 21. The port setting should always be 21 unless you need to connect to a specific FTP server available on a non-standard port.
    - System: FTP server type. If unknown, Auto is the default. File and Disk Replicator will try to identify the system.

  Note: Specifying an incorrect system may result in faulty replicating and even data loss (if the Delete local files that were deleted from remote site box was checked). Therefore it is advised that you set this field to Auto.
- Login section: Specify the login details for the remote Host to replicate.
    - User: The user name that will be used when logging into the FTP remote host. If no user name is specified, anonymous will be used as a default.
    - Password: The password used to log into the FTP remote host. The password is not displayed.
- Directory section: Specify which directories should be replicated.
    - Remote: The directory on the remote host to replicate. The sub directories of the specified directory will also be replicated.
    - Local: The directory on your local machine to which the remote directory will be replicated.

You can use the Browse button to select a local directory.

- Replication frequency: Specify the time frequency (in Days, Hours and Minutes) that the remote host will be replicated.
- Profiles: Displays a list of all the profiles set in the current configuration file. An asterisk to the left of a profile name indicates it is enabled. To enable and disable, select the profile name and choose the Enable Profiles or Disable Profiles button. The disabled profiles are kept in the configuration file, but are not executed when replicating.

In its enlarged mode (after choosing the More>> button) the dialog also contains the following settings:

- Only replicate files: when checked, sets the cut-off date specified below:
  - Check the first radio button to replicate files changed only within the last number of specified days.
  - Check the second radio button to replicate files changed after the specified date (in Hours, Day in Month, Month and year).
- Add: Specify the amount of time (Hours and Minutes) that should be added to the time of the remote host, in order to get the local time (value may be negative). Failing to set this correctly might cause File and Disk Replicator to fail to update local files when the remote file was changed in the last 24 hours.
- Check the Delete local files that were deleted from remote site box for all local files which no longer exist on the remote site to be removed from the local site. This option ensures a complete replication.
- Profile-specific data transfer: Specifies the FTP data transfer mode (ASCII or Binary). If not specified, the data transfer setting from the preferences will be used.

When complete, choose the Add button. The new profile is added to the list of profiles and it is enabled. To modify a profile, select the profiles, perform any required changes and choose the Modify button.

Making File and Disk Replicator Connections

Choose the Connect option to start the replicating process.

Note: The menu option changes to Disconnect allowing you to disconnect the replicating session.

All the enabled profiles begin the replicating process, then repeat it automatically at the specified intervals. The main window displays the replicating messages. At the end of each connection, statistics will be printed to the screen detailing the number of directories and files checked, and the number of files and total bytes retrieved.

Choose the Disconnect option to stop the replicating processes. A confirmation pop up will appear.

Note: The menu option changes to Connect allowing you to connect the replicating session.

Setting Preferences

The Preferences dialog allows you to specify the various global replication settings which affect all profiles.

- When the Write logfile box is checked, the File and Disk Replicator messages are written to a log file (in addition to displaying messages in the main window). The specified log file is used for all the replication profiles. When the box is not checked, no log file is written. Specify the log file name and directory to be used. Check the Append logfile box for log entries of each session to be appended to the entries from the previous session.
- The Alarm level specifies if messages should also appear as pop-ups (in addition to being displayed to the screen and written to the log file - if one has been specified), and if so, which messages.
- The Messaging level specifies which level of messages will be displayed to the screen and written to the log file (if one has been specified).
- The Data transfer section specifies the FTP data transfer mode. The default is Binary.

Additional Online Information

The online help file for File and Disk Replicator includes a description of each menu command, plus the following information in the "How to Use" section:

- Creating Log Files
- Creating, modifying and deleting a replication profile
- Creating a new configuration file
- Opening a replication session and running Profiles

Troubleshooting

If you experience difficulties using the File and Disk Replicator program, refer to the following items:

❑ Due to the fact that local files may be deleted or overwritten during a replicating process, it is advised to refrain from using the local replicate site as an actual working environment. If you wish to modify or save files that have been replicated, it is advised to move them away from the local replicate directory.

❑ All error messages begin with the word Error:, to help searching for errors in the log.

- The Connect option is dimmed when no replicate profiles exist (or none are enabled) in the Connection Profiles dialog.
- Specifying an incorrect system in a connection profile may result in faulty replication and even data loss. It is advised to set this field to Auto.
- If the remote host is a UNIX host, enter the directory name exactly as it appears on the UNIX host. That is:
  `/market/asia/stats`

  and not

  `C:\market\asia\stats`
- As a precaution, File and Disk Replicator will not delete local files in the following cases:

  When the local site bears no resemblance to the remote site

  When the remote site is empty

  If certain connection problems occur while replicating.
- The default replicate settings from the Preferences dialog are used only when a specific replicate profile does not contain overriding information.
- File and Disk Replicator is not case sensitive, but UNIX machines are case sensitive. File and Disk Replicator will not differentiate between similar file names, written in a different case. Therefore, only one file will be replicated in the instance of two similar names of UNIX files, but with different cases. Also, an incorrect file maybe copied over an existing local file, because the name, (but not the case, making it a different file on UNIX), is the same.

Chapter 14. NFS Utilities

The file conventions used by Windows are different than those used by NFS files, which are specified by UNIX standards. NFS Utilities provides smooth integration between your local Windows files and your network files. This chapter discusses using NFS Utilities to do the following:

- converting text files between UNIX and Windows
- displaying filename mapping so you can view UNIX filenames and their corresponding mapped Windows filenames

Note: You can display filename mapping only while one or more NFS drives are mounted. However, you can convert files at any time.

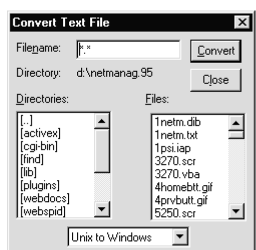
Converting Files

To indicate the end of a line, UNIX and Windows text files use different termination characters. Specifically, UNIX uses the <LF> character to mark the end of a line while Windows uses the two-character combination <CR> <LF>.

To convert one end-of-line convention to the other in a text file, do the following:

1. Choose the NFS Utilities icon to display the NFS Utilities window.
2. Choose the Convert... command from the Utilities menu.

   The Convert Text File dialog box appears.



3. Select or type the name of the file you want to convert.

   If the desired file is not in the current directory, select items in the Directories box until you locate the correct directory.
4. Select the conversion direction (Windows to UNIX, or UNIX to Windows) and choose the Convert button.

Displaying Filename Mappings

Windows filename conventions are more restrictive than those in UNIX. Because of this, some UNIX filenames need to be mapped to Windows-compatible filenames.

NFS automatically maps UNIX filenames to Windows filenames. This mapping is identical to the mapping performed in a variety of other DOS-based NFS products. Mapped filenames appear in the form XXXXX~YY.ZZZ. A tilde (~) is inserted as the sixth character and the next two characters are a uniquely generated key. In addition, a tilde (~) replaces any periods that appear in the UNIX filename.

To display original UNIX filenames and their corresponding mapped Windows filenames, do the following:

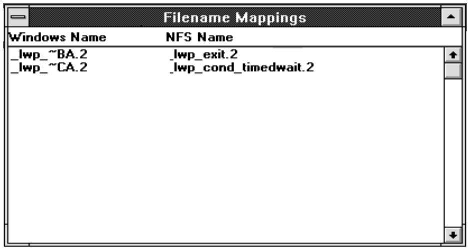
1. Mount an NFS drive from a UNIX computer or server. You mount drives in Windows NT or Windows 3.1 through the File Manager. You mount drives in Windows 95 through the Windows Explorer. For details, see the NFS chapter.

   The connection you establish must be maintained during the rest of this procedure. You can minimize the File Manager or Windows Explorer window after you make the connection.

2. Choose the NFS Utilities icon.

3. Choose the Mappings... command from the Utilities menu.

   The current filename mappings are displayed in a Filename Mappings window similar to the following figure. Filenames listed under the NFS Name column are the original UNIX filenames.



Additional Online Information

The online help file for NFS Utilities includes a description of each menu command, plus the following information in the "How to Use" section:

- Converting Files
- Displaying Filename Mapping

Chapter 15. Z-Mail® Pro and Z-Mail® Pro Administration

The Z-Mail® Pro application provides electronic mail services for both TCP/IP and SNA network environments. To communicate within the standard Internet (TCP/IP) environment, Z-Mail® Pro uses the Simple Mail Transfer Protocol (SMTP) to both send and receive mail, and the Post Office Protocol (POP2 and POP3) to retrieve mail from a server. You can use these protocols to directly integrate with UNIX mail. Z-Mail® Pro supports MIME (Multi-purpose Internet Mail Extensions) which allows attachment of files to mail messages.

Z-Mail® Pro in a previous version was called Mail.

Features include the ability to:

- read and prepare messages off-line, saving you connection costs
- attach multimedia files to your messages through MIME (Multi-purpose Internet Mail Extensions)
- view a message in regular text format or in standard HTML (Hypertext Markup Language) format and edit HTML formatted text directly in WebSpider
- support MAPI-enabled (Messaging Application Programming Interface) applications
- route messages to others one after another
- retrieve message header only information using the Smart Pop feature
- create rules for automatically filtering and filing incoming mail
- create your own personal address books
- attach binary files through uuencode/uudecode
- set up multiple mail accounts
- create rules to activate the new Z-Mail® Pro scripting language (using VBA scripts)
- view graphic attachments using Images

IMAP

Z-Mail® Pro uses the IMAP4 (Internet Message Access Protocol), which is a client/server protocol , for manipulating remote message files. Remote and local folders are fully integrated in Z-Mail® Pro, messages are viewed and folders managed in the same way. IMAP accesses mailboxes (folders) on remote servers as if they were local mailboxes, using the same commands such as moving to folders, search, marking messages. For further information on these functions, see the IMAP Transport section in this chapter. IMAP features include:

- access to multiple hierarchical structured mailboxes on the same or different remote servers
- shared mailboxes between users
- only headers displayed - only when you view a message is it fetched, avoiding unnecessary retrieval of large attachments and saves bandwidth.
- allows you to manipulate remote mailboxes (as if they were local folders): creating, renaming, opening and deleting mailboxes, and searching the current remote folder.
- allows you to download specific messages or the full contents of a specific remote mailbox.
- empties remote trash
- by selecting your private remote INBOX folder, you may read you new mail through the IMAP server.
- Periodic/on demand checking and retrieval of recent mail from all connected INBOX folders.
- dragging and dropping between folders; messages from your old folders can be easily moved into new IMAP remote folders.
- right click of the mouse (or alt + enter) to access drop down menu commands on remote host.
- connecting and disconnecting of remote servers from the main window.
- supports other NetManage messaging applications, such as Phone Tag, Tag-it!.

Fax

To send and receive faxes, Z-Mail® Pro uses the separate Chameleon Fax Server to translate the messages into/from a format suitable for faxing. For further information see the Using Z-Mail® Pro Fax section in this chapter.

- send and receive faxes and route incoming faxes
- view, store, and print faxes from various file formats

PROFS/OV

To communicate with an IBM host, Z-Mail® Pro uses the PROFS/OV protocol. For further information see the PROFS/OV Transport of, section in this chapter.

Z-Mail® Pro Administration

The Z-Mail® Pro Administration application allows you to manage your Z-Mail® Pro database. Z-Mail® Pro Administration is a separate icon that is used exclusively with Z-Mail® Pro.

Note: For detailed online help information about Z-Mail® Pro Administration, select the Help button from within the Z-Mail® Pro Administration application.

Setting up Z-Mail® Pro

This section describes, step-by-step, how to set up a Z-Mail® Pro account for the first time user and how to add additional users. It also includes a brief discussion about Z-Mail® Pro configuration files and the Z-Mail® Pro log feature.

Setting Up a New Z-Mail® Pro Account for a User (First Time Only)

To set up a Z-Mail® Pro account for a user for the first time, you need to complete the New User dialog box. Note that if you want to use the Phone Tag, Tag-It!, Forum, NEWTCal, Mailbox Converter and InPerson applications, you need to have this account set up.

To set up a Z-Mail® Pro account for a user for the first time, follow these steps:

1. Open Z-Mail® Pro. The Z-Mail® Pro **-** New User dialog box appears.

2. Under Local, enter the user's name, user's password (optional), user's name in real life, and local mail directory in the designated fields.

   The local mail directory identifies where all configuration files and individual mail messages are saved on your machine. You may set up a separate mail directory for each user. For example, the mail directory, c:\netmanag\email\steven, is set up only for user Steven.

   Choose the OK button.

3. The Z-Mail® Pro Transport Setup dialog box appears. You may select any one of the check box items:

   - Internet Mail: Indicates that you want the Z-Mail® Pro application to be set up for handling standard Internet e-mail. You need this transport to use Internet Mail and Forum.

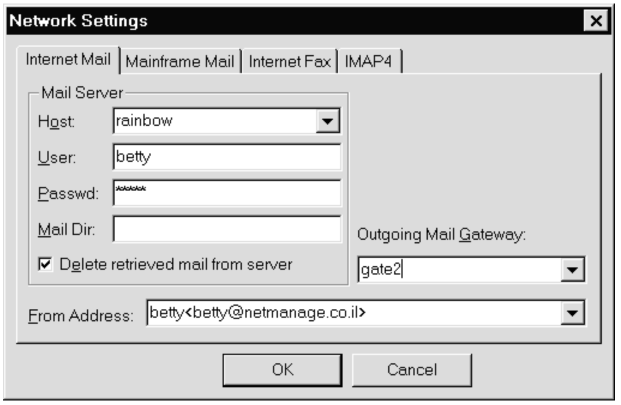
- IMAP4: Indicates that you want the Z-Mail® Pro application to be set up for handling mail on remote servers using the IMAP4 protocol. If you want to select this option see the IMAP section in this chapter.
- Internet Fax: Indicates that you want to email the Chameleon Fax Server, and have it fax the message to a fax machine. If you want to select this option see the Using Z-Mail® Pro Fax section in this chapter.
- Mainframe Mail: Indicates that you want the Z-Mail® Pro application to be set up for handling mail between your PC and an IBM host. If you want to select this option, see the PROFS/OV Transport of Mail section in this chapter.

Choose the OK button.

4. The Network Setting dialog appears, with a tab for each of the Mail transports you selected earlier. Use the various tabs to set the appropriate information.

Internet Mail Tab

Use the Internet Mail Network Settings tab dialog to specify information for standard email:

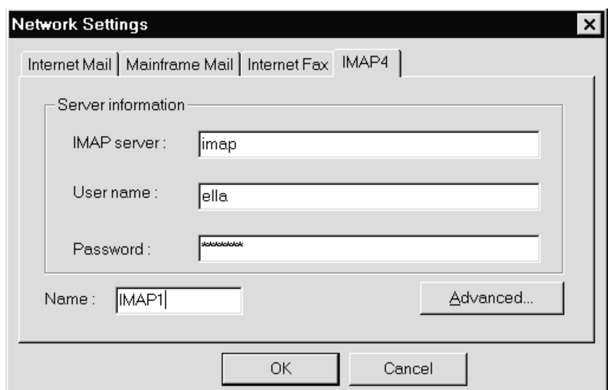


Internet Mail Setting	**Description**
Host	Specify the name of the computer system acting as the mail server.
User	Specify the mail user.
Passwd	Specify the password for the mail user.
Mail Directory	Specify the mail directory (the directory where incoming mail is stored). This is optional.
Delete retrieved mail from server	Select this option if you want your mail automatically deleted from the server after you have retrieved it.

Internet Mail Setting	Description
Outgoing Mail Gateway	Type or select the name of the mail gateway. A mail gateway routes all mail messages that are transmitted to the Outbox. A mail gateway is often used for access to the Internet. The mail gateway must be configured to perform as an SMTP relay. If you do not define a mail gateway in Z-Mail® Pro, the mail gateway you define in Outbox is used instead.
From Address	Type or select the name in the From field you want to appear as the sender when mailing a message. Example: Jane Doe <jane@netmanage.com>

IMAP4 Tab

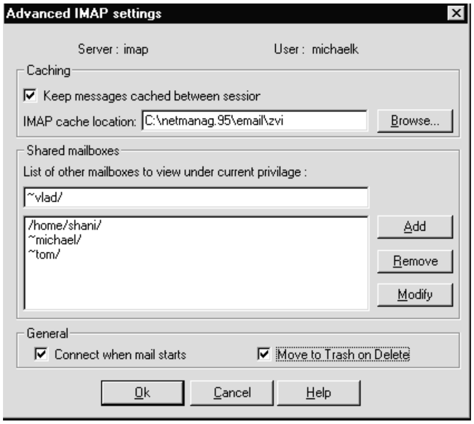
Use the IMAP4 Network Settings tab dialog to specify information for Remote mailboxes using the IMAP4 protocol:



IMAP Mail Setting	Description
Connection Name	Enter the name you want to appear on the IMAP tab in Network Settings. This field will appear only when installing multiple IMAP transports.
IMAP Server	Specify the fully qualified name or IP address of the computer system acting as the mail server.
User name	Specify the mail user for remote login authentication.
Password	Specify the password of the mail user for remote login authentication.

Use the Advanced button to set the following:

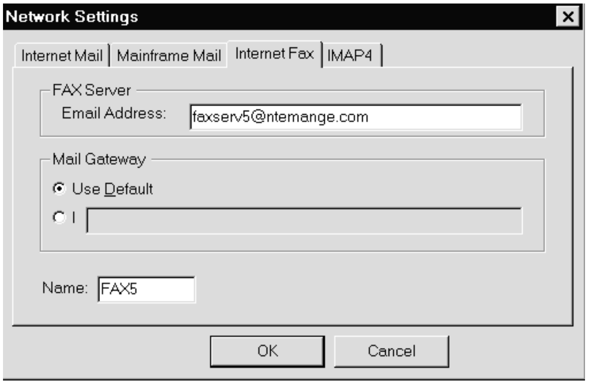
Keep messages cache between sessions	Viewed messages are automatically saved on the local drive for while Z-Mail® Pro is running. Select this option, to save these messages after exiting mail, and retrieve on reactivating mail.
IMAP cache location	Enter, or use the browse button to select the directory where to store the cache files.
Shared Mailboxes	Enter the additional mailboxes you want to access into the top text box Choose the Add button. The additional mailboxes are listed. To modify or delete a shared mailbox, select the mailbox from the list and choose the appropriate button. Use this option only with Washington style servers. this is dependent on mailbox privileges allocated to your logged in user.
Connect when mail starts	Select this option, for connection to this specific server to be automatically established when you open your Z-Mail® Pro application.
Delete through Trash	Messages are deleted immediately, and not moved into the trash folder. This make deletion faster, but removes any possibility to retrieve messages accidentally trashed.



Note: More than one IMAP transport can be added by selecting the Add button and then double clicking on the IMAP4 item.

Internet Fax Tab

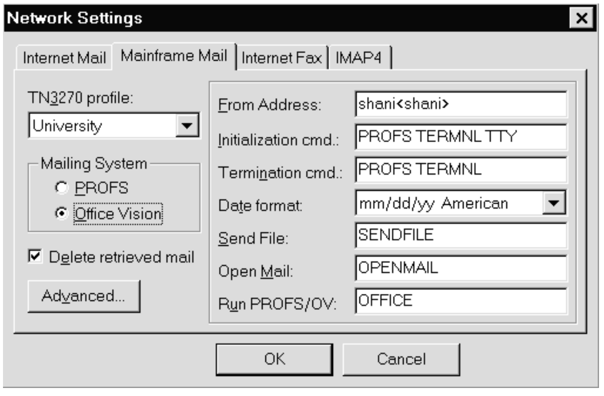
Use the Fax Network Settings tab dialog to specify information for sending faxes:



Fax Setting	Description
Fax Server	Specify the email address of your Chameleon Fax Server (available from your system administrator)
Mail Gateway	Select the Use Default radio button (the mail gateway), or enter the name of the gateway you require.
Name	Enter the name you want to appear on the fax tab in Network Settings.

Mainframe Mail Tab

Use the Mainframe Mail Network Settings tab dialog to specify information for PROFS or OV mail:



Mainframe Mail Setting	Description
TN3270 Profile	Select the TN3270 profile that you want to be invoked upon opening mail. If you do not have an appropriate connection profile, use the 3270 Terminal application to set a connection profile. Then, exit the 3270 Terminal application and start Z-Mail® Pro. The profile must contain the following information: Model = 2 Port = 23, the Telnet communications port.
Mailing System	Select PROFS or OfficeVision. You must select one of these systems.
Delete retrieved mail from server	If this option is selected **-** your mail is automatically deleted form the mainframe after you have retrieved it. If this option is not selected **-** messages are not deleted from the mainframe after their retrieval.
From Address	Enter the address you want to automatically appear in the from field of messages you create. This is a required field.
Initialization	Indicates either a command that puts PROFS/OV in line mode, or an EXEC that contains initialization commands for the PROFS/OV application running on the host system (Such a command or script is generally provided by the host administrator).
Termination	Additional CMS commands or EXECs that should be executed after running PROFS/OV
Date Format	Select the date format for your PROFS/OV system.
Send File	Default: sendfile (or sf)*. This is a required field.
Open Mail	Default: openmail*. This is a required field.
Run PROFS/OV	Default: PROFS (or OFFICE)*. This is a required field.

After you have specified these settings, Z-Mail® Pro will automatically invoke the appropriate settings when you select the type of mail that you want to create.

Note: When upgrading from a previous version of Z-Mail® Pro, you will not need to create a user account again. The following message will be displayed the first time Z-Mail® Pro is opened after upgrading from release 4.5 and earlier: "This version requires a conversion of your current messages into a new format. This may take a while." Choose the Continue button to complete the conversion. Your rules, folders, and messages will remain intact. If you have release 4.5 or earlier you must use this conversion of your old messages to use them with this version of Z-Mail® Pro.

Setting Up Z-Mail® Pro Accounts for Additional Users

Once you have set up an account for the first user, you can set up mail accounts for additional users, by first logging in as Postmaster and then as User. The steps are as follows:

Postmaster Login Steps

To log in as Postmaster and set up accounts for additional users, do the following:

1. Open Z-Mail® Pro. The Z-Mail® Pro - Login dialog box appears. Select Postmaster from the Username drop-down list box and choose the OK button.

2. The Z-Mail® Pro Transports Setup dialog box appears.

   - If you do not want to receive or send mail as a postmaster, for instance you only want to add/modify additional users, choose the cancel button.
   - If you want to send/receive mail as Postmaster, select the appropriate check box item(s): Internet Mail, IMAP4, Internet Fax or Mainframe Mail and choose the OK button. The Network Settings dialog box appears. Choose the Apply and then OK buttons.

3. Choose the Mailboxes... command from the Services menu to display the Mailboxes dialog box.

Note: The Mailboxes... command under the Services menu only appears when you are logged in as Postmaster. This is the only dialog box in which you can add and delete users and specify a user's mail directory.

4. Enter the user's name in the User text box and choose the Add... button. The Enter Information for new user dialog box appears.

5. Under Local, enter the user's name, user's password, user's name in real life, and local mail directory in the designated fields.

   The local mail directory identifies where all configuration files and individual mail messages are saved. You may set up a separate mail directory for each user. For example, the mail directory, c:\netmanag\email\steven, is set up only for user Steven.

   Choose the OK button.

Logging into Z-Mail® Pro as a User

To login into Z-Mail® Pro as a user, do the following:

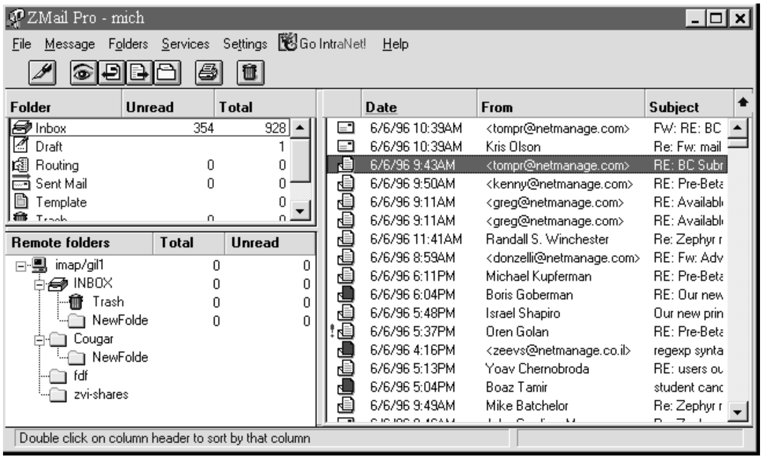
1. Open Z-Mail® Pro. The Z-Mail® Pro - Login dialog box appears. Enter your previously defined user name and password and choose the OK button.
2. The Z-Mail® Pro Transport Setup dialog box appears. You may select any one of the check box items: Internet Mail, IMAP4, Internet Fax and Mainframe Mail . Choose the OK button.
3. The Network Setting dialog appears, with a tab for each of the Mail transports you selected earlier. Refer to the previous section that discusses how to set up a new Z-Mail® Pro account to enter network settings information.
4. After you have entered these settings, Z-Mail® Pro will automatically invoke the appropriate settings when you select the type of mail that you want to create. You can now begin using Z-Mail® Pro.

Z-Mail® Pro Icon

When you minimize the Z-Mail® Pro application, an animated icon of a mailbox appears. When new mail arrives in the current folder, the mailbox door opens.

Organizing Your Z-Mail® Pro Messages

In order to organize your messages efficiently, the Z-Mail® Pro application allows you to create folders into which you can place your mail messages.



When the main Z-Mail® Pro window is displayed, you will notice that the window is split into two panes: the left pane is called the Containers pane and the right pane is called the Contents pane. The Containers pane is also split into two panes; the top pane contains a list of all local mail folders, and the bottom pane contains a list of remote mail folders accessed by IMAP4 (Internet Message Access Protocol). The Contents pane contains a list of all the messages in the current folder.

In the Containers pane, the names of existing folders, for both local and IMAP, are displayed along with each folder's icon, number of unread messages and total number of messages. The default folders in descending order are: Inbox, Draft, Routing, Sent Mail, Template, and Trash. All folders you create are listed alphabetically.

Note: You cannot delete the Inbox, Draft, Routing, Sent Mail, Template, or Trash folders. These folders always appear at the beginning of the Folder list even when they are empty.

In the Contents pane, a list of all the messages in the current folder are displayed. You can drag and drop a message from the right side of the window into the desired local folder on the left side, much like moving files into directories in File Manager.

Creating and Sending Z-Mail® Pro Messages

To create and send a Z-Mail® Pro message:

1. From the main Z-Mail® Pro window, choose the Create button, or choose the Create command from the Message menu, or CTRL-E.

2. A new mail message window appears. If you have enabled settings for more than one transport (Internet Mail, Internet Fax or Mainframe Mail), the New Mail dialog box contains a drop-down list to select the desired Mail Transport.

    - Internet: Select this transport to create a standard Internet Mail email message.
    - Internet Fax: Select this transport to send a fax using NetManage’s Z-Mail® Pro Fax Server
    - PROFS/OV: Select this transport to create an IBM mail message.
    - News: select this transport to post a message to a specified newsgroup.

    The Date field is automatically set to the current date and time. The From field is automatically set to the From field that you defined in Network Settings. If this has not been defined it is automatically set to your user@localhost.domain and when this host has also not been defined, to user@popserver. .

3. You may address your message as follows:

    - Enter the e-mail name and address of the person you want to send the message to in the text box next to the To field. Then, using the down arrow, select either To (primary recipient), cc (carbon copy), or Bcc (blind carbon copy).
    - To add additional recipients from an address book: choose the Address button to display the Address Book dialog box containing your address book(s) and their entries. Then, select or type each entry you want to add as a recipient, and

choose the To, Cc, or Bcc buttons. Choose the OK button to return to the new mail window.

- News: If you have selected News, type the newsgroup you require and press ENTER. When the address appears in the field below, `<@News:>` is automatically added to the address.
- PROFS/OV: If you have selected PROFS/OV, type the address you require and press ENTER. When the address appears in the field below, `<@PROFS:>` is automatically added to the address. (If Internet is selected, pressing ENTER copies the address to the field below, without any additions.)

  If the PROFS/OV address is written on the mainframe as: `(userid)host`, type the mail address field as `userid@host`. After pressing ENTER, it appears in the field below as

  <@PROFS:userid@host> (or as <@OV:userid@host>).

Note: To route your message to others, choose the Add Routing Slip command from the Message menu.

4. If desired, you can select the Confirm Read and Confirm Delivery options. Select Confirm Read if you want the system to return an acknowledgment that the addressee opened the message. Select Confirm Delivery if you want the system to return an acknowledgment that the message reached the addressee's mail server. Acknowledgments are returned directly to your Mail Inbox folder.

   Note: This option is applicable only if the receiver of the message is running Z-Mail® Pro.

5. If desired, select the priority type of the message you are sending. You can select high, normal, or low priority. Normal priority is the default. If you assign high priority to your message, then a red exclamation point appears next to your message header in the recipient’s Inbox. Assigning low priority to your message causes a down arrow to appear next to your message header in the recipient's Inbox.

6. Type the subject and message text, and if desired, choose the Feelings menu to add a smiling, frowning, or winking symbol in your message.

7. Choose the Send button to transmit your message to the Outbox. The message is immediately queued in the Outbox for subsequent delivery to it defined destination.

Encryption

Encryption is a means of permitting higher security for your mail. The contents of a mail message are encoded, and secured with a key. A key is like a password. The Sender creates a key and informs the receiver of the encrypted message of the value of the key used. The receiver will use this same value to create a key to decrypt the text. If the receiver does not have the appropriate key (key with the same name and value) in the key list, a dialog box appears, and the value must be entered. The user may create numerous keys, each key has a name and value. In one message, you may use as many different keys as required. Encryption uses algorithm DES-40.

Encrypting Text

Encrypt text encodes a message or a selected portion of a message, which can later be decoded, to return to plain text. To encrypt text do the following:

1. Select the text in the mail message you want to encrypt.

   Note: If no text is selected, the whole message will be encrypted.

2. Choose Encrypt Text from the Message menu.

   The Encryption dialog appears.

3. Select the Key you require for encryption, or create a new key.

   To create a new key, enter the following:

   - Name: the name that will be listed in the Key List.
   - Value: the password needed to decrypt the message. The password uses the first ten characters. The password is not case sensitive.

   The key icon (under Value) changes to represent the security level of the password. The more characters you enter, the safer the password, and the longer the key icon. It is recommended for maximum security you use the ten characters.

   Note: You must obtain the appropriate Key value from the person that sent the message.

4. Select the OK button. The text is now changed into encrypted text.

Decrypting Text

Decrypting text removes the encryption and returns the text to readable text. To Decrypt text, do the following:

1. Choose Decrypt from the Message menu.
2. If you have the correct key in your key list, the message is automatically decrypted.
3. If you do not have the key, the New Key dialog appears. Enter the value for that key. For messages encrypted with a number of different keys, you will be prompted for the value for each key.

   The new key and its value is saved in the key list.

   To decrypt just one block of encrypted text, select only that text, then choose Decrypt.

All encrypted messages are saved in the database as encrypted. After decrypting, when you forward the message, encryption is again enforced.

Viewing Z-Mail® Pro Messages

You can view a message in standard text format. You can also view a message in HTML format or the Web page of a referenced URL path contained in the message (see section on viewing HTML documents).

1. Select the desired folder, and then the message you want to view.
2. Choose the View button to display the message or double-click on the message.

   The view window displays the date, source address, subject, list of recipients, and the message text. You cannot modify any of these fields, however the Copy function within the Edit menu is available. The Copy function allows you to transfer highlighted message text to other applications.

   Choose the uudecode option to decode and save a uuencoded binary file that has been sent as a message you are viewing.

An override option to show the mail header is also provided. The initial setting of Show Header is determined from the setting in the Preferences dialog box.

Replying to and Forwarding Z-Mail® Pro

To reply to a message in any folder:

1. Select the message.
2. Select the Reply to Sender or All option from the Message menu, or choose the Reply button located on the mail message

   A window identical to the sending mail display appears. The Date and From text boxes are set normally. If you Replied to All, the list of recipients contains the source address of the original message and also any carbon copy addresses. Note that in many cases your own address also appears. If you do not want to receive a copy of the message you want to send, delete your name from the list.
3. Enter your message and choose the Send button to send the mail.

   The text of the original message, for both reply and forwarding, can be left intact with a separator indicating where the original message begins (depending on your setting in Preferences). The text insertion point is placed above this separator so that you can type your reply or forward text above the original message.

To forward a message from any folder:

1. Select the message.
2. Choose the Forward button. A window identical to the sending mail display appears.
3. Enter the recipient and your message and choose the Send button to send the mail.

Viewing and Editing HTML Documents

Z-Mail® Pro provides the ability to view an HTML formatted document in standard HTML format when it is part of the message body, included as an attachment, or referenced as a URL path within a message. You can also edit an HTML formatted document, in WebSpider, when it is part of the message body, included as an attachment, or referenced as a URL path within a message.

Viewing an HTML Document

You can easily view an HTML document, or directly link to its web page without having to use a web browser. To view an HTML document when it is part of the message body or as an attachment, do the following:

1. Select the message that contains the HTML formatted document. If it is in an attachment, view the attachment.

2. Choose the Display in HTML command from the Message menu.

3. The HTML formatted document appears in standard HTML format. You can view this document and any of its jumpers or links as you would if using a web browser.

   Note: The Display in HTML command is a toggle. When selected, it displays an HTML formatted document in standard HTML format, when unselected, it displays an HTML formatted document in standard text format.

To view an HTML document when it is referenced as a URL path within a message, do the following:

1. Select the message that contains the URL path reference.

2. Choose the Display in HTML command from the Message menu. The referenced URL path appears highlighted in blue.

3. Single click the highlighted URL path to directly link to its web page. You can view its web page in Z-Mail® Pro or in WebSurfer.

Editing an HTML Document

You can also edit an HTML formatted document by sending it directly to WebSpider, NetManage's Web authoring tool. After editing in WebSpider, you can send it as a mail message to another user. To do this:

1. Select the message that contains an HTML formatted document.

2. Choose the Edit HTML command from the Message menu. WebSpider is automatically launched and the HTML formatted text from mail message now appears in the WebSpider window.

3. Edit the HTML document as desired.

4. Choose the WebSpider Send button to send the edited HTML document as a mail message.

Routing Messages to Others

The Routing Slip feature lets you send a message to a predefined route of addresses in the order you specify. Instead of sending a message to multiple recipients simultaneously, the Routing Slip feature lets you send a message to recipients one after the other. That is, after the message is viewed by a recipient, he or she can choose whether to send it to the next recipient on the routing slip.

Creating and Routing Messages

To create and route a message to others, do the following:

1. From the main mail window, choose the Create button, or choose the Create command from the Message menu, or CTRL+E. A New Mail window appears.

2. Choose the Add Routing Slip command from the Message menu. A New Mail-Routing Slip dialog box appears.

3. Choose the Add Address button to display the Address Book dialog box that contains your address book(s) and their entries. Then, select or type each entry you want to add as a recipient to your routing slip and choose the To button. Choose the OK button to return to the New Mail-Routing Slip dialog box. Notice that all mail recipients are listed and numbered in the Route text box.

4. To modify the order of recipients, select an address and move it using the arrow buttons to the right of the address list.

5. If desired, select the Return At End, and/or the Allow Modifications options, and choose the OK button. The name "Routing List" appears in the address textbox.

   The Return At End option sets your address as the last address on the route, thus making the route into a loop. The Allow Modifications option indicates that you allow the recipients to modify the message's text as the message is routed along.

   If desired, you may later edit the route before sending it or deleting it. Do this by displaying the Routing Slip dialog box or by selecting Routing List in the address text box and choosing the Modify or Delete buttons, respectively.

6. Finish completing your message and choose the Send command from the Message menu, or choose the Send button to mail the message. The newly sent message is placed in the Routing folder.

7. The status of the routed message is updated as the message moves along from one recipient to the next. Whenever the status is updated, the message is marked as unread. All routed messages are stored by default in the Routing folder.

Viewing Routed Messages

You can view a routed message the same way you view a regular message. Routed messages are stored in the Inbox folder unless otherwise moved by a previously set rule.

When you view a routed message, a new menu named Routing appears in the menu bar. This menu has three options: Route to Next, Information, and Block Route.

The Route to Next option lets you send the message to the next recipient listed in the route and notify the originator of the progress along the route. If the route has the Allow Modifications option set, you may modify the message text by typing in the text area before routing to the next recipient. The original text of the message will be retained, while the new text is available to future recipients.

The Information option lets you display information about the route. The Routing Information dialog box appears containing a list of addresses (if you are a recipient) or the message status (if you are the originator).

The Block Route option lets you stop the message from being routed to any other recipients. The originator of the message is also automatically notified that the routing has stopped.

Deleting Z-Mail® Pro Messages

To delete one or more messages:

1. Select the message(s) you want to delete.
2. Choose the Trash icon or the Delete option from the Message menu.

   The deleted message is automatically transferred to the Trash folder. All messages remain in the Trash folder until you select Empty Trash from the Services menu or until you exit Z-Mail® Pro (depending on your setting in Preferences).

To delete a folder, refer to the Folders section in Z-Mail® Pro online help.

To empty the local Trash folder, select Empty Trash from the Services menu. You can also select the Empty trash on exit option from the Preference menu if you want to delete Z-Mail® Pro in the Trash folder each time you exit Z-Mail® Pro. To empty a remote Trash folder, select the Trash folder, click the right mouse button, and select Empty.

Appending Z-Mail® Pro Messages to a File

Z-Mail® Pro allows you to append the contents of one or more messages to the end of a specified existing file.

While viewing or creating a message, select the Append to File... option from the Message menu and specify the file you want to append to. The contents of the message will be appended at the end of the specified file.

Note: If you select more than one message, the information will be appended in the order in which you selected the messages.

Saving Z-Mail® Pro Messages to File(s)

Z-Mail® Pro allows you to save the contents of one or more messages to a new or existing file.

While viewing or creating a message, select the Save to File... option from the Message menu and specify or select the file to which you want to save the message. The contents of the message will be saved to the specified file.

Note: If you select more than one message, the information will be saved in the order in which you selected the messages.

Creating and Updating a Draft Message

You can save and store an unfinished mail message and then send it at a later date by using the Save As Draft and Update Draft mail features. This is useful when you wish to complete other tasks, before sending the mail.

To create a draft message:

1. Select the Create... option from the Message menu. A New Mail message form appears.

2. Enter any information or text required, as you normally would when creating a new message.

3. Save the unfinished message by selecting the Save As Draft option from the Message menu. The unfinished message is now saved in the Draft folder.

Note: The Draft folder always appears at the top of the Folder list, after the inbox folder, even when it is empty. You cannot delete the Draft folder.

To update an existing draft message.

1. Select the Draft folder and then select the draft message you want to update.

2. Make the desired changes to the message and select the Update Draft option from the Message menu and close the message.

   The draft message is now updated and stored in the Draft folder.

To complete and send a draft message, simply select the desired draft message and complete it. Then send the message by choosing the Send icon on the Toolbar or select the Send option from the Message menu. The draft message is automatically deleted after you send it.

Creating and Saving a Template

To create a template:

1. Select the Create... option from the Message menu. A New Z-Mail® Pro template appears.

2. Enter the subject and message text you want to be part of the template. If desired, you can also enter an addressee. You can create a template without entering a

subject, however a template is easier to identify in the Template folder if it contains a subject.

3. Save the template by selecting the Save Template option from the Message menu. The template is now saved in the Template folder.

Note: The Template folder always appears at the top of the Folder list even when it is empty. You cannot delete the Template folder.

To update an existing template

1. Select the Template folder and then select the template you want to update.

2. Make the desired changes to the template and select the Update Template option from the Message menu and close the template.

   The template is now updated and stored in the Template folder.

To complete and send a template, simply select the template and add any additional information. Then send the template by choosing the Send icon on the Toolbar or select the Send option from the Message menu.

Using Attachments

The Z-Mail® Pro application allows you to attach a physical file or a pointer to a file in a mail message. If you send a pointer to a file, then the recipient can access the file through the FTP client. However, for the recipient to access the file through the FTP client, the sender’s FTP server must be running on his or her system.

All files can be either binary or ASCII. For example, you can attach a spreadsheet file (binary) to a mail message that you are sending to a list of recipients.

Sending a File as an Attachment

To attach a file to a mail message, do the following:

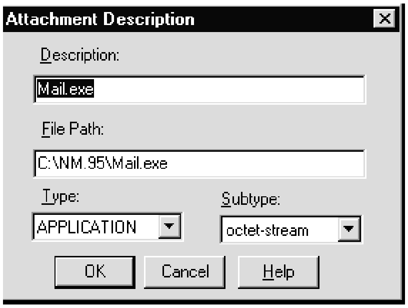
1. Create a new message.

2. Choose the New... command from the Attachments menu.

3. Select or type the name of the file you want to attach to your current message.

4. File Type selection includes text, Message, Application, Audio, Image and Video. The Subtype field will change according to the defined Type.

5. Choose the OK button to attach the specified file to your mail message.



Viewing and Saving File Attachments

To view a file attachment of an incoming message, do the following:

1. Select the message that includes the attachment., and click on the file attachment you want to view.

2. Choose the View command from the Attachments menu. The attached file appears using the appropriate editor. For example, if the attached file a .BMP file, the file will be opened using Window's Paintbrush application. Files use the File Manager Association file extensions.

To save a file attachment to a specific location, do the following:

1. Select the message that includes the attachment, and click on the file attachment you want to save to another location.

2. Choose the Save command from the Attachments menu. The Save Attachment As dialog box appears.

3. Select the desired drive, directory, and file type. Enter the name of the file you want to save the attachment as in the Filename text box.

4. Choose the OK button.

Uuencoding and Uudecoding Messages

Uuencode and uudecode allow the mail user to convert binary files from and into ASCII format. Choose the uuencode option to uuencode a binary file before you send it out and the uudecode option to uudecode a message you have received as a specified file on your machine.

Note: A mail message can have both a file attached to it and a uuencoded file included in its body.

To uuencode a file, do the following:

1. Create a new Z-Mail® Pro message.

2. Select the uuencode... command from the Message Menu.

3. Select or enter the file that you want to uuencode and choose the OK button. The file will display in ASCII form in the body of your mail message.

4. Fill out both the To and Subject fields and send your message.

To uudecode a file mailed to you, do the following:

1. Select the message you want to uudecode.

2. Choose the View button to display the message.

3. Select the uudecode... command from the Message menu.

4. Select or enter the file to which you want the uudecoded message to be saved. Then, choose the OK button. The file is now converted into its original binary format.

Note: Some mail applications can uudecode only one uuencoded file at a time. Therefore, you may want to uuencode only one file at a time.

Using the Spelling Checker

To check the spelling of your mail message:

1. With the cursor in the body of the mail message you are creating, select the text in which you want to check spelling. If you do not select text, the entire mail message is checked beginning at the cursor.

2. Select the Check Spelling... option from the Tools menu. The Spelling Check dialog box appears if there are errors.

   The spelling checker begins checking the selected text or the mail message. When it finds a word that is not in any the default dictionary it displays the word in the dialog box and highlights it in the message.

   For each word found, you can:

   - Correct the spelling **-** In the Replace With field, type the correct spelling. Or, in the Suggestions field, select a correction. Then, choose the Replace button to change the current word only, or Replace All to change all instances of the word.
   - Leave the word unchanged and continue the spelling check by choosing the Ignore button.
   - Skip the current word and all other instances by choosing the Ignore All button.
   - Add the word to a user dictionary so the spell checker will not consider it misspelled in the future.

3. The Spell Checking is Complete dialog box appears when the speller is finished. Or, choose the OK button to return to the mail message.

Retrieving Message Headers

The Smart POP feature lets you retrieve message header information only (subject, date, time, sender) without needing to retrieve the entire message or attachments. This feature applies only to Internet protocols.

Note: The Smart POP feature is considered an advanced Z-Mail® Pro option and should be used with caution.

Normally, most users want to retrieve their mail messages in their entirety including attachments. However there may be circumstances, during serial dialup for example, when you only want to retrieve the message headers only. This is very convenient for users who do not want to retrieve large messages from the server to their own machine when working remotely.

The Smart POP dialog box has the following options.

Option	Description
Retrieve Headers Only for messages over xx KB	Lets you retrieve message headers only for messages that are above a certain size. All messages under the stated size are retrieved in their entirety. You can set the maximum message size in the KB text box. (Default is 1024 KB)
Always retrieve headers only	Lets you retrieve message header information only from messages on the server. The entire message still resides on the server. If desired, you can later retrieve an entire message by double clicking on the header message icon or by selecting a command from the menu or toolbar that needs the entire message (such as reply or forward).
Always retrieve entire message	This is the default setting. Lets you retrieve all messages as you normally do. That is, in their entirety including attachments.
Do not apply rules to the Header Only messages	If selected, this option does not apply all rules to entire messages that you retrieve. This option applies rules only to the entire message and not the header only message.
Confirm before retrieving the entire message	If selected, displays a prompt stating that the message resides on the server and it asks you if you want to retrieve the message now. You can also choose not to display the prompt again by selecting the Don't display this prompt again option.

When a message belongs to another Z-Mail® Pro related application such as Phone Tag or Tag-It!, or is part of a routing slip, then the message is retrieved in its entirety.

Note: If you want to retrieve header only messages at a later date as entire messages, first change the option on the Smart POP tab to Always retrieve entire message, then choose the Refresh from Host command from the Message menu.

Using the Offline Option

Anytime you launch Z-Mail® Pro, after you have an established account, you will need to enter information into the Z-Mail® Pro Login dialog box. This dialog box prompts you for your username and password. It also provides you with the Offline option.

The Offline option disables the automatic timers for sending and receiving mail. If you select Offline, you will need to:

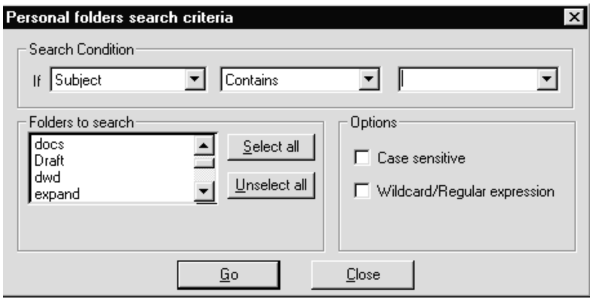
1. Manually retrieve your messages by selecting the Retrieve Messages option from the Mail menu.

2. Open Outbox and manually send all queued outbound messages.

Suppose you select one interface in Custom for Dial On Demand and want to work offline before actually sending or retrieving mail. Selecting the Offline option allows you to prepare a batch of e-mail for sending and it will not cause your dial-up connection to be initiated until you explicitly choose to send or retrieve mail.

Search

Search lets you search Z-Mail® Pro headers and mail messages for specific strings. A string can be a letter, word, or phrase. To search for a specific string in a mail header or message body, do the following:



1. Choose the Search... command from the Message menu. The Search dialog box appears.

2. In the Search Condition section enter the following:

- Left box: Select which field to search for the string: Subject, From field, To field, Sender or Message body.
- Middle box: Select the determining parameter: Contains, Does not contain, Equals to or Is not equal to.
- Right box: Enter the string match in the search.

3. In the Folders to search section, select the folder or folders you want searched. Choose the Select all button if you want all the folders searched.

   Note: On remote servers, only the selected mailbox is searched.

4. In the Options section, select Search Trash if you want the Trash folder automatically searched each time. Select Case sensitive if you want the string you are searching for to be in the exact case you entered. Select Wildcard/Regular Expression if a wildcard symbol is included in your search string.

5. Choose the Go button. A window containing a list of all messages that contain the stated search conditions appears. The date, subject and sender of each message are listed.

   Double click to view the selected message.

Creating Your Personal Address Book(s)

You can access the Address Book by choosing the Address Book... command from the Services menu in Z-Mail® Pro, or by choosing the Address Book... button located on the Z-Mail® Pro compose window. The Z-Mail® Pro compose window appears when you choose the Create... command from the Message menu.

Use the Address Book, which you access from the Services menu, to set up or create your Address Book. This dialog box lets you add individuals to your list, create new groups, and add members to new or existing groups.

Use the Address Book, which you access from the Z-Mail® Pro compose window (Create command), to add recipients to a message you are about to send. You can add new individuals to your list or add members to an existing group, but you *cannot* create a new group.

Adding Individuals to an Address Book

To add an individual to an Address Book:

1. Open the Address Book from the Services menu or the Z-Mail® Pro compose window and enter the user's name and e-mail address. For example,

```
John Doe <john@netmanage.com>
555-111@mcimail.com
71234.1234@compuserve.com
```

```
john@[156.27.10.107]
```

2. Choose the Add Name command from the Edit menu. The name appears on the list and is automatically added to the Address Book.

Adding Groups to an Address Book

To add a group to the Address Book, do the following:

Note: You can only add a group to the Address Book that you accessed from the Services menu.

1. Choose the Address Book... from the Services menu.
2. Open the Address Book you want to add the group to.
3. Type the group name and then check the Group option.
4. Choose the Add Name command from the Edit menu.
5. Select the name or names you want to add as a group members and choose the OK button. The selected names appear in the Group Members list. All group names appear in the Address Book dialog box in square brackets. For example, [*groupname*].
6. Select an individual name and choose the Members... command from the Edit menu to see if they belong to a group.

You may rename entries by selecting them and pressing the Rename button. Select one or more entries and press the Delete button to remove them from the Address Book. Addresses are automatically saved when you enter them.

Creating Additional Address Books

You can create a new address book or add additional address books that are stored locally. You can also allow access to an existing address book that resides on a remote host.

To create a new or add another address book:

1. Open the Address Book from either the Services menu or the Z-Mail® Pro compose window.
2. Select the Add Address Book... option from the Address Book menu. The Add Address Book dialog box appears.
3. Select the location of your address book: Local or Remote. For creating and accessing address books which are stored on a mainframe, please refer to the PROFS/OV Transport section.

4. Name: Enter the name of the new Address Book, this will appear in the Address book list.

5. File text box: Enter the Address Book's path and filename. You can enter the path directly or use the Browse button to browse for the selected drive and directory. For a new address book, you will be prompted to confirm you want to create a new file.

   Note: Your Address Book filename must use a .ADB extension. For example, the new Address Book called Joe could reside in: C:\NETMANAG\JOE.ADB.

6. Host (remote): Enter the name of the host in the Host text box. This is where the Address Book you want to access remotely is stored.

7. Choose the OK button.

You can open the remote Address Book by double clicking on its name in the listbox. The first time you open the book, the application connects to Directory Server, downloads all the addresses, and displays them in the listbox.

These addresses are cached in a local file so you can open the Address Book over and over without having to go through the network. However, if you want to update the Address Book contents from the network, then choose the Update Remote command from the Address Book menu.

Deleting Existing Address Books

You can also delete existing address books by selecting the address book you want to delete, and then selecting the Delete option from the Address Book menu.

ECCO Setup Option

The ECCO Setup option in the Address Book dialog is provided for the convenience of ECCO users. If you have the ECCO software package on your system, you can add e-mail addresses from your ECCO phonebook into one of your Z-Mail® Pro address books. You can choose to add all the e-mail addresses from your ECCO phonebook or only a selected set of addresses. Do the following:

1. Open the Address Book from the Services menu and choose the Add New Book... command from the Address Book menu.

2. Enter the name of your ECCO phonebook as you want it to appear in your Z-Mail® Pro Address Book listings in the Name text box. Enter the path and filename of the ECCO phonebook file in the File text box.

3. Choose the ECCO Setup...command from the Options menu. The ECCO Setup dialog box is displayed.

4. Enter the column name from your ECCO Address Book that contains e-mail addresses into the Field Name text box in the ECCO Setup dialog box. The default is E-Mail.

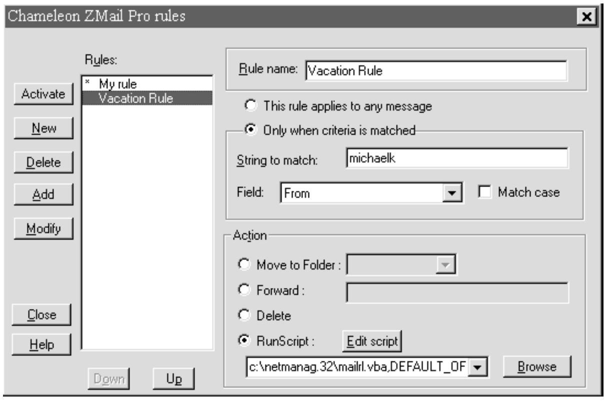
   Note: In releases prior to ECCO 3.0, the column name for e-mail addresses was an optional, user-specified name in the ECCO phonebook. Therefore, the name you enter in the Field Name text box depends on what you named the column in your ECCO phonebook. For example, if the ECCO phonebook column name was E-Mail, then you need to enter E-Mail in the Field Name text box.

5. In the Criteria drop-down list box, select All to add all the e-mail addresses from your ECCO Address Book to the Z-Mail® Pro Address book. Or scroll to select an alpha character to add only the addresses beginning with that character.

   For example, if you want to include only e-mail addresses beginning with the letter B, select the letter B from the list.

Setting Rules

You can associate attributes to your incoming messages by selecting the Rules... option from the Services menu. For example, you can create a rule so that all messages addressed to you are automatically moved to a specific folder. Or, create a rule so that all messages you receive from a particular person are automatically forwarded to another person.



The following steps show you how to create a rule where all messages whose chosen field contain a specific name or string are automatically moved to specific folder.

1. Enter the name of the rule in the Rule name: field. In this example, test2 is the name of the rule.

2. Enter the string (single word or several words) you want the system to search for in the String to match: field. In this example, the name Support is the string.

3. Select the field (From, To, Subject, or Sender) where the string is located. In this example, the name Support would have to appear in the To field.

4. Select the action you want executed when the matched string is found: Move to Folder, Forward, Delete or Run Script (see online help for information on running scripts). In this example, any message whose To field contains the name Support will be automatically moved to a folder named Urgent.

   If you want to select a folder to which incoming messages are automatically sent, the folder does not have to previously exist. You can define the folder when you make the rule.

5. Choose the Add button to add the new rule.

   The Rules list window, in the right of the dialog box displays all rules. Active rules are marked with an asterisk. Select a rule and chose deactivate or activate (the button toggles between the two states) to change the status of the rule. The Up and Down buttons allow you to change the positions of rules in the list. If you have more than one rule that contains the identical string to match, then the system will apply the first rule that is listed in the Rules list.

Using Command Line Options

You can use one or any possible combination of the following options to customize your Z-Mail® Pro login. The options are:

- **-u**{*username*} automatically enters the user's Mailbox name.
- **-p**{*password*} automatically enters the user's Mailbox password
- **-offline** enables the Offline option
- **-nologo** removes the splash screen (large Mailbox with copyright information) upon startup

To set one or more Z-Mail® Pro command line options, do the following:

1. Single-click (highlight) your mail icon. Z-Mail® Pro can be closed or opened.

2. From the Program Manager, choose the Properties... command from the File menu. The Program Item Properties dialog box appears.

3. Enter the option or options at the end of the Command Line text box.

   For example, to set up Z-Mail® Pro so that your username and password are entered automatically, thus bypassing the login box, enter the -p and -u options as follows:

   ```
   C:\NETMANAG\MAIL.EXE -UJack -Pmypassword
   ```

   where Jack is the user's mailbox name and mypassword is the user's (Jack) mailbox password.

Note: All options are case insensitive.

If you use the -p option your password will appear in the command line text box, thus making mail less secure.

Sorting the Date, From, Subject and Status Fields

You can sort by the Date, From and Subject fields while viewing the contents of a folder. The Date column is The From field is arranged The Subject field is also

1. Select the folder containing the messages you want to sort.
2. Double-click in the field above the column of :
   - Date: sorted by either the date the message was sent or received.
   - From: sorted alphabetically to allow you to view all messages sent from a particular sender.
   - Subject: sorted alphabetically to allow you to group messages sent about the same topic.
   - Message icons: sorted by Status

     Message Priority: All messages in the folder are sorted in order of High, Normal, and Low priority.

     Message State: Select this option to sort your messages in order by the following states: marked, header only, unread, and read.

     Message Type: Select this option to sort your messages in order of text messages only and then messages with attachments.

The arrow to the right of the Subject field determines how messages in each field are sorted:

Attaching a Signature

You can design a signature that is automatically attached to the end of each mail message you send out. To create a Z-Mail® Pro signature, do the following:

From within Z-Mail® Pro, select the Signature... option from the Settings menu. The Signature... dialog box appears.

1. Type in your own personal signature that you want to attach to outgoing mail.
2. If desired, select the Attach to outgoing mail box if you want it automatically attached to every outgoing message.

Changing Your Password

You can change your password at any time. To change your password, do the following:

1. Select the Password... option from the Settings menu. The Password dialog box appears.
2. Enter your current password in the Old Password field.
3. Enter your new password in the New Password field.
4. Enter your new password again in the Confirm Password field and choose the OK button.

Note: You can change the passwords for all users by logging in as Postmaster and accessing the Mailboxes dialog box.

Setting Timing Parameters

You can specify Retrieve timing parameters by selecting the Timers... option from the Settings menu.

Retrieve time values specify how often mail is retrieved from the mail server. Your Inbox queries the server periodically, as specified by this time value. You can also select the Retrieve Mail option from the Message menu to manually retrieve server mail, if mail is not automatically retrieved.

Using IMAP Transport

IMAP4 is used to manage messages on a remote host. IMAP provides access to multiple hierarchically structured mailboxes on the same or different remote servers and to shared mailboxes between users. Only headers are retrieved, which avoids unnecessary retrieval of large attachments and saves bandwidth. When a message is viewed, it is fetched and stored online in the local cache for the duration of the mail session.

Configuring IMAP

You may access multiple remote servers, or add numerous occurrences of the same server with different user names. Additional servers are added through the Network Settings dialog. Refer to the previous section Setting Up a New Z-Mail® Pro account for a User, for detailed information on the Network Settings Dialog.

Each server you add appears as a separate branch on the Remote tree displayed in the left pane of Z-Mail® Pro’s main window. Messages can be moved between mailboxes on the SAME server (same user tree), or to local folders.

Shared Mailboxes

IMAP allows access to shared mailboxes on the remote host. Messages can be accessed by a number of users. Messages are not deleted from the shared mailbox unless they are explicitly deleted by the Delete command.

Mailboxes are shared in one of two ways, depending on the type of server being used.

- Cyrus types of servers: When you log in, all shared mailboxes are displayed, and private mailboxes are in a sub-folder of the users inbox. The system administrator, by modifying permissions, is responsible for restricting the access to private mailboxes.
- Washington university types of servers: When you log in, only the user's private inbox and tree of private folders. To share mailboxes on this system, additional mailboxes must be added through the IMAP4 tab of Network Settings. Enter the path o the other mailboxes you need to see into the shared mailbox textbox. The mailboxes are displayed in accordance with the permissions the login user has been allocated.

Using Z-Mail® Pro Fax

Z-Mail® Pro, used in conjunction with the separate Chameleon Fax Server application lets you create, send, receive, and view faxes. When creating a message within Z-Mail® Pro, you can send a message to both mail and/or fax addresses. Working with fax messages and attachments is basically the same as working with mail messages.

Faxes are sent to a Fax Server computer for translation into a faxable format before being sent. The Fax Server translates the fax message into TIFF or DCX images. The Fax Server is a mail client with unique capabilities designed specifically for dealing with fax messages. For information about the Chameleon Fax Server, refer to the Chameleon Fax Server guide.

Note: Before proceeding, confirm that the Fax Server Administrator has authorized you as a licensed Z-Mail® Pro Fax user.

Configuring Z-Mail® Pro Fax

To configure Z-Mail® Pro Fax, choose the Internet Fax tab from the Network Settings command and enter the Fax Server's e-mail address (for example, faxserver@hostserver). You can add multiple fax servers, each with its own name. If uncertain, contact your Fax Administrator.

Folders are used in the same way as Z-Mail® Pro folders, except for fax messages, the special folder defined will be displayed as black, by default. Fax folders can be designed to automatically store any incoming fax messages, outgoing faxes, fax attachments and notifications according to the specifications set.

Creating/Sending a Fax Message

You can both send and receive faxes, directly from Z-Mail® Pro. The same fax message can also be sent as mail to an e-mail address.

To create and send a fax message:

1. From Z-Mail® Pro, choose the Create...command from the Message menu.
2. Fill in the From, Subject, and Confirm information as requested. You may also enter a comment, if desired.
3. Choose the Internet Fax transport in the Transports drop down list and type in the fax phone number.

The message will now be marked as a fax. When sent, the message will be routed to the Fax Server of your choice. Once at the Fax Server, the message and any attachments are converted to TIFF or DCX images then transmitted to the fax address provided. Notification will be sent when the fax arrives at its destination. If the fax fails, for whatever reason, the sender also receives notification.

Sending Attachments with a Fax

Files created and saved in any application that is supported under Windows (and installed on the Fax Server) can be attached to the fax message. The recipient of the fax message can then view and print the message and attached document as a TIFF or DCX image.

To attach a file to fax messages:

1. Choose the Attach button.
2. A dialog similar to the Open File dialog is displayed. Choose or enter the file name.
3. Choose the OK button. The attached file will now be seen in the mail list and will be sent along with the fax message to the designated fax address.

Notification

Whenever a fax has been transmitted, successfully or unsuccessfully , a notification message is sent to the originator of the fax stating the fax status.

Receiving and Viewing an Incoming Fax

Incoming faxes are very similar to incoming mail except that fax messages and their attachments are images that are converted from a text file or data to an image file such as a TIF or DCX (depending on the fax board). A fax message can only be manipulated as text if an Optical Character Recognition (OCR) utility is available. Incoming faxes are listed as such in the incoming mail display.

To view a fax select the fax message from within Z-Mail® Pro. You can also use NEWTView to view the fax. Before viewing a fax message, save the fax image to a directory of your choice. Then associate the viewer with either TIFF or DCX file type to display the message.

Using PROFS/OV Transport

Z-Mail® Pro provides electronic mail services between your PC and IBM hosts using PROFS/OV. The PROFS/OV communications run over a 3270 Terminal session, using HLLAPI.

PROFS/OV is an electronic mail transport that allows your PC to communicate with an IBM mainframe, using the IBM internal mailing system (PROFS or OfficeVision/VM). This universal Mail client allows easier migration from a Mainframe-only environment to a LAN-based environment, as follows:

- You can send a mail message to both Internet and PROFS/OV recipients at the same time.
- You can receive mail in your Inbox folder from both Internet and PROFS/OV senders.

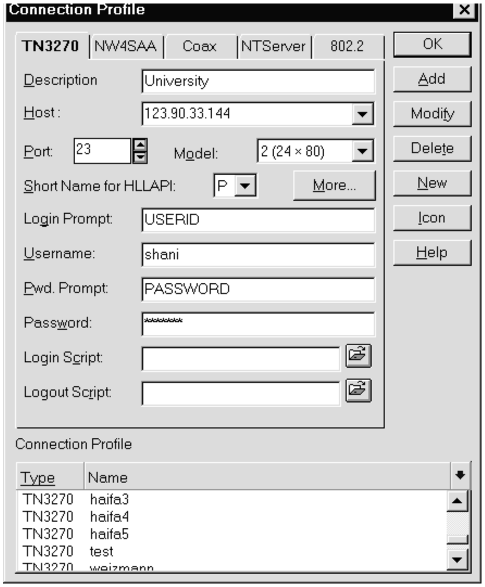
Configuring PROFS/OV Mail

Before starting PROFS/OV for the first time, you must set up information for the underlying 3270 Terminal session and Z-Mail® Pro's connection to that session, as follows:

1. Create and save a TN3270 Connection Profile, then exit the TN3270 application.
2. Open the Z-Mail® Pro application and choose Mainframe Mail from the Z-Mail® Pro Transports Setup dialog.
3. Specify the appropriate information in the Mainframe Mail dialog box of Network Settings.

Note: After you have specified these settings, Z-Mail® Pro will automatically invoke the TN3270 Connection Profile when you choose to create PROFS or OV mail. Z-Mail® Pro will also automatically close the connection when it is exited.

Setting Up the TN3270 Connection Profile



The TN3270 Connection Profile should contain the following information:

Field	**Description**
Description	Specify the name of the connection profile. Later you will specify this same name in the Network Settings of the Z-Mail® Pro.
Host	Specify the name or IP address of the IBM host to which you want to connect.
Port	Specify port 23.
Model	Specify Model 2.
Short Name for HLLAPI	To fill in the short-name box, select a letter from the drop-down list. You cannot simply click the cursor in the box and type the letter. Optional.
Login Prompt	The login prompt that you would see on your screen. Should be used if automatic login is desired. Case-sensitive.
Username	The user name that you use to login. Should be used if automatic login is desired.
Pwd Prompt	The host password prompt that you would see on your screen. Should be used if automatic login includes password. Case-sensitive.

Field	Description
Password	The password that you normally enter after you have entered your username. Component of automatic login with password.
Icon	You can save the current Connect Profile description as an icon. This is unnecessary for use with Mainframe Mail because the TN3270 profile is invoked by Z-Mail® Pro automatically.
Login Script	You can type or browse to select the name of a script file that automatically logs into the host when you use connect to a host by using this profile.
Logout Script	You can type or browse to select the name of an existing script file that automatically logs out of the host specified by this profile when you close the application. This is unnecessary for use with Mainframe Mail because Z-Mail® Pro automatically logs out of the TN3270 session.
Connection profile list	The list box contains the connection profiles that you have already defined.

Prompt for Automatic Login

If you want to include the Login Prompt and the Password Prompt as part of your Connection Profile, you should connect and log in to the host and write down the prompts that are displayed for username and password. The specified text string must match the prompts (or a portion of the prompts) as they are displayed. They are case-sensitive.

Getting to the Ready Prompt

To set up the underlying TN3270 session for Mainframe Mail, you must create a profile that gets you to the host's Ready prompt. Therefore, your profile must either contain a login script that brings you automatically to that point, or the following components of automatic login: login prompt, username, pwd prompt, and password.

Note: See TN3270 online help for additional information.

Create Mail

If only the PROFS/OV settings are enabled, mail is automatically sent via the PROFS/OV transport. If the SMTP Network settings are also enabled, the New Mail dialog box contains a drop-down menu to select PROFS/OV or SMTP:

After selecting PROFS/OV, type the address you require and press ENTER. Note that when the address appears in the field below, `<@PROFS:>` is automatically added to the address. (If the SMTP is selected, pressing ENTER copies the address to the field below, without any additions.)

If the PROFS/OV address is written on the mainframe as: `(userid)host,` type the mail address field as `userid@host`. After pressing ENTER, it appears in the field below as

`<@PROFS:userid@host>` (or as `<@OV:userid@host>`).

Optimize Sending of Large Messages

You can optimize the way your message is sent to the mainframe according to your personal usage:

1. Choose the Setting -> Network Setting dialog. Select the PROFS/OV property sheet.
2. Choose the Advanced button.
3. Select the `Upload Mode` parameter. Its possible values are:

- `TEXT` - upload message line-by-line. This is suitable for short messages, possibly containing dot commands.
- `FILE` (or `.GF`) - message file is transferred to the mainframe and inserted to PROFS/OV in a screen-by-screen mode. When sending large messages this mode is faster that the previous one. It supports dot commands.
- `.IM` - message file is transferred to the mainframe and inserted to PROFS/OV at once. This is the fastest method, but any dot commands in the message will not be processed by PROFS/OV.

4. Choose OK.

Downloading Mainframe Address Books

PROFS/OV can access and download directories from the host’s mainframe to your local system. To update an existing address book with the mainframe address books (nicknames and Distribution Lists), choose Update Remote from the Address book option. You can create a new address book to contain these addresses:

1. Choose Address book from the Services menu.
2. Choose Add New Book... option from the Address book menu.
3. Select PROFS or OV.
4. Enter the name you want the address book to be called.

   Note: This can be any name you choose **-** it does not have to be the name of an address book on the mainframe.

5. If you want to download a specific file, type its mainframe file name (i.e. <file name> <extension> <disk>) in the Host text box.
6. Choose OK.
7. Choose the Update Remote command from the menu Address Book., or double click on the Address Book's name.

Restricting Address Book Download

The default address book download will bring all the nicknames and distribution lists from the mainframe, regardless of their location (i.e. including other disk you have access to). In large enterprises this operation might take a long time and overflow the capacity of the Z-Mail® Pro Address Book.

If you want to restrict the scope of download to your local disk, do the following:

1. Post the Setting **-**> Network Setting dialog. Select the PROFS/OV property sheet.
2. Choose the Advanced button.
3. Select the `ADB Retrieve Locally` parameter and set it to `YES`.
4. Choose OK.

News

You can configure news servers through the news tab of network settings. For details on using this option, refer to the Forum chapter.

Rapid Search

Choose Rapid Search from the Services menu to activate the Rapid Search application. Refer to the Rapid Search chapter and online help for more information.

Mail Alias Viewer

Choose Mail Alias Viewer from the Services menu to activate the Mail Alias Viewer application. Refer to the Mail Alias Viewer chapter and online help for more information.

Using the Outbox

Anytime you send a message it first goes to the Outbox and then is delivered to the mail gateway, if defined. This includes all messages sent using Z-Mail® Pro, Tag-It!, Phone Tag, and messages you post using Forum and NEWTCal.

The Outbox attempts to deliver a message as soon as it receives it. Messages that remain in the Outbox past a specified amount of time expire and Z-Mail® Pro messages are returned to the sender as undeliverable. If one of the messaging applications (such as Z-Mail® Pro, NEWTCal, and Forum) was invoked in an Offline mode, the Outbox would accumulate the sent messages until an explicit Send is performed.

Opening the Outbox

You can start Outbox as you do other applications or you can choose the Outbox command from the Services menu.

Using the Messages Menu

The Messages Menu in Outbox lets you view, send, and delete messages in the Outbox. Simply select the desired message and then select either the View, Send, Delete, or Abort options.

You can also use the Pause Sending Message option to stop messages in the Outbox from being sent. This is especially useful if you are using a dialup connection.

For example, before you disconnect you can select this option and the Outbox will not try to send messages. Then, when you reconnect you send your messages again by deselecting this option.

Use the Check Unmatched Message Files... option from the message menu when sent messages fail to arrive to the Outbox. This option manually places the sent messages into the Outbox list.

Z-Mail® Pro Administration

The Z-Mail® Pro Administration application allows you to manage your Z-Mail® Pro database. To use Z-Mail® Pro Administration, do the following:

1. Quit the Z-Mail® Pro application if it is open.
2. Start Z-Mail® Pro Administration. The Z-Mail® Pro Mailbox Administration-Login dialog box appears.
3. Enter the username and password and choose the OK button. The Z-Mail® Pro Administration window appears. There are five options available:
   - Merge two mailboxes
   - Compact mailbox
   - Generate mailbox from message files
   - Remove unmatched entries
   - Rebuild folder information
4. Select one of the five options and choose the Start button.

Merge Two Mailboxes

Select this option to merge the messages from any mailbox into the current mailbox. Messages in the source mailbox remain unchanged. All messages are copied into the current user's mail directory, and are added to the current mailbox. The mailboxes (user's .DAT files), in the email/user directory, must have the same name.

Compact Mailbox

Select this to reduce the mail database file size by eliminating empty space in the Mailbox that occurs when you have deleted numerous messages.

Generate Mailbox from Message Files

Select the Generate mailbox from message files to generate or create a new database.

Remove Unmatched Entries

Select the Remove unmatched entries option to remove entries in the mail database that do not have actual message files associated with them.

Rebuild Folder Information

Select the Build Administration folder information option to have the Z-Mail® Pro Administration rebuild the folder hierarchy from the files and directories on the disk. Choosing this will automatically be followed by a Generate to rebuild the mail database.

Note: Z-Mail® Pro Administration renames the folders, based on the names of the directories on the disk. If desired, you may rename your Z-Mail® Pro folders later. However, this should only be necessary for folder names that could not be used as a DOS directory name.

Additional Online Information

The online help file for Z-Mail® Pro includes a description of each menu command, plus the following information in the "How to Use" section:

- Creating and Sending Mail Procedures
- Creating and Using an Address Book
- Defining a Z-Mail® Pro server and Gateway
- Displaying Mail Messages
- Faxing a Message
- IMAP Transport of Mail
- PROFS/OV Transport of Mail
- Saving Changes to a Configuration File
- Setting Up Z-Mail® Pro
- Using Command Line Options
- Using the Outbox
- Viewing and Editing HTML Documents

Note: For more information about Z-Mail® Pro Administration, Outbox, and the address book, refer to the application's online help.

Troubleshooting

If you experience difficulties using Z-Mail® Pro, refer to the following items:

General Information:

❑ If you are having trouble starting Z-Mail® Pro, make sure the path in your mail.cfg file correctly points to your mailbox directory.

❑ If you cannot open your mailbox because you forgot your password, then edit the MAIL.CFG file and remove the encrypted password next to your name. For example, if John=fg78%$, then fg78%$ is the encrypted password.

❑ Use the Ping application to verify connectivity to the domain portion of any destination address if Z-Mail® Pro reports a failure to deliver. Verify connectivity to your mail server is defined.

- ❑ Use the logging feature (form the Settings menu) to help diagnose client transmission and mail server retrieval problems. Check replies for connection errors or destination address errors.
- ❑ Make sure the user name portion of the destination address is defined at the destination. The Postmaster may define new users by using the Mailboxes item under the Services menu.

Internet Mail Information:

- ❑ If you are having trouble retrieving Internet mail, make sure that the remote system supports the SMTP protocol. Verify that the POP2 or POP3 daemons are running on the mail server system. The POP2 known port is 109 and the POP3 port is 110. To verify if the daemons are running, use the Telnet application to connect to ports 109 or 110. The system replies with a message indicating if the correct daemons are running on ports 109 or 110.
- ❑ If you have trouble sending Internet mail, verify that SMTP is available on the remote host, and you have specified the correct mail gateway address under Internet Mail Network. Also verify that the To: field (destination address) is complete. For example, steven@netmanage.com.
- ❑ For POP2 only, you also need to verify that the mail directory settings are correct. Make sure that your password and mail path (/user/spool/mail/username) are correct. Contact your system administrator for more information.

Mainframe Mail Information:

- ❑ Make sure a TN3270 profile has been defined and saved.
- ❑ Make sure that the TN3270 profile specifies port 23, model 2, and brings you to the host Ready prompt. See TN3270 Profile for additional information.
- ❑ View the TN3270 window to verify that it is at the Ready prompt.
- ❑ Check the Initialization command specified for Mainframe Mail Network Settings: it has to link the PROFS/OV mini-disks and set PROFS or OV/VM to line mode.
- ❑ PROFS/OV in line mode will generate the following sequence of requests:

1. Addressee
2. Subject
3. Message Body

Outbox Information:

- ❑ If you are having trouble sending mail, make sure that the remote system supports the SMTP protocol.
- ❑ If you have trouble sending mail, verify that SMTP is available on the remote host, and you have specified the correct mail gateway address under Network settings.

- ❏ Use the Ping application to verify connectivity to the domain portion of any destination address if Outbox reports a failure to send.

Mail Administration Information:

- ❏ Back up the user's e-mail directory and then do a Generate or Rebuild.

Chapter 16. Indexing Agent and Rapid Search

Indexing Agent

The Indexing Agent automatically tracks and indexes all text you have retrieved. It is used with Rapid Search to provide full-text index creation and rebuilding for the WebSurfer and Z-Mail® Pro applications.

The Indexing Agent must be running before you can open the Rapid Search application. Also, the Indexing Agent must run continuously so that Web pages can be indexed. To ensure that the Indexing Agent is always running, the Chameleon setup program places it in the Program Manager startup group. The Indexing Agent will not work unless it is minimized.

Note: Additional Indexing Agent functions are available through the Control Menu. To access the Control Menu, click once on the minimized Indexing Agent icon to display the list of commands available.

Rapid Search

Rapid Search finds all text indexed by the Indexing Agent in both the WebSurfer and Z-Mail® Pro applications. All indexing is performed as a background task and requires no action by the user.

In WebSurfer, all text from all Web pages you have viewed will be indexed. In Z-Mail® Pro, all text from all messages in your Z-Mail® Pro database will be indexed, whether or not you have read them. You can start Rapid Search from the Rapid Search icon, or directly from within WebSurfer and Z-Mail® Pro.

To start a search, do the following:

From WebSurfer, choose the Rapid Search command from the Tools menu. From Z-Mail® Pro, choose the Rapid Search command from the Services menu.

The Rapid Search dialog box is displayed.

Enter information in the fields that are described in the following table:

Field	Description
Search For	Enter the text that you want to find.

Field	Description
In Title Only	Search for text that appears only in the title of a Web home page or the Subject field of a Z-Mail® Pro message.
Only Include Items Box	
Web	Search for text that appears only on Web pages.
Mail	Search for text that appears only in Z-Mail® Pro messages.

Note: Select both check boxes to search for text that appears on Web pages and in Z-Mail® Pro messages.

Field	Description
From/Site	Enter the name of the message's author to search for in Z-Mail® Pro messages. Enter the name of the Web site (for example, netmanage.com) to search for in WebSurfer.
To/Cc	Enter the name of the recipient to search for in Z-Mail® Pro messages. This field does not apply to Web sites.
Word List	Shows the list of words you want to search for.
Folder/Date/From/Site/ Subject Box	Shows results of a completed search. For Web pages: • Folder column shows the location of the stored text (for example, from your hot list) on your local drive. • Date column shows the date the search was conducted. • Site column shows the Web site name. • Subject column shows the text found. For Z-Mail® Pro messages: • Folder column shows the location of the text found (for example, in your Inbox) on your local drive. • Date column shows the date the message was created. • From column shows the name of the person who created the message. • Subject column shows the topic of the message.

When you have filled out the fields required, choose the Find button or choose the Find command from the Edit menu to begin the search.

Note: You need to type text in at least one field in the Rapid Search dialog box to conduct a valid search. You can also type text in more than one field to do multiple searches.

Additional Online Information

The online help files for Rapid Search and Indexing Agent include a description of each menu command.

Troubleshooting

If you experience difficulties using Rapid Search or Indexing Agent, refer to the following items:

- ❏ If you cannot open Rapid Search, make sure that the Indexing Agent is already open. It can only run minimized.
- ❏ Make sure that you have enough memory and disk space to run both Rapid Search and Indexing Agent at the same time.
- ❏ Close and reopen Rapid Search if you are receiving frequent error messages.
- ❏ Rebuild your index by choosing the Re-create Index command from the Control menu if your system starts to run very slowly or crashes.

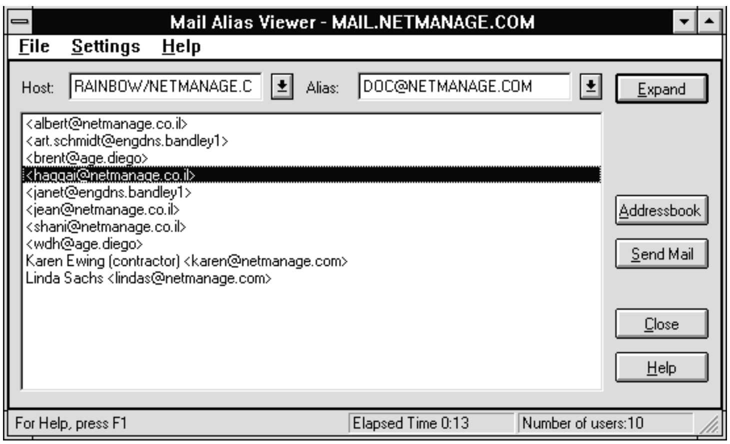
Chapter 17. Mail Alias Viewer

Mail Alias Viewer provides you with a list of all mail addresses listed under an alias. For example, Mail Alias Viewer will list all the mail addresses that are included when you send mail to the alias, salesteam@netmanage.com.

Using Mail Alias Viewer

To use Mail Alias Viewer do the following:

1. Open the Mail Alias Viewer application or Alias Viewer from the Services menu in Z-Mail® Pro.



2. Enter the following information:

   - Host: the name of the Host where the alias list for the group you want to expand is stored. (Ask the system administrator of this mail gateway for the name of the Host).
   - Alias: the name of the Alias you want to expand.

3. Choose the Expand button.

   All addressees under this alias are listed.

4. Select one of the email addresses. It appears in the status bar.

5. Select one of the following options:

- Addressbook: Saves the address to the address book you select.
- Send Mail: Brings up the edit message dialog, with the address automatically placed in the from field.
- Abort: The Expand button toggles to Abort, when the expand process is in process. Choose Abort to terminate the process, before completion.

Personal Icon Database

To view Personal icons associated with an email addressee, select the personal icon library location:

1. Select Personal icon database from the Settings menu.
2. Select the appropriate drive and directory where the icon files are stored.
3. Choose the OK button.
4. The personal icon of the addressee is displayed, when the address is selected.

Additional Online Information

The online help file for Mail Alias Viewer includes a description of each menu command and a "How to Use" section.

Troubleshooting

If you experience difficulties using the Mail Alias Viewer, refer to the following items:

- ❏ Make sure the host name to which you are trying to connect is correctly entered.
- ❏ If Mail Alias Viewer reports a failure to connect, use the Ping application to verify the connection to the host.

Chapter 18. Mailbox Converter

The Mailbox Converter is used to convert messages stored in various Mail systems into the Z-Mail® Pro message base. This conversion process needs to be run only once, to convert accumulated messages from a different Mail system into Z-Mail® Pro. You may also run the Mailbox Converter from a command line.

How to Use the Mailbox Converter

To use the Mailbox Converter, close your messaging applications and Forum, then complete the following steps:

1. Start the Mailbox Converter application.

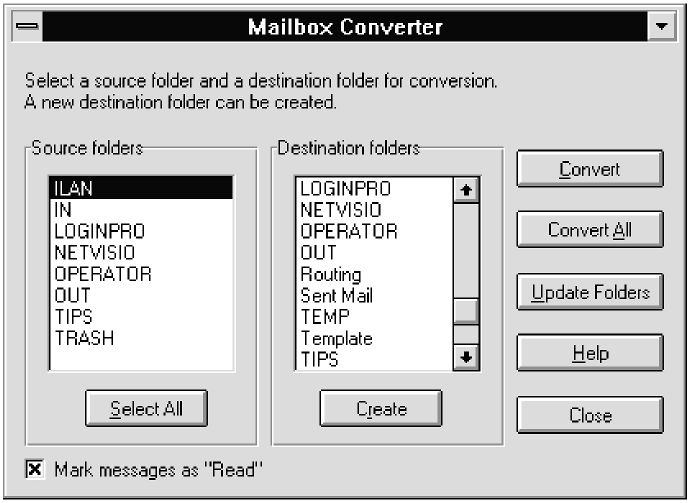
2. At the prompt, Z-Mail® Pro Login, enter the username and password as you use when you login to the Z-Mail® Pro messaging applications .

Note: If any messaging applications or Forum are already running, close them at the prompt.

3. Select the type of source Mail system you want to convert and enter any specific information that the source Mail system requires:

   - cc:Mail: Enter the VIM Login required when you run cc:Mail.
   - Eudora: Enter the directory where Eudora Mailboxes (files) are located or the Mailbox name.
   - PROFS/OV: Not necessary to enter any details. The necessary information is retrieved from Mail's settings.
   - UNIX: Enter the directory where the UNIX Mailboxes (files) are located or the Mailbox name.
   - Z-Mail® Pro: Select the Z-Mail® Pro username for the mailbox you want to convert. Select the Convert Aliases check box to have all your aliases converted into a Chameleon ATX mailbox.

4. The Mailbox Converter dialog appears.



5. Select one or more source folders or choose the Select All button if all folders are required.

   Source folders are the Mail folders containing the Mail in the original format that you want to convert into Z-Mail® Pro.

   When converting Eudora Mail, PROFS/OV or UNIX, select Mark messages as Read check box for all Mail to be displayed as read. If the check box is not selected, all Mail will be displayed as unread.

   When converting cc:Mail, the source folders list contains only the Inbox folder.

6. Select the destination folder where you want the selected mail to reside after conversion.

   If you require an additional folder select the Create button and type the folder name in the New Folder dialog.

   Selecting the Convert All button overrides the destination folder selection.

7. Choose the Convert button, to convert the Mail in the Source folders and place into the destination folder.

   Or choose Convert All, to convert all the Source folders into Z-Mail® Pro folders with the same name as the original source folder.

   In case of a duplicate name (same name for source and destination folder), the Folder already exists dialog appears.

8. Repeat this procedure, as necessary, for converting other folders.

9. Choose the Update Folders button. Your Mailbox is regenerated to include the converted files.

10. Choose the close button. If you did not chose to update folders earlier, you will be prompted to do so now.

Folder already exists!

This dialog appears when the folder you are trying to create already exists as a Z-Mail® Pro folder. Choose one of the following options:

- Append messages to existing folder

  The messages in the selected source folder will be converted and placed into the existing Z-Mail® Pro folder of the same name.

- Always Append messages to existing folders

  The messages in the selected source folder will be converted and placed into the existing Z-Mail® Pro folder of the same name. This action will automatically be taken for any future source folders with the same name as destination folders. This message will not appear again.

- Skip this folder

  The selected folder will not be converted.

- Create a new Z-Mail® Pro folder

  Enter the name you require for a new folder. This folder will be created and the Mail for the selected folder converted and placed here.

- Select another existing folder

  Select which folder you want the Mail from the selected source folder to be placed in after conversion.

Running Mailbox Converter from a Command line

Do the following to run Mailbox Converter from a command line:

1. Select Run... from the File menu in the Program Manager.

2. Enter the application name and path, such as

   C:\netmanag\converter.exe

3. Add the appropriate source system parameter:

   -s source system, such as PROFS, ccmail, Eudora, UNIX

   Note: this parameter must be entered

4. Add any other required source system parameters:

-d	Eudora mailbox directory
-f	Eudora or UNIX mailbox file
-mp	Z-Mail® Pro password
-mu	Z-Mail® Pro user
-p	password in cc:Mail
-u	username in cc:Mail system

Note: -d and -f parameters can not be used together.

Additional Online Information

The online help file for Mailbox Converter includes a description of each menu command and a "How to Use" section.

Troubleshooting

If you experience difficulties using the Mailbox Converter, refer to the following items:

- ❑ Ensure that you are already defined as a Z-Mail® Pro user, before running Mailbox Converter.
- ❑ Ensure a PROFS/OV transport has been defined in Z-Mail® Pro before converting Mainframe folders
- ❑ Before running Mailbox Converter, close all messaging applications (Z-Mail® Pro, Phone Tag, Tag-it!, NEWTCal and Forum), otherwise the "Mail is already running" dialog appears.
- ❑ If the first dialog displays garbage entries, check that mailcvt.cfg exists in your netmanag directory. Its items should be of the type:
  <source name>=<existing DLL name>.

  If there is a problem, please contact NetManage Tech Support for a valid file.
- ❑ If "Missing DLL" error message appears after selecting cc:Mail, you must install cc:Mail on your machine, and its directory should be in your path. Furthermore, if running on Windows NT or Windows 95, the 32 bit version of cc:Mail should be installed.

Chapter 19. Directory Server

The Directory Server lets you keep a company-wide address book, or an ECCO phone book available so that other users can easily access it. The Directory Server can also be accessed as a regular WhoIs server through NetManage's WhoIs client application.

You can place any regular address book that has an .ADB extension or an ECCO phonebook that has a .CEO extension on the Directory Server. However, only one file (one address or phone book) can be made available on the server at a time.

Launching the Directory Server

You can launch Directory Server manually, or you can launch it through the Internet Services Daemon (INETD). To launch it through INETD, refer to the NetManage Configuration Guide for more information.

Note: You cannot launch the Directory Server using INETD if you are using Windows NT.

To launch it manually, choose the Directory Server icon to display the Directory Server dialog box.

Configuring the Directory Server

The Configure command in the Settings menu lets you select the address book file you want to make available to users. You can select a regular address book file (.ADB), or an ECCO file (.CEO).

For example, suppose you have two files: one is a regular address book called NetManage, with the filename C:\NETMANAG\EMAIL\JANE\NETMAN.ADB, and the other is an ECCO-based phone book with the filename C:\ECCO\NETMAN.ECO. You can choose which of these files you want to make available on your server.

To place an address book on the Directory Server, do the following:

1. Start Directory Server.
2. Choose the Configure... command from the Settings menu. The Configure dialog box appears.
3. Under Data File Type, select the Address Book (.ADB) option.
4. Enter the complete path of the address book you want to make available in the Filename text box. Choose the Browse button to search for the address book's correct path if you do not already know what it is.
5. Choose the OK button.

To place an ECCO-based phone book on the Directory Server, do the following:

1. Start Directory Server.
2. Choose the Configure... command from the Settings menu. The Configure dialog box appears.
3. Under Data File Type, select the ECCO (.CEO) option.
4. In the Filename field, enter the complete path of the ECCO phonebook, which you want to make available on the server. Choose the Browse button to search for the phonebook's correct path if you do not already know what it is.
5. In the Email Name field, enter the name of the field in the ECCO phone book that specifies the e-mail address. The default is Email.
6. Choose the OK button.

Additional Online Information

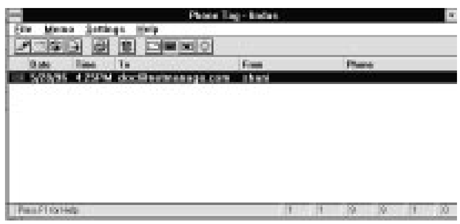
The online help file for Directory Server includes a description of each menu command, plus the following information in the "How to Use" section:

- Launching the Directory Server
- Configuring the Directory Server
- Manually Setting Up an Address Book File

Chapter 20. Phone Tag

The Phone Tag application provides an easy-to-use method of recording and sending phone memos. It allows you to create phone memos from your PC and send them directly to people, without having to write the memo down on a piece of paper and physically handing that person the memo. This application is also helpful for telephone operators and other employees who take a lot of phone memos for other people.

You can set Phone Tag (through the Preferences menu) so that a Phone Tag memo someone sends you automatically pops up on top of any other Windows application you are using.



Phone Tag provides a template for your mail memo, so you can simply type in the memo, and you do not have to worry about typing the other extraneous information (for example, date time and so on). This information automatically appears in the template when you bring up the phone memo template.

Opening Phone Tag

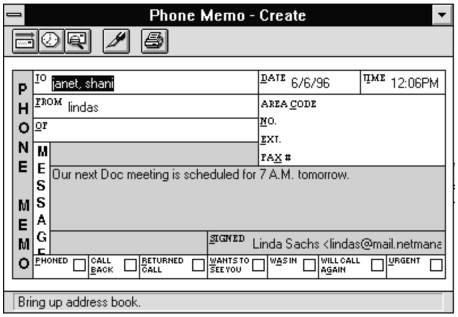
You do not have to log in to use Phone Tag if you are already logged into Z-Mail® Pro, Tag-It!, Forum, or NEWTCal. All five applications share the same login ID. Note, however, that you must have a Z-Mail® Pro account and have Z-Mail® Pro running to receive Phone Tag memos. You also can send a Phone Tag memo to someone who does not have the Phone Tag application. This person would receive the Phone Tag memo as a regular e-mail message.

Phone Tag uses the same login information and gateway settings used in Z-Mail® Pro. Set up the username, password, gateway, and server through the Z-Mail® Pro application (using Postmaster to add a user). Refer to the chapter titled, Z-Mail® Pro and Z-Mail® Pro Administration, in this manual for additional information.

Creating Memos

To create a memo, do the following:

1. Choose the Create... command from the Memo menu. The Phone Memo-Create template appears. The following toolbar icons appear on the top of this template: Send the memo, Date and time stamp outgoing memo, Bring up Address book, Create a memo, and print a memo.



2. Enter information into the following fields:

   - **To:** The e-mail address of the person to whom you are sending the memo. This field must contain a valid mail address or alias recognized by Chameleon Z-Mail® Pro. To send to more than one user or mail alias, separate them with commas.
   - **From:** The caller. This is the person leaving the phone memo.
   - **Of**: The company name of the person listed in the From field.
   - **Date:** The date the memo is sent. This field is automatically stamped. It can be overwritten with current time by checking the Time Stamp button.
   - **Time:** The time the memo is sent. This field is automatically stamped. It can be overwritten with current time by checking the Time Stamp button.

Note: The Date and Time stamp must conform to the international date and time setup under the Control Panel.

- **Area Code:** Area code of the caller.
- **No.:** Phone number of the caller.
- **Ext.:** Phone extension of the caller.
- **FAX#:** Fax number of the caller.
- **Message:** Contains a message from the caller.
- **Signed:** The name of the person (operator) who is recording the phone memo. This field is automatically filled in with the sender's default From field set in Z-Mail® Pro.

The following fields may be checked to indicate what the caller wants to convey in the message:

Phoned, Call Back, Returned Call, Wants To See You, Was In, Will Call Again, Urgent.

Sending Memos

To send a memo after you enter all information into the phone memo, select the Send Memo toolbar icon or press CTRL+S to send the memo.

Note: The To, From, and Signed fields are required. The To field must be a valid e-mail address or alias recognized by Chameleon Z-Mail® Pro. The Signed filed is automatically signed in with the name set as default in the Mail from field.

Forwarding Memos

You can forward a Phone Tag memo by selecting the memo you want to forward and selecting the Forward option from the Memo menu.

Deleting Memos

You can delete a Phone Tag memo by selecting the memo you want to delete from the main Phone Tag window and selecting the Delete option from the Memo menu.

Viewing Memos

You can view a Phone Tag memo by selecting the memo you want to view from the main Phone Tag window and selecting the View option from the Memo menu.

Note: You can scroll through a Phone Tag memo you are viewing by using the up and down arrow buttons.

Using the Settings Menu

The Settings menu allows you to set preferences, fonts, colors, and use the logging feature. Refer to Phone Tag online help for detailed information about options in the Settings menu.

Additional Online Information

The online help file for Phone Tag includes a description of each menu command, plus the following information in the "How to Use" section:

- Opening Phone Tag
- Creating Memos

- Sending Memos
- Forwarding Memos
- Deleting Memos
- Viewing Memos

Troubleshooting

If you experience difficulties using the Phone Tag application, refer to the following items:

❑ If you receive Phone Tag memos as regular e-mail, ensure that the directory in which NetManage is installed is in the DOS path and that PHONETAG.EXE is installed.

❑ If you are not receiving Phone Tag memos, make sure Z-Mail® Pro is running and retrieving messages. Have the person sending the Phone Tag send a regular e-mail message to the same address. If the regular e-mail message is not received, see the Troubleshooting section in the Z-Mail® Pro application for more information.

Chapter 21. Tag-It!

The Tag-It! application lets you send informal messages to other people. The Tag-It! window looks similar to an online version of a Post-It™ note. The messages are displayed as yellow popup notes. If you want to create longer and more formal messages, you should use the Z-Mail® Pro application.

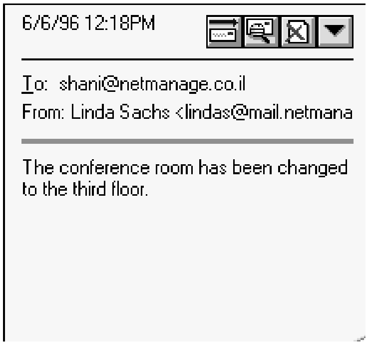
You log into Tag-It! the same way you do in Phone Tag, Z-Mail® Pro, NEWTCal and Forum because Tag-It! uses the Z-Mail® Pro SMTP send mechanism. Tag-It! messages you receive are automatically opened by Z-Mail® Pro.

Note: Unlike Z-Mail® Pro and Phone Tag messages, you cannot save or print Tag-It! messages.

Sending Tag-It! Messages

You can easily send a Tag-It! message by taking the following steps:

1. Start Tag-It! to display a Tag-It! window.



   The window may include a title bar and provides access to the Control menu if you have specified Tag-It! to do so (as discussed in "Setting Preferences," later in this chapter). If not, you can display the title bar and make the Control menu accessible by double-clicking the window or by pressing ESC.

   Note: To access the Control menu, you should first make the title bar available by double-clicking on the Tag-It! window or by pressing the ESC key. Then press ALT+SPACEBAR to display the Control Menu or click on the small box in the upper left corner of the window (the Windows Control Menu icon).

The window also includes toolbar buttons for the Send, Address, Close, and Minimize functions.

1. Specify to whom you want to send your message. To do so, perform either of the following operations:

   - Type the e-mail address of each user in the To: field. Separate addresses with a comma (,).
   - Display the Address Book either by choosing the Address button from the toolbar, by choosing the Address command from the Control menu, or by pressing CTRL+A. After Tag-It! opens the Address Book, select a name and then choose the To button. Continue selecting names and choosing the To button until you specify all the users to whom you want to send the message. Then choose the OK button.

   The Address Book lists the addresses of the users included in the Z-Mail® Pro Address Book. For additional information, see the chapter on Z-Mail® Pro in this manual.

1. Type your message in the window.
2. Either choose the Send button on the toolbar or choose the Send command from the Control menu or press CTRL+S.

Forwarding Tag-It! Messages

After Z-Mail® Pro opens a Tag-It! message sent to you by another Tag-It! user, you can forward that message to other users. To do so, take the following steps:

1. Choose the Forward button from the toolbar or choose the Send command from the Control Menu or press CTRL+F.

   The To: field is cleared, the From: field automatically changes to your login username, and the Forward button becomes the Send button.

2. Specify to whom you want to send your message. To do so, perform either of the following operations:

   - Type the e-mail address of each user in the To: field. Separate addresses with a comma.

- Either choose the Address button from the toolbar or choose the Address command from the Control menu. After Tag-It! opens the Address Book, select a name and then choose the To button. Continue selecting names and choosing the To button until you specify all the users to whom you want to send the message. Then, choose the OK button.

3. If you want to, type additional information in the message you are forwarding.

4. Either choose the Send button on the toolbar or choose the Send command from the Control menu, or press CTRL+S.

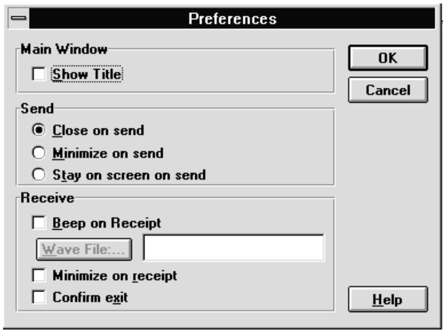
Setting Preferences

To set up preferences for Tag-It!, take the following steps:

1. Start Tag-It! to open a Tag-It! window.

2. Choose the Preferences option from the Control menu.

Note: To access the Control menu, you should first make the title bar available by double-clicking on the Tag-It! window or by pressing ESC. Then press ALT+SPACEBAR to display the Control Menu or click on the small box in the upper left corner of the window (the Windows Control Menu icon).



1. Configure Tag-It! by performing one or more of the following operations:

- Select the Show Title check box if you want Tag-It! to always display the title bar and make the Control menu available upon startup.
- Specify whether Tag-It! remains running after a message is sent.
  - To close Tag-It! once a message is sent, select the Close on send option.
  - To continue running Tag-It! but minimize the window, select the Minimize on send option.
  - To continue running Tag-It! and leave the window open, select the Stay on screen on send option.

- Select the Minimize on receipt option if you want any Tag-It! messages that you receive to be displayed as minimized icons on your desktop.
- Select the Beep on receipt option if you want Tag-It! to emit a sound when you receive a Tag-It! message. If you specify a .WAV file, the specified sound file will be played. If you do not specify a .WAV file, a beep will be emitted.
- Select the Confirm exit option if you want Tag-It! to display a prompt verifying whether you want to exit Tag-It! This option is selected by default.

Note: If this option is not selected, you may unintentionally press ALT+F4 and thereby close the Tag-It! application without having a chance to read a note that you have received. Therefore, it is recommended that the Confirm on exit option be selected.

2. Choose the OK button.

Additional Online Information

The online help file for Tag-It! includes a description of each menu command.

Troubleshooting

If you receive Tag-It! messages as regular mail messages (instead of Tag-It! messages), refer to the following items:

❑ Ensure that the Chameleon Z-Mail® Pro application is activated (can be minimized).

❑ Ensure that tagit.exe is present in your system.

❑ Ensure that the directory containing the Tag-It! application is in the DOS Path.

❑ If the above items do not apply, Tag-It! may not be able to start for some other reason, such as insufficient memory. When Tag-It! is not available, any Tag-It! messages received by you will be available in Z-Mail® Pro instead.

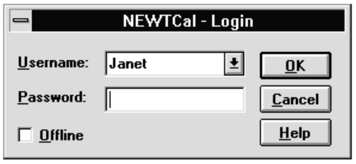
Chapter 22. NEWTCal

The NEWTCal application lets you schedule appointments in an online calendar. When you schedule an appointment, NEWTCal notifies people about the proposed meeting. You can then use NEWTCal to track responses to your appointment requests. You can also view your online calendar in a daily, weekly, monthly, or yearly format. NEWTCal lets you schedule new appointments and modify or delete existing appointments.

Login

When you open the NEWTCal application, the NetManage NEWTCal Login dialog box appears unless you have already logged into Z-Mail® Pro (or one of the related Z-Mail® Pro applications, Tag-It! or Phone Tag).

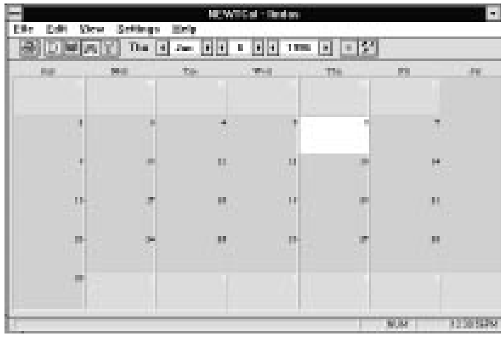
1. To log in to NEWTCal, type your username and password in the appropriate fields. (This is the same username and password that you use for Z-Mail® Pro.)



After the login has been completed, NEWTCal displays your online appointment calendar.

Viewing Your Appointment Calendar

You can view your online appointment calendar in a daily, weekly, monthly, or yearly format. To do so, select the appropriate option (Daily View, Weekly View, Monthly View, or Yearly View) from the View menu.



Setting NEWTCal Preferences

NEWTCal lets you set the number of hours your daily calendar includes. It also gives you the option of printing your calendar with or without appointment information.

Scheduling Appointments

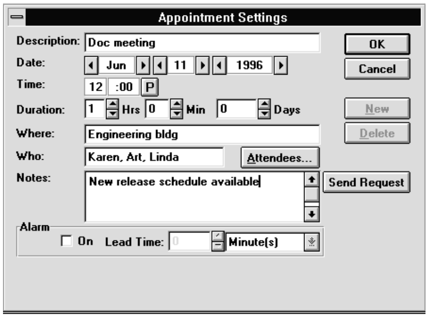
Set up Z-Mail® Pro before scheduling appointments because NEWTCal uses Z-Mail® Pro to notify people of proposed meetings. Your mail application must be running (it can be minimized) for you to send and receive notifications of appointments.

Scheduling New Appointments

To schedule new appointments, take the following steps:

1. Perform one of the following operations to open the Appointment Settings dialog box:

    - If your calendar is in a weekly, monthly, or yearly format, double-click the day you want to schedule the appointment.
    - If your calendar is in a daily format, double-click the hour you want to schedule the appointment.



2. Enter the appointment's description, date, time, length and location in the appropriate fields.

   In the Date, Time, and Duration fields, you can enter information by either typing it or choosing the up or down buttons.

3. If desired, type additional information about the appointment in the Notes field.

4. Either type addresses in the Who field or choose the Attendees... button to open the Address Book.

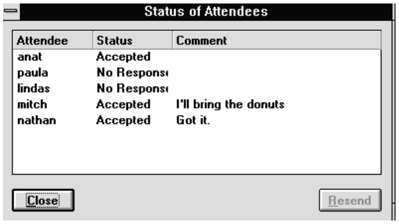
   a) From the Address Book list, select an e-mail address, and then choose the To button.

   b) Continue selecting addresses in the Attendee list box and choosing the To button until the Attendee List box includes all the desired addresses.

   Note: The Address Book is a shared feature of Z-Mail® Pro, NEWTCal, Phone Tag, and Tag-It! It is accessible from all these applications. To add an e-mail address to the Address Book, use the Z-Mail® Pro application's Services Menu, or choose the ECCO Setup option within the Address Book dialog.

5. Notify people of a proposed appointment by choosing the Send Request button from the Appointment Setting dialog box.

   The Z-Mail® Pro application notifies attendees about the appointment as discussed in "Responding to Appointment Requests" later in this chapter. If you want to view the responses to your appointment request, choose the Show Status button. A Status of Attendees dialog box appears.



6. When you finish scheduling appointments, close the Appointment Settings dialog box by choosing the OK button.

Modifying Existing Appointments

To modify existing appointments, take the following steps:

1. In your online appointment calendar, double-click the appointment you want to modify. Or click once on the appointment in the calendar display and choose the Edit Menu/Edit Appointment option. The Appointment Settings dialog box, containing information for the selected appointment appears.

2. Modify the fields of the Appointment Settings dialog box as necessary. (See also "Rescheduling Tips.") Then choose the OK button.

NEWTCal automatically notifies the attendees of the changes you have made to the appointment time and date.

Rescheduling Tips

When modifying appointments, you can quickly reschedule them by moving them (using drag/drop) from one location to another in your online calendar:

- To reschedule appointments to another day, select the appointment you want to reschedule (to/from another day on the display or to/from the busybar) and move it to the new day.
- To increase the time of an appointment, stretch the block on the busy-bar.
- To reschedule appointments to another time on the same day, display your online calendar in the daily format as discussed in "Viewing Your Appointment Calendar." Then select the time of the appointment you want to reschedule and move it to the new time.

Deleting Existing Appointments

To delete existing appointments, take the following steps:

1. In your online appointment calendar, double-click the appointment you want to delete. The Appointment Settings dialog box, containing information for the selected appointment appears.

2. Choose the Delete button. Or click once on the appointment in the calendar display and choose the Edit Menu/Delete Current option.

NEWTCal automatically notifies the attendees that you have deleted the appointment information.

Responding to Appointment Requests

If another NEWTCal user sends you an appointment request, a dialog box with information about the meeting is displayed on your computer terminal.

The Meeting Request dialog box contains all the relevant information pertaining to the meeting.

To respond to an appointment request, do the following:

1. Enter optional comments in the Comment box. These comments will be delivered with your response.

2. Choose one of the following response buttons:

   - **Accept**: NEWTCal will automatically add the appointment to the appropriate date and time in your calendar, and Z-Mail® Pro will send a notification of your acceptance to the user who sent the Meeting Request.
   - **Decline**: Z-Mail® Pro will send a notification of your decline to the user who sent the Meeting Request.
   - **Defer**: NEWTCal will hold the Meeting Request in its buffer, so that you can make a decision later. To display the Meeting Requests that you have deferred, use the Get Pending Requests option from the View Menu.
   - **Discard**: The Meeting Request will be discarded.

Additional Online Information

The online help file for NEWTCal includes a description of each menu command.

Chapter 23. InPerson

The InPerson desktop conferencing software lets you have an online working session (conference) with people on a network.

During a working session, each conference member can sketch drawings, type notes, paste captured images, and import text and image files into the InPerson whiteboard.

The information included in an InPerson whiteboard can be viewed and modified by all members. For example, suppose you want your co-workers to review your design for a new gadget. You can take a screen shot of your design and then paste it into your InPerson whiteboard where everyone can examine it and make comments or changes to it.

InPerson can also provide audio and video support so that members can talk to and view one another during a conference.

This chapter

- discusses video and audio support requirements
- describes the InPerson window
- provides an overview of using InPerson

Video Support Requirements

The following items are required if you plan to send video images to other conference members. However, you will be able to receive video images from other members without these items.

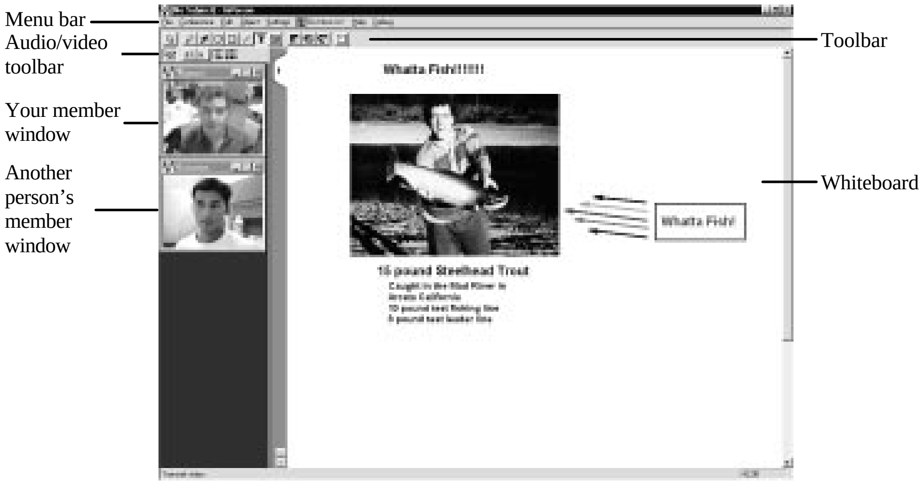
- video capture card that can support real-time capture in preview mode (as opposed to overlay)
- a video camera compatible with your video capture card
- Video for Windows 1.1 or later software

Audio Support Requirements

InPerson's audio feature requires any Windows compatible sound card.

About the InPerson Window

The InPerson window looks similar to the following figure:

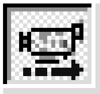
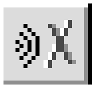
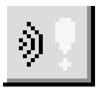
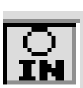


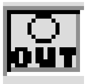
InPerson displays a member window for each participant. A member window image can be either a video image or a default image file that you specify as discussed in the online help. The name of a member is displayed in the title bar of that person's member window.

The whiteboard is the area in which you enter text and draw objects. The whiteboard consists of one or more pages. Each whiteboard page is marked by a numbered tab. If there are multiple whiteboard pages, the tab of the current page is white and the tabs of the other pages are gray. To turn to a whiteboard page, select its tab. You can also add or delete whiteboard pages by choosing the appropriate command from the Edit menu.

The InPerson menu bar and toolbar are similar to the menu bars and toolbars in other Windows applications. For details on the menu bar and toolbar, see your InPerson online help.

The following table describes the various buttons that are available on the audio/video toolbar. The table also indicates hardware and software requirements you need for the functionality of these buttons to be supported.

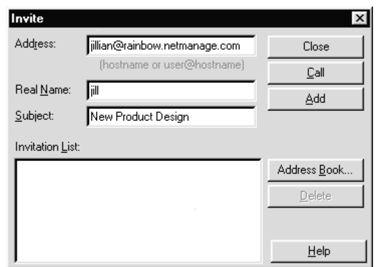
Button	Requirements	Description
 Send Image	For video images: • Video capture card • Compatible video camera • Video for Windows 1.1 or later software For default images: none	Depress this button to transmit the image displayed in your member window. Release this button if you do not want to send other members the image in your member window.
 Mute	Windows compatible sound card	Depress this button if you do not want other members to be able to hear you. You will be able to hear other members however.
 Talk	Windows compatible sound card **Note:** The audio/video toolbar does not include the Talk button while InPerson is operating in full-duplex mode.	Depress this button if you want to lock the recording mode. You can then talk to other members without being interrupted. Depressing this button is equivalent to pressing the F9 key or holding down the F8 key. If you want to return to listening to the other members, release the Talk button or press the F9 key. This is equivalent to releasing the F8 key if you are holding it down.
 In	None	When this light is on, it indicates that you are receiving sound.

Button	Requirements	Description
 Out	Windows compatible sound card	When this light is on, it indicates that you are recording or sending your recording to other users. **Note:** Other users will be able to hear you only if your Mute button is released.

Inviting Conference Members

You can invite a person or a group of people to an InPerson conference by doing the following:

1. Choose the Invite... command from the Conference menu. An Invite dialog box similar to the following appears.



2. Specify who you want to invite by adding names to the Invitation List box. To add names, do the following:

   - If the InPerson address book already includes the name of the person or group you want to invite, choose the Address Book... button. In the Address Book dialog box, double-click each name or group you want to add to the Invitation List box. When you finish adding all the names or groups you want to invite, choose the Close button.

     You can store the names and addresses of frequently invited members in the InPerson address book. For information about how to make entries in the address book, see the Adding New Addresses to the Address Book section.

   - To invite someone not included in the InPerson address book, do the following:

     b) Type the person's InPerson address in the Address text box.

        InPerson addresses can be in any of the following formats: user@hostname, hostname, user@IPaddress, or IPaddress. When inviting SGI InPerson users, be sure to use the format user@host or user@IPaddress. For example, to specify user Jillian on host rainbow, type jillian@rainbow or jillian@156.27.50.200.

c) Type the name you want to appear in the title bar of the person's member window. This step is optional. If you leave the Real Name text box blank, the address entered in the Address text box will appear in the member window.

   Suppose you enter jillian@rainbow in the Address text box and Jill in the Real Name text box. Jill rather than jillian@rainbow will appear in the title bar of Jillian's member window.

d) Choose the Add button.

3. If you want to provide members with a brief description of the conference, type one in the Subject field.

4. Choose the Call button.

   Users are prompted to accept or decline your invitation unless they have selected the Auto Accept Incoming Call check box as discussed in the online help.

   Note: Users can receive an invitation only if they are currently running InPerson. Therefore, any users planning to frequently participate in InPerson sessions should add InPerson to their Windows Startup group.

The InPerson conference will continue until all members disconnect from the session. However, the conference will end for all members if the person who established the conference disconnects.

Defining the InPerson Address Book

The InPerson address book contains a list of names and addresses. Define the InPerson address book if you want to be able to invite people by selecting names rather than typing information in the Invite dialog box.

When defining address book entries, you can

- add addresses to the address book
- create address book groups
- change address book entries
- delete address book entries

Separate discussions on each task follow.

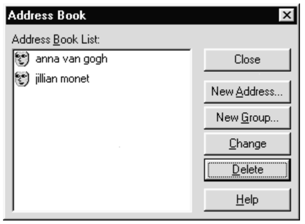
Adding New Addresses to the Address Book

To add addresses to the address book, do the following:

1. Choose the Invite... command from the Conference menu.

2. In the Invite dialog box that appears, choose the Address Book... button.

   The Address Book dialog box appears.



3. Choose the New Address... button. The New Address dialog box appears. In the Address text box, type the InPerson address of the person you want to add.

   InPerson addresses can be in any of the following formats: user@hostname, hostname, user@IPaddress, or IPaddress. When inviting SGI InPerson users, be sure to use the format user@host or user@IPaddress. For example, to specify user Jillian on host rainbow, type jillian@rainbow or jillian@156.27.50.200.

4. If you want the address book entry to include the person's actual name, type that name in the Real Name text box. This step is optional. If you leave the Real Name text box blank, the address entered in the Address text box will appear in the member window.

   Suppose you enter jillian@rainbow in the Address text box and Jill in the Real Name text box. Jill rather than jillian@rainbow will appear in the title bar of Jillian's member window.

5. Choose the OK button.

   InPerson adds the person to the Address Book List box.

6. You can continue adding names to the address book by repeating steps 3 through 5.

7. When you are done adding addresses to the address book, choose the Close button

Creating an Address Book Group

If you frequently invite the same group of people to conferences, you may want to create a group for those users. An address book group consists of two or more address entries previously added to the InPerson address book. You can send an invitation to all members of the group by simply selecting the group name rather than having to select each individual's name.

To create an address book group, do the following:

1. Choose the Invite... command from the Conference menu.

   The Invite dialog box appears.

2. Choose the Address Book... button.

   The Address Book dialog box appears.

3. If you have not already done so, add an address book entry for each person you want to include in the group. For details, see the Adding New Addresses to the Address Book section.

4. Choose the New Group... button.

   The New Group dialog box appears.

5. In the Address Book List box, select the names of each person you want to include in the group.

6. In the Group text box, type a group name.

7. Choose the OK button.

   InPerson adds the group to the Address Book dialog box.

8. To close the Address Book dialog box, choose the Close button.

Changing Address Book Entries

To change address book entries, do the following:

1. Choose the Invite... command from the Conference menu. The Invite dialog box appears.

2. Choose the Address Book... button.

3. In the Address Book dialog box, select the name or group you want to change.

4. Choose the Change button.

   If you selected an individual’s name, the Change Address dialog box appears.

If you selected a group, the Change Group dialog box appears.

5. Do one of the following:

   - If you are modifying an individual's entry, edit the text boxes of the Change Address dialog box as necessary.

     If the entry you modify is included in any groups, you will also need to change those groups to reflect your change. Suppose you change the entry Jillian to Jill. You will need to make sure that any groups that include Jillian list her as Jill instead.

   - If you are modifying a group's entry, select each name you want to include in the group. Unselected names will not be included in the group.

     You can make multiple selections using the CTRL and SHIFT keys as discussed in your Windows documentation.

6. Choose the OK button.

7. To close the Address Book dialog box, choose the Close button.

Deleting Address Book Entries

To delete an address book entry, do the following:

1. Choose the Invite... command from the Conference menu. The Invite dialog box appears.

2. Choose the Address Book... button to open the Address Book dialog box.

3. Select each name or group you want to delete from the Address Book List box. If you want to select more than one name or group, hold down the CTRL key while making your selections.

4. Choose the Delete button.

5. When InPerson prompts you to confirm you want to delete the selected entry, choose the Yes button.

6. To close the Address Book dialog box, choose the Close button.

Conversing with Other Members

How you carry on InPerson conversations depends on whether the application is in half-duplex mode or full-duplex mode.

Full-duplex cards are bi-directional and can record and play sounds at the same time. This lets you talk and listen at the same time.

Unlike full-duplex cards, half-duplex cards are unidirectional and are unable to record and play at the same time. When you are in half-duplex mode, InPerson can record your conversation if it is not playing another member's recording. You cannot talk and listen at the same time if InPerson is in the half-duplex mode.

If you are in the half-duplex mode and want to talk to other members without being interrupted, depress the Talk button. To return to listening to the other members, release the Talk button.

In either the full-duplex or half-duplex mode, other users will be able to hear you if the Mute button is released. This is the default. If you want to listen to other users but do not want them to be able to hear you, depress the Mute button.

Drawing on the Whiteboard

The whiteboard is designed so you can easily draw simple shapes. InPerson provides five drawing tools – a tool for drawing freehand and tools for drawing arrows, lines, circles, and boxes. You draw on the whiteboard by selecting one of the drawing tools and a pen color and width.

To select a drawing tool, choose the appropriate button from the toolbar or choose the Pen, Arrow, Circle, Line, or Rectangle command from the Object menu. When you select a drawing tool, the cursor changes to reflect the selected tool. For example, the cursor changes to a circle when the circle tool is selected.

If you choose the pen tool (the default), you can draw freehand. If you select one of the other drawing tools, draw the selected shape by dragging the mouse until the arrow, circle, line, or box is the desired size.

You can select an object's color using the Pen Color command. You can also select the width of an object's lines by using the Pen Width command. For details on these commands, see the InPerson online Help.

You can find out who created an object on the whiteboard. To do so, move the cursor to the object you want to examine. The name of the person who placed the object on the whiteboard page is highlighted in that person's member window.

InPerson automatically assigns a different cursor symbol to each conference member. This helps you identify who is currently working on the whiteboard. When a person works on the whiteboard, the cursor on your computer will look like that person's cursor symbol.

Adding Text to the Whiteboard

You can add text to a whiteboard page at any time. To do so, choose the text button from the toolbar and then begin typing in the whiteboard.

When adding text to the whiteboard, you can perform the following operations:

- Start a new line of text by pressing the Enter key.

- Delete characters by pressing the Backspace key.
- Paste text copied from another application by choosing Paste from the Edit menu. When you paste text from another application, tab settings and font characteristics are lost.

Additional Online Information

The online help file for InPerson includes a description of each menu command, plus the following information in the "How to Use" section:

- Adding New Addresses to the Address Book
- Changing Address Book Entries
- Conversing with Other Members
- Creating an Address Book Group
- Deleting Address Book Entries
- Inviting Conference Members
- Moving Objects
- Selecting Objects
- Setting Preferences

Troubleshooting

The following list includes troubleshooting information for InPerson:

- ❑ If you cannot hear other members but you can talk to them, you may be locked in recording mode. (The Out light is on while you are recording.) If you are in half-duplex mode, unlock the recording mode by either releasing the Talk button or pressing the F9 key. This is equivalent to releasing the F8 key if you are holding it down.
- ❑ If other members cannot hear you and you selected GSM voice compression, verify whether the other members are using version 5.0 or earlier of the NetManage InPerson application.

  You can use GSM voice compression only if all members are using either the Silicon Graphics InPerson application or version 6.0 of the NetManage InPerson application. If one or more members are using version 5.0 or earlier of the NetManage InPerson program, select either IDVI8 or IDVI16 voice compression as discussed in your online help.
- ❑ If you cannot establish a connection to conference members, ensure that all the invited conference members are in the correct format and make sure that all invited members are currently running InPerson.

- ❏ If you can work in the whiteboard but cannot send or receive audio or video, use the Ping application to verify that all members’ machines are able to resolve each others host names.
- ❏ If the video images you receive are not coming in well, adjust the video quality as discussed in your online help.
- ❏ If the audio you receive is not coming in well, adjust the audio quality as discussed in your online help.

Chapter 24. WebSurfer

WebSurfer allows you to browse multimedia information on the World Wide Web (WWW). WebSurfer displays hypertext documents and supports multimedia such as sound, video, and interactive graphics.

About the World Wide Web

The WWW is a set of servers on the Internet that are that are linked. Hypertext is a method of presenting information that lets you jump easily from one topic to another or to an expanded base of information on that topic.

WebSurfer Features

WebSurfer supports the following features:

- Netscape standard web frames, which enable multiple HTML view spaces
- Rapid Search, which lets you search previously visited pages using specific search criteria by means of the Indexing Agent
- Microsoft video tags, which provide interoperability between Microsoft and NetManage video services
- AVI streaming, which provides the ability to view audio and video in a continuous, uninterrupted manner
- stationary background tags, a Microsoft extension, which provide support for non-scrolling backgrounds
- Microsoft background sounds extension, which is similar to background images
- Microsoft font tag support, which is used to change fonts within a text range
- Microsoft cookies extensions, which provides for HTTP-oriented, client-specific Web page customization (This is defined by each Web site individually.). Cookies actually contain a sub-domain ID from a specific server and include a cache of tags from this server that correspond to certain information. This information is saved in a cookies.txt file.
- Netscape support for full-screen or embedded plug-ins
- formatted documents display (electronic text in a variety of fonts, bold and italic, bulleted lists, extended characters sets)
- Direct access to NetManage Tech Support using the Support Wizard
- hypertext-linked documents browsing

- links to FTP, Gopher, Telnet and other NetManage applications using the Go IntraNet! feature
- hypermedia document display, including interactive graphics, sounds, and movies
- creation of hot list folders for organizing documents
- access to other tools without closing WebSurfer
- context-sensitive menus using the right mouse button
- caching of information downloaded to your local drive from the Web
- tracking of links of your search through Web documents, and quick backtracking using the History list
- TrueSpeech and RealAudio embedded third-party applications: the TrueSpeech Player and Encoder lets you bring real-time, streaming audio playback over the Internet; the RealAudio Player lets you bring live and on-demand audio to the desktop over the Internet using RealAudiolinks
- Chaco VRML external viewer for 3D graphics embedded plug-in (works on Windows 95 only)
- Adobe Acrobat Reader embedded plug-in so you can read and print documents in PDF (portable document file) format
- Macromedia's Shockwave for Director plug-in, which provides multimedia in a Web page on the Internet (works on Windows 95 only)
- external editors and viewers, such as WebSpider, NEWTView, and NEWTSound, so you can view text, image, and video files, and listen to sound files

About Uniform Resource Locators (URLs)

Uniform Resource Locators (URLs) let you identify where a resource resides anywhere on the Internet by linking Web documents on HTTP servers. If you know the document’s URL, you can retrieve it. URLs provide information about resource type, location, and path.

Resource types indicate the type of server on which the document resides. WebSurfer supports the following resource types:

- http:// is the HTTP Server
- ftp:// is the FTP Server
- file:// is the local HTML file
- telnet:// is the Telnet Server
- gopher:// is the Gopher Server
- mailto:// is an e-mail address
- news:// is a Usenet Newsgroup

Note: For more information about using resource types other than HTTP, refer to WebSurfer online help.

For example, the NetManage home page document title is NetManage Home Page and its URL is:

`http://www.netmanage.com:80/netmanage/index.html`

where:

http is the resource type (HTTP Server)

//www.netmanage.com:80 is the resource location

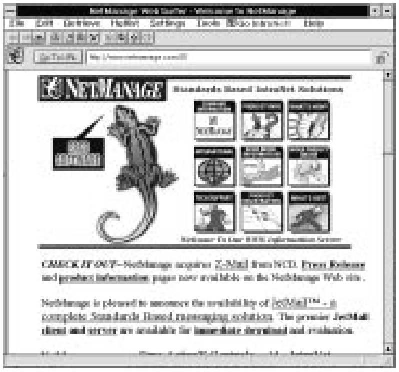
/netmanage/index.html is the path and filename

Setting Preferences in WebSurfer

You can set preferences in WebSurfer to indicate your selections for the startup document, network software, document viewers, proxy server, document retrieval behavior, and plug-ins. Preferences are grouped within six tabs: General, Network, Viewers, Proxies, Cache, and Hotlist.

Using WebSurfer

When you start WebSurfer, the NetManage home page is displayed. The interface consists of the document window and the menus and controls surrounding the document.



The Title bar at the top of the window displays the document title. The Menu bar displays all the menu names, with each menu containing drop down lists of commands. The Toolbar contains buttons that serve as shortcuts for WebSurfer menu items.

Below the Toolbar is the Dialog bar, which displays the URL address of the current document. As you move the mouse around the document, the Status bar displays the URLs for any anchors or links over which the mouse passes.

A document can contain text and pictures, some of which are hypertext links to other multimedia documents or objects. The links are highlighted in blue. You can change the document format (that is, how it appears on your screen) by using the Style Schemes command from the Settings menu.

To gather information about the document you are currently viewing, choose the Properties command from the Retrieve menu. The Properties dialog box is displayed, and shows information such as: title, URL, file type, and the local file where the document is located.

You can browse through the document and choose the highlighted text to link to another document. WebSurfer downloads and displays the new document, including any graphics that may be a part of it. To return to the previous page, choose the Back command from the Retrieve menu or the Back button on the toolbar. To return to the home page, choose the Home command from the Retrieve menu or the Home button on the toolbar.

Opening Documents in WebSurfer

There are two methods of opening documents in WebSurfer:

- using its URL
- using an existing local .HTM file from the local file system

Opening Documents Using URLs

While navigating through a Web document, you can open another document by:

- single-clicking the left mouse button on the highlighted text
- choosing the Go To URL command from the Retrieve menu, then entering a document's URL
- entering a document's URL in the Dialog bar

All three methods will retrieve the chosen document from the network or local cache (local drive) and display it for you.

Opening Existing Local HTML Files

You can open existing local .HTM files from your local file system. This is useful if you want to edit your own .HTM files. You can open either an unformatted or formatted version of the file.

To open an unformatted version of a local .HTM file, choose the Source command and then the Text Editor option from the Edit menu. To edit a document, you can also choose the Properties command from the Retrieve menu and choose either the Edit HTML command or the View/Play button in the Properties dialog box.

Both commands display the .HTM file in the default text view if the file is a text file. It displays the file with the default image viewer if it is an image file, and plays the file with the default player if it is a sound or movie file.

After you have edited and saved the file, you can view its changes immediately by choosing the Reload From Disk command from the Retrieve menu.

To open a formatted version of a local .HTM file, choose the Source command, then the HTML Editor option from the Edit menu. The formatted file is displayed in the HTML editor you have designated. WebSpider is the default HTML editor for WebSurfer.

Caching Documents

A major feature that WebSurfer provides is the ability to cache documents or files. Caching keeps a copy of documents that you have opened, in your local cache (local drive). When you reopen a previously opened document, you do not have to wait while the information downloads again from the network.

Default Caching

Initially when you retrieve a document from the Web, it is retrieved directly from the network and automatically placed in your local cache for the current session. If you want to reuse a document you have already retrieved during the current session, WebSurfer will retrieve it for you automatically from your local cache.

However, when you close the current session, the documents in your local cache are not saved. If you want to revisit a document you opened during a previous session, then the document is automatically retrieved from the network and once again placed in your local cache.

Default caching behavior for documents is established using the Preferences command (global behavior of all documents) and the Properties command (individual behavior of a specific document). Note that the settings for the caching behavior of a document in the Properties command overrides those set in the Preferences command.

Overriding Default Caching

There are several ways you can override default caching of documents:

- Choose the Reload From Network command from the Retrieve menu to automatically refresh a document retrieved from your local cache with its latest version from the network.
- To retrieve documents directly from the network and not your local cache, regardless of how many times you re-visit the same document in the current session, deselect the Reuse Retrieved Documents Within Session check box located in both the Properties and Preferences dialog boxes.
- To save documents to your local drive between sessions, choose the Save Cached Documents Between Sessions check box, located in both the Properties and Preferences dialog boxes. Use this setting to retrieve the same document from your local drive between sessions.

Using the Hotlist

The Hotlist contains documents (Web pages) that you find particularly interesting and want to access regularly without having to search for them within the Web. You can display the Hotlist by doing one of the following:

- choosing the Show Window command from the Hotlist menu
- selecting the Hotlist button in the toolbar
- positioning the mouse cursor on the left-hand border of the WebSurfer window and dragging the border to the right.

The hot list appears in the left pane of the WebSurfer window, while the selected document is displayed in the right pane.

The Hotlist feature lets you add items, create folders for organizing items, display folder properties, import Mosaic or Netscape hot list information, and merge existing hot list items into your current hot list.

You can add items to your hot list by doing one of the following:

- choosing the Add Item command from the Hotlist menu
- placing the cursor on the highlighted link or image, then clicking the right mouse button and dragging it to a folder in your hot list

You can also split the Hot list screen by choosing the Split Hotlist button on the toolbar or the Split Hotlist command from the Hotlist menu. Splitting the hot list lets you better organize the items in your hot lists and move items easily from one hot list to another.

Launching Other Applications in WebSurfer

You can now launch other Chameleon applications directly from the WebSurfer Tools menu instead of Program Manager. All applications other than Forum will appear in separate windows. Choose the Customize command from the Tools menu, then select the application you want to start. The application selected automatically opens in a new window while WebSurfer is still open.

Plug-Ins

WebSurfer now supports embedded third-party applications, called plug-ins. You can start these applications directly from within WebSurfer. Plug-ins include the following:

- Acrobat Reader
- TrueSpeech Player/Encoder
- RealAudio Player
- Chaco VRML 3D Viewer
- Shockwave Multimedia Viewer/Player

To access these applications, choose the Preferences command from the Settings menu, then choose the Viewers tab. WebSurfer opens a new window when it runs a plug-in application.

Additional Online Information

The online help file for WebSurfer includes a description of each menu command, plus the following information in the "How to Use" section:

- Accessing Other Resources
- Caching Documents
- Importing NCSA MOSAIC.INI and Netscape File Information
- Launching Forum and Other Applications
- Modifying Document Styles
- Printing Documents
- Using Rapid Search
- Using Short Cut Menus
- Using the Hotlist
- Using Web Frames
- Using WebSurfer
- Viewing a History of Documents
- Viewing Files in WebSurfer

Troubleshooting

If you experience difficulties using the WebSurfer application, refer to the following items:

❑ If you get a message, "Failed to launch viewer for file *.bmp, *.tif and so on, check file associations; check for viewer;" do the following:

a) Go to File Manager.

b) Choose the Associate command from the File Menu.

c) For the file type: bmp, tif, and so on., select the viewers using the Browse button.

You can also add, modify or delete viewers automatically from the Settings menu, by choosing the Preferences command, then the Viewers tab.

❑ NEWTSound and NEWTView are NetManage sound and viewer applications provided for your use. Other example viewers are PAINTBRUSH.EXE and WINGIF.EXE.

❑ When you get messages such as "Unable to resolve the hostname," check for the following:

- Can you ping the hostname?
- If you have upgraded to the current version, make sure that you are not using the old configuration file and that the corresponding entry for this application is removed from the TCPIP section of the WIN.INI file.
- Check whether the correct configuration file has been opened in Custom, and all the settings are correct.

Note: The previous item refers to users running the Windows 3.1 or Windows 95 operating system only.

❑ If you are not retrieving any images in the documents, see whether the Defer Images command from the Settings menu is enabled. Disabling it will help you to retrieve the images automatically.

❑ If you experience difficulties in retrieving the updated documents for example, Stock Quote information and reading newspaper articles, make sure that the Reuse retrieved documents within session check box is deselected. If it is selected, then the cached documents are displayed instead of the new updated documents from the network.

Chapter 25. Personal Web Server

The Personal Web Server application allows you to publish your own Web page on the World Wide Web so that other users can access it using a Web client such as WebSurfer. Your Web page can include HTML (Hypertext Markup Language) other documents, images, sound files, and movies.

What is a Web Page?

A Web page is a collection of HTML documents and other files you want to serve to Web clients. This collection of documents and files reside in your Windows file system and the starting point of this collection is called the Web Root Directory.

You can have subdirectories under the Web Root Directory and you can place the file that you want to serve in both the Web Root Directory and subdirectories. Only the Web Root Directory and its subdirectories are visible to Web clients.

NetManage provides a sample Web page to help you set up your own Web server. These sample files are located in the Web Root Directory. Open the Personal Web Server icon and choose the Configure... command from the Settings menu to view this directory.

Default HTML File

The Web Server can serve an HTML file by default if you have configured it to do so. The sample Web Server is configured to serve a file called INDEX.HTM if this file exists in the directory requested by the user. You can change the name INDEX.HTM to another name by changing the filename in the Default HTM File field in the Personal Web Server Configuration dialog box. To use this feature, create the default files in each directory where you want this capability.

Note: The name of these default files must be exactly what you specify in the Default HTML File field in the Configuration dialog box. If you do not change this field, then your default files must be named INDEX.HTM.

Directory Listings

The Personal Web Server first checks to see if the default file that is being requested exists. If it does not exist, the Web Server will try to generate a directory listing for the user. The user can browse this listing and request further file fetches by selecting the filenames in the listing.

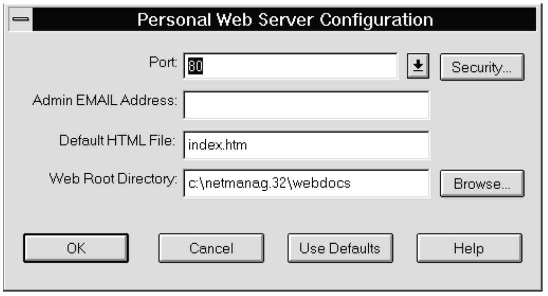
Setting Up Your Own Web Page

Look through the files that make up the sample Web page can help you understand how to set up your own Web documents and files. Once you have created your Web files, you are ready to reconfigure the Web Server to make it serve files from your Web rather than the sample Web. To do this follow the instructions in the next section that describes how to configure your own Web server.

Configuring the Personal Web Server for the First Time

To start and configure the Personal Web Server, do the following:

1. Start Personal Web Server.
2. Choose the Configure... command from the Settings menu. The Personal Web Server Configuration dialog box appears.



3. Enter the communications port number in the Port field. The default port number is 80.
4. In the Admin EMAIL Address field, enter your local e-mail address (for example, lindas@netmanage.com). The Default HTML File field contains the NetManage supplied default file DEFAULT.HTM, which resides in the default directory specified in the Web Root Directory field. This file contains a sample server home page that you can change at any time. If you rename this default file, then you must also change its name in the this field.

   When a user is trying to access your Web server (using a Web client), and specifies a directory but does not request a specific file, than the NetManage default file DEFAULT.HTM (sample server home page) is displayed. If the .HTM file exists in the requested directory, it is displayed. If it does not exist or is not in the requested directory, a directory listing is displayed. The user needs to select the file desired from this directory listing.
5. You can create your own home page or customize the sample home page, if desired. However, if you rename the DEFAULT.HTM file or place it in another directory,

then change the default HTML filename and the document root directory to reflect your changes.

6. Enter the root directory where the Web resides in the Web Root Directory field. NetManage has already created a directory called WEBDOCS that contains the DEFAULT.HTM file. Note that you are not limited to one directory for your Web documents; you can have a multiple directories from which to select files.

7. Choose the Browse... button to display the Select Document Root Directory dialog box. Select a new root directory, then choose the OK button.

8. Choose the Security button if you want to limit the number of users who access your Web server. If you are already a Root Administrator, the Administrator Login dialog box appears. Fill in your username and password in the Administrator Login dialog box, then choose the OK button.

   Root administrators are authorized to add, modify, or delete Web administrators, and have complete authority to remotely manage all Web server ports. Web administrators are authorized to remotely manage user and group access to secure Web pages on a certain port's server. Certain users may be authorized to access secure Web areas. To establish secured Web server settings, you must fill in the fields shown in the Secured Settings dialog box for each of the three tabs: Root, Administrators, and Users. You can also add or delete root administrators, Web administrators, and users.

Your Personal Web Server is now configured. You can reconfigure the server at any time by following the steps listed previously.

Additional Online Information

The online help file for Personal Web Server includes a description of each menu command, plus the following information in the "How to Use" section:

- Configuring the Personal Web Server for the First Time

- Requesting a File from the Web Client
- Setting Up Your Own Web Page

Troubleshooting

If you experience difficulties using the Personal Web Server application, refer to the following items:

❏ After configuring the Web Server, if the web client is not able to access the Web Server, check the port number in the Personal Web Server Configuration. If the port number is other than 80 (this is the default port number), then it has to be specified in the URL. If the port number is not specified, the error message "World Wide Webserver not active on hostname" is displayed.

Chapter 26. WebSpider

WebSpider is used to create and design documents to be viewed over the World Wide Web (WWW). Documents for the Web are written in the Hyper Text Markup Language (HTML), which is a text-based language. HTML documents, unlike word processor documents, can be viewed over many different platforms using various browsers, such as the NetManage WebSurfer.

HTML is simple ASCII text with embedded tags. These tags allow you to add graphics to your text, change the appearance of the text, and create hypertext links. The WebSpider allows you to create such HTML documents without having to learn HTML.

WebSpider makes formatting an HTML document as easy as formatting any regular text document. You can use its convenient toolbar and menus to apply styles, headings, links, lists, forms, and insert images. WebSpider automatically inserts tags, without making you aware of the HTML implementation. Therefore, HTML tags are defined, but not displayed.

The Editor displays the HTML in a WYSIWYG format (What You See Is What You Get), meaning the created documents appear in the Editor as they will appear when viewed in the Browser.

The advanced editing features are fully supported for HTML 2.0 files or files created in the WebSpider. HTML syntax validation and the WYSIWYG viewing also supports HTML 3.0 files.

WebSpider can be used with other NetManage applications to publish your own HTML pages for the World Wide Web (WWW). Refer to online Help for details on Publishing HTML pages.

Formatting Text

WebSpider provides a quick and easy way to format text. The typical word processor functions are available through the Edit menu: cut, copy, paste, delete, find and replace. All of these options are documented in the online Help. In addition, HTML formatting tools are available: character styles, paragraph styles (including various headings), document styles and lists.

Character Styles

Character styles apply the following style options to the characters selected:

- Bold
- Italics

- Underline
- Strong (HTML tag)

  Used to emphasize text. The display implementation is Browser dependent. The common implementation is Bold.

- Cite (HTML tag)

  Used to mark citations. The display implementation is Browser dependent. The common implementation is italics.

- Fixed Font (HTML TT tag)

  Indicates that the text should be displayed using monospace (fixed size) font. The font is Browser dependent. The common implementation is Courier.

- Strikethrough (HTML S tag)
- Size

Character styles are accessed through the Format menu. Refer to online Help for details on applying these styles.

Paragraph Styles

Paragraph styles apply to the paragraph containing the current selection. The HTML editor creates tags specifying how to format that paragraph.

The following styles are available:

- H1 to H6

  Defines text in relatively different size fonts for emphasis. H1 is the larger font, common implementation is bold. Each of these styles becomes progressively smaller. H6 common implementation is smaller than the normal font.

- Preformatted

Defines that the current paragraph should be displayed in a fixed font, and that no Browser adjustments should be performed on this text. For example, Browsers will usually eliminate white space such as double spaces and tabs. In a preformatted paragraph, no adjustments will be performed. The common implementation uses the Courier font.

- Address

  Defines that the current paragraph should be displayed as an Address. The common Browser implementation is italic. This style is often used for email addresses.

- Normal

  Defines that the current paragraph should be displayed as a regular paragraph, no styles applied.

Paragraph styles are accessed through the Format menu.

Left Indent and Center

The Left Indent and Center styles are applied to blocks of text, to format their justification. Indent styles are accessed through the Format menu or button bar.

Document Styles

Document styles apply to the whole document. You may set a background image (wall paper) for your document or the following colors: background color, text color, link color. Visited link color and active link color.

Document styles are accessed through the Format menu. Refer to online Help for details on applying these styles.

Lists

WebSpider allows you to create different types of lists to enhance text. All lists are indented from the left. The list styles available are:

- Ordered

  Automatically inserts item numbers.

- Unordered

  Automatically inserts bullets.

- Dir

  Bulleted list similar to the Unordered list, consisting of short items, typically up to 20 characters each. Items in a directory list may be arranged in columns, typically 24 characters wide. The common browser implementation is as an Unordered list.

- Menu

  Bulleted list of items with typically one line per item, making it more compact than the ordered list style. The common browser implementation is as an Unordered list.

Increase List Indent creates a nested list by moving a list item right by one tab stop. Decrease List Indent moves a nested list item back (to the left) one tab stop.

Note: Increase and Decrease List Indent commands will be available only when cursor position is in a list.

Links

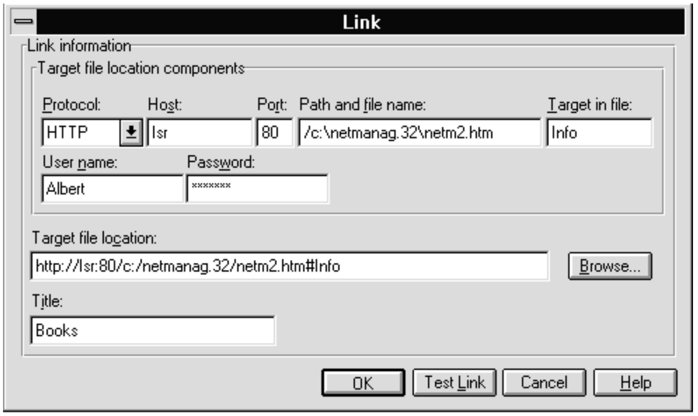
Links are used to create a jump (hyperlink) from the selected text to another specified document or Internet resource. Links are used to reference any resource on the Internet. When a link appears in the document, the Web Browser uses the link to go directly to that resource, using the protocol selected when creating the link. There are three types of links:

- Internal links jump to a reference in the same document
- Relative links jump to a file in the same directory
- External links jump to a file on another computer.

To create a link:

1. Select the text you want to link to the destination document.

2. Choose Create Link from the Link menu or use the icon from the tool bar.



A Link dialog box appears.

Specify the target file location components (URL). The information that you must supply depends upon the type of link that you are creating:

- Protocol **-** Select the communication protocol required to access the destination Host: HTTP, GOPHER, FTP, FILE, TELNET, MAILTO, NEWS.

  For Relative links, select none for Protocol and leave the Host and Port empty. For Internal links, select none for Protocol, and leave the Host, Port, and Path and File Name empty.

- Host **-** Enter the name of the destination Host.
- Port **-** The Port displayed is the standard port used by the chosen protocol. It is recommended to keep the standard port.

- Path and file name - The destination file name, including directory path if necessary. Use the Browse button to select the desired file.
- Target in file - Optional, by default the link transfers to the beginning of the targeted document. With HTTP and File protocols, it is possible to link to a specified point inside a document, if it has been defined as a Target. Enter the Target name you want to link to. Use this option to link to a different point within the same document.
- Username - Enter the appropriate username for Protocols that require a username for connection.
- Password - Enter the appropriate password for Hosts that require a password for connection.
- Target file location - The information that you entered in the Target file location components fields, automatically appears in this field.
- Title - Allows you to provide an alternative name for the destination document. Enter the name you require.

You can enter the string, if known, directly into the Target file location field, eliminating the need to fill in all the components. You may also use the Browse button to browse for a target file.

Note: It is advised to enter the details into each of the Target file location components fields, since the input is automatically converted into a valid URL, displayed in the Target file field. Advanced users, or users that know the exact target file location (URL) they wish to link to, may enter the information directly into the Target file field.

- Test Link - Choose this button to test the link you just created.

3. Choose the OK button.

The style of the selected text changes to reflect the existence of a Link.

Note: The Create Link command is available only when text, that is not already a link, is selected.

You can test, modify and remove Links - refer to the online help for details.

Targets

Links automatically jump by default to the beginning of documents. If you want a Link to jump to a specific part of a document, you must create a Target at that point. The Link can be specified to jump to the target. The Create Targets command is accessed from the Link menu. Refer to online help for details on creating, editing, and moving targets.

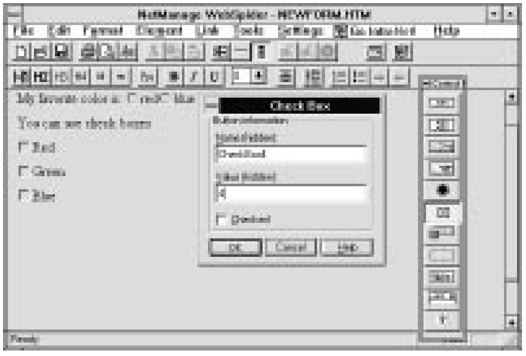
If the Highlight Targets command from the Settings menu is selected, the Targets in the document will appear in a different style so that they can be distinguished from the other text. The reflection of the Targets in the Editor is for your convenience. The common Browser implementation of Targets will not display the targets in a different style.

Note: The Create Target command is available only when text is selected which is not an existing target.

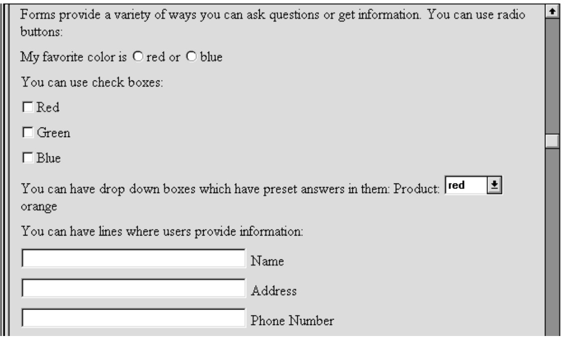
Forms and Controls

A Form is the computer equivalent of a paper form that enables the client to interact with a Web page. Forms enable two-way communication through the use of HTML. You create a Form in WebSpider and place it on the Web and this information is returned to the URL specified when the Form was created. Commands include Create Form..., Edit Form Attributes..., and Select Form

Forms may contain various Controls, such as radio buttons, submit buttons, image buttons, text boxes, selection lists and password boxes. These are interactive building blocks that can be brought together and defined as Forms. These allow the user of the Web page to enter information and submit the form. WebSpider provides ready made controls for easy use. The user just uses the convenient controls palette and enters the relevant information into the pop-up dialog box, as in the example below.



After you launch the page on the Web that you have created it will appear like the example below. See Online help for more details on Forms and Controls.



Insert Inline Image

You can insert a picture into the document as an inline image at the current cursor position, and set the image's attributes. Various graphic formats are supported, but it is advised to use the GIF format for maximum browser compatibility. If your graphics are of a format other than the GIF format, you may use NEWTView to convert the existing graphic to the GIF format. You can access Insert Inline Image through the Element menu. Refer to the online Help for more details.

WebSpider provides you with a library of icons and graphics, found in the NETMANAG\WEBSPID directory. To use these icons or graphics, refer to online help.

Example HTML files

WebSpider provides you with a small library of example HTML files and graphics in the NETMANAG\WEBSPID directory. Also, you may look at the WebSurfer library, provided in NETMANAG\WEBDOCS. To use these files first open an example file, save as another name, and edit as you wish.

Launch WebSurfer

This command can be found under the Tools menu and as an icon and will launch the WebSurfer with the current document, allowing you to view and test your document in the Browser environment.

Spell-Checking

You can use the Check Spelling command from the Tools menu (or use the toolbar icon) to check the current document for spelling mistakes. Detected errors can be fixed, using one of the suggested replacements or a replacement of your own. Detected errors may also be ignored or added as a valid word into one of your dictionaries. You may set various spell-checking options.

Additional Online Information

The online help file for WebSpider includes a description of each menu command, plus the following information in the "How to Use" section:

- Examples
- Formatting
- Forms and Controls
- Graphics
- Hypertext
- Publishing HTML Pages

Chapter 27. Forum

The Forum application allows you to subscribe to News Servers in your local Intranet environment or on the Usenet. You can connect to more than one news server, subscribe to newsgroups, work offline, read articles, and post and mail articles containing multi-media (HTML) and attachments.

Forum is used to exchange information in the form of news articles that are broadcast to discussion groups to which a group of members are subscribed. News articles are posted into newsgroups according to their subject matter. The number and type of newsgroups available to you are set by your system administrator.

Forum in previous versions was called NEWTNews. The icon for NEWTNews (Internet Access group) and the icon for Forum (Group Collaboration group), both open the same application.

Using Forum

To use Forum:

1. Choose the Forum or NEWTNews Icon

2. Log into Forum using the same name and password which you would use for mail.

   If you are already logged into a NetManage mail application, the login pop up will not appear.

3. The first time you are using Forum, enter the name of the directory where you want your configuration files saved and articles downloaded and choose the OK button.

4. You are automatically connected to the enabled Servers specified in the news tab of the Network Settings dialog box. If there are no active servers specified, select Network Settings from the Settings menu and enter the appropriate information (see Section News Servers) . The groups you are subscribed to are displayed. If you are not subscribed to any groups, select Subscribe from the Groups menu and subscribe to newgroups.

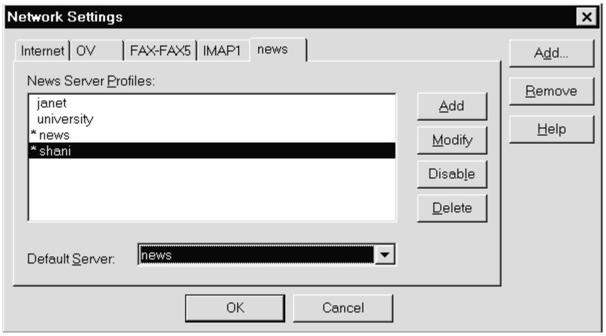
Whenever there are no servers defined, or none of the defined servers are active, or no subscribed newgroups exist for a server, a Forum popup tip outlines the above step.

5. You may toggle the Forum connectivity state, by choosing the Go Online and Go Offline commands.

News Servers

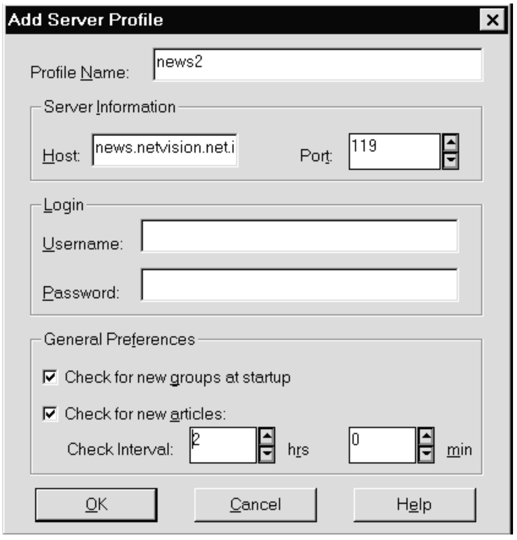
To specify the news servers used during your Forum sessions complete the following:

1. Select Network Settings from the Settings menu.



2. The Network Settings dialog appears with the news tab selected. This dialog contains a list of all the profiles in the current configuration file. All enabled profiles appear with an asterisk to the left of the profile name (disabled servers do not have an asterisk).

3. To create a new profile choose the Add button, the Add Server dialog appears.



4. Specify the Profile Name: This is the name that will appear in the News Server Profiles list box.

5. Specify the News Server:

Host: the name or IP address of the remote news server.

Port: the number of the port to connect to on the news server. The default is 119.

Note: There may be several profiles for the same news server.

6. Specify the Login details (for servers that require Authentication):

   Username: the user name used to log into the news server.

   Password: the password used to log into the news server. The password is not displayed.

7. Specify any required option in the General Preferences section:

   Check new groups at startup: when checked, each time the Forum program is started, a check for new groups will be performed on the news server. You can view new groups in the Subscribe dialog.

   Check for new articles: when checked, performs periodic checks for new articles in the subscribed groups on the news server, according to the set interval.

      Check interval: specifies the interval for checking for new articles (in hours and minutes).

8. Choose the OK button

The new profile appears in the profile list, in the Network Settings dialog.

Enable / Disable button: used to enable or disable the currently selected profile. Enabled profiles are active during the Forum session, whereas the disabled profiles are kept in the configuration file, but are not active and are not displayed. All enabled profiles appear with an asterisk to the left of the profile name.

The Modify, Add buttons: activate the server dialog box. The Delete button deletes the selected profile.

Default Server: select which profile should be used as your default server. It is advised to set the profile defining the Usenet server, as this default. Forum will use the server specified in this profile, in the following cases:

- When as articles sent form Forum specifies news group addresses that do not specify a server. Forum will post the article to the news group in the Default Server.
- When Forum receives a URL request (e.g. when activating a jump to a news group from the WebSurfer) which does not specify a server. The specified news group or article will be searched for on the Default Server.

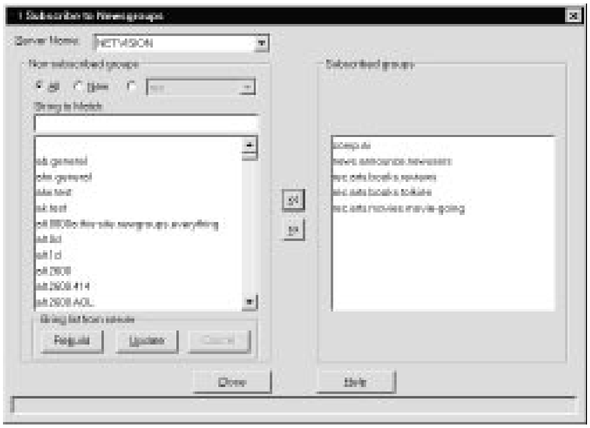
Note: Forum supports URLs using the NEWS protocol.

Subscribing and Unsubscribing to Newsgroups

This section describes how to subscribe and unsubscribe to available newsgroups.

To subscribe to a news group:

1. Choose the Subscribe command from the Groups menu. The Subscribe dialog box appears.



2. Select the News Server you require.
3. Select the appropriate radio button for the view you require of Unsubscribed groups:
   - All: full list of groups - default view
   - New: a list of the newest groups on the server

   Note: The list of the new groups is preserved locally and can be displayed at any time by choosing the New view radio. This list is recreated each time Forum performs a check for new groups on the server, either at startup or when using Update List.

   - Pull down list of hierarchies: a list of the groups according to the type of group which can be viewed by selecting the radio button.
4. If you know the name or part of the name of the news groups you want to subscribe to, then type text into the edit box above the list. The current list will be filtered according to what you enter.
5. Once you locate the desired news group(s) to which you want to subscribe, highlight them and choose the right arrow (>>) button. The selected groups will appear in the Subscribed groups list.

Additional activities available in the Subscribe dialog:

- Rebuild List - choose the Rebuild List button to connect to the server and retrieve the full list of all groups from the server. The groups will appear in the list of Unsubscribed groups (except those you are already subscribed to - which will appear in the list of Subscribed groups, on the right side).

  Note: This operation may take several minutes - depending on the amount of groups on the server.

- Update List - choose the Update List button to connect to the server and retrieve a list of all the new groups from the server. These groups will be added into your current list of Unsubscribed groups.
- Cancel - choose the Cancel button to abort the Rebuild List or Update List operations.

When you are finished, choose the Close button to update your subscriptions.

To unsubscribe or delete your subscription to a news group:

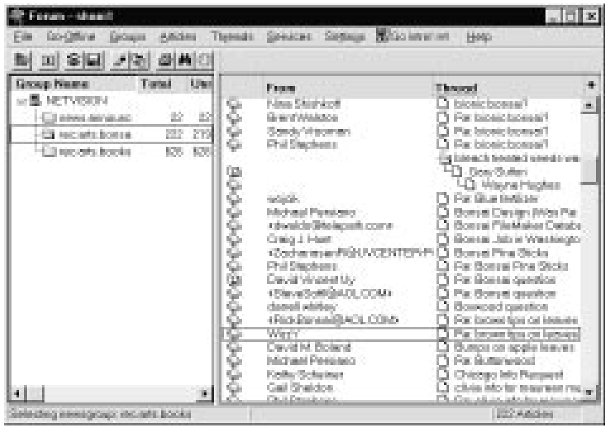
1. Select the news group which you want to unsubscribe.
2. Choose the Unsubscribe command from the Groups menu. The selected news group will disappear from the view or
3. highlight the group in the list of Subscribed groups and choose the left arrow (<<) button.

Handling and Reading Articles

To list and view the articles in a news group, do the following:

1. Select the news group whose articles you want to view. Shown next to each news group are the number of unread articles and the total number of articles.
2. Choose the Open command from the Groups menu or double-click on the group. The articles will appear in the right pane of the Forum main window.

   The Threading feature organizes the article listing so that all articles and their associated responding articles (RE: and FW:) are grouped together. You can display an article and its associated responding articles hierarchically by selecting the Expand All/Current command or double-clicking on the thread. Use Expand All only if you want to expand all threaded articles. If you want to hide all the responding articles hierarchically, then select the Collapse All/Current



3. Select the article(s) you want to view and choose the View command from the Articles menu or double click the article.

4. After you read the article, choose the Exit command from the File menu to close the article and return to the main screen.

   Note that an open book icon appears to the left of a previously read article.

Use Catch Up to mark all articles in a particular news group as read, even if you have not read any of them. All read articles may disappear when you exit the current news group (if so set in the Preferences).

Printing Articles

To print an article, do the following:

1. Select or view the article, then choose the Print command from the File menu to display the Print dialog box.

2. Choose the OK button to print the article.

Saving Articles as Text

To save an article to a specified file, do the following:

1. Select or view the article, then choose the Save as Text from the Articles menu.

2. Enter the Filename, File Type, Directory, and Drive and choose the OK button. Note that you can select multiple messages and save them all at once.

Posting and Replying to Articles

You may post articles to news groups and email addresses. The following options are available from the Articles menu:

- Post: Send an article to the selected news group.
- Reply to: Send a reply to the selected article, you have a choice of different recipients: Newsgroups, Current Newsgroup, Sender and All.
- Forward: Forward the selected article.

Downloading Articles to Your Local Drive

Downloading articles enables reading the articles at your own leisure, when Offline. To download articles, do the following:

Select or view the article, then choose the Download command from the Articles menu (you may select more than one article by holding down the Shift key while selecting the articles).

or

Select a group, then choose the Download Group command from the Groups menu.

The articles will be downloaded to your local drive. The articles are automatically downloaded to your news directory.

Note: The icon of downloaded articles or groups has a picture of a small diskette on it, indicating the article or group are downloaded.

Settings Menu

You may set the mode for editing and viewing of articles to either the plain text mode or the HTML mode. The HTML mode will enable creating multimedia articles, with styles, images, links and so on.

You can set preferences to control which articles you want to read and how they should be displayed.

You can set fonts, colors and display information.

You can define the News Servers that you want active during your Forum session.

Refer to Forum online help for detailed information using the Settings menu.

Services Menu

You may define and edit Address books, launch the Outbox, display the log and test the connection to a server.

Additional Online Information

The online help file for Forum includes a description of each menu command, plus the following information in the "How to Use" section:

- Additional Publications
- Addressing Articles
- Login Directory
- News Group Types
- Posting and Replying to Articles
- Reading and Handling Articles
- The Usenet
- Using Address books
- Using Forum
- Using the Outbox

Troubleshooting

If you experience difficulties using the Forum program, refer to the following items:

❑ Make sure the host name to which you are trying to connect to and its profile information are correctly entered in the Network Setting dialog, and that the system administrator assigned you access permissions to that server.

❑ If Forum reports a failure to connect, use the Diagnose Connection command from the Services menu, which will verify the connection to the remote host.

❑ Verify that the Forum settings are correct and the timer settings for retrieval of new articles from the host are set appropriately.

Chapter 28. Archie

Archie helps you to retrieve text, graphics, and sound files from anonymous FTP hosts (hosts that can be accessed by anyone on the Internet using a special userid named anonymous).

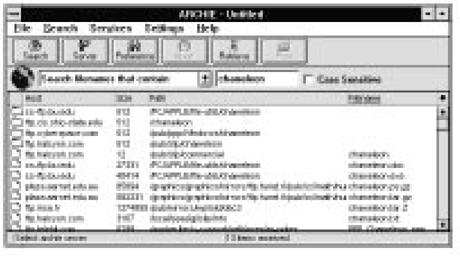
The NetManage Archie client program provides a menu-driven interface that lets you browse through the various Archie servers on the Internet as though you were browsing through card catalogues of remote libraries.

Archie is an Internet information service for gathering data and indexing widely distributed collections of data. The anonymous FTP hosts contain certain directories that are designated for public access. Other computers, called Archie servers, provide an archiving search service to help Internet users find directories and files on the anonymous FTP hosts.

Starting an Archie Search

You can start a search in either of the following ways:

- Specify the desired search criteria in the dialog bar and press ENTER.
- Select the Search menu. Specify the search criteria in the Search dialog box, and choose the OK button.



Note: Search parameters can be specified either in the dialog bar or the Search menu popup dialog box.

Selecting an Archie Server

You can start a search using the Archie default server. You can change the server by selecting a different server using the Server command from the Services Menu, or by selecting a different server from the scrolling list in the Search menu dialog box. The contents of the Archie database are usually the same on different servers. If you cannot connect to one server (the servers are often busy), you can select a different server. You can use the Query option from the Search menu to find additional Archie servers.

Specifying the Object of the Search

You can specify the object of the search in either of the following ways:

- Type a text string in the empty field in the Archie dialog bar.
- Select the Search menu and type a text string in the Keyword field of the Search dialog box.

WhatIs

WhatIs searches for the specified text string in a collection of information called the "Software Description Database." The Software Description Database contains short descriptions of programs, documents, and other files that are located on the anonymous FTP server archives. These descriptions are provided optionally by the person who submitted the file to the anonymous FTP server. Note that these descriptions are submitted on a voluntary basis, are not edited, and are not updated on a regular basis.

WhatIs is an alternative to the Search command. If your WhatIs search is successful, it can help to narrow your search of the Archie archives themselves. However, you may often be able to find files in the Archie archives even though the WhatIs search was not successful.

When you select WhatIs, Archie displays a popup dialog box with a blank field in which you can type a word (or text string).

You can select the WhatIs command either from the Search menu or from the scrolling Query list in the Search dialog box.

Retrieving Information

After Archie has successfully connected and logged in to the selected server, Archie displays a list of items retrieved.

The display of items retrieved includes the following columns (from left to right): the name of the anonymous FTP host, the size of the file, the path of the directory or, and either a file name or short comment. Note that the icons in the furthermost left column indicates whether the retrieved item is a directory or a file.

Quote

The Chameleon Archie application provides you with a menu-driven point-and-click interface for conducting an Archie search. However, if you wish, you can also type the UNIX-based Archie commands by using the Quote command in the Services menu. To display a listing of all the Archie commands, type the word "help" within the Quote text box.

Additional Online Information

The online help file for Archie includes a description of each menu command, plus the following information in the "How to Use" section: command

- Retrieving Information
- Selecting an Archie Server
- Setting Up Archie
- Specifying the Object of the Search
- Starting an Archie Search

.

Chapter 29. Talk

The Talk application lets you participate in real-time conversations with remote users. Talk acts as both a client and server, and notifies you of remote Talk invitations by displaying a popup dialog box with or without sound.

Preparing to Talk

You can specify a remote user and host (using the format <user>@<machine>) in the Talk With field of the main Talk window, or you can use a conversation profile.

The host name identifies the physical machine to which you want to connect. If the machine is being used by only one user, you can specify just the host name (@<machine>). For example, you could specify: @lizzie.netmanage.com.

If the physical machine is being used by more than one user, you can send a Talk invitation to the particular user that you want to talk with by specifying both the user name and the host name (<user>@<machine). For example, you could specify: chamboy@lizzie.netmanage.com.

A conversation profile stores necessary connection information, so that you can establish a conversation with a remote user without reentering connection details. After you create a Conversation Profile for a remote user, you will be able to send a Talk invitation by selecting the user's profile from a drop-down menu. You can define as many conversation profiles as you wish.

Establishing a Conversation

To establish a conversation with a remote user:

1. In the Talk With field, type a remote user and host name or select an existing Conversation Profile from the drop-down menu (click the down arrow next to the Talk With field to display the menu).

   - **user**: Enter the name of the remote user with whom you wish to connect. In general, you should specify both the user and the host name.
   - **host**: Enter the name of the remote host to which you wish to connect. The remote host identifies the physical machine that is being used by the user with whom you wish to talk.

     For example, if Camille had a PC, we might identify her PC's host name as `camille.netmanage.com` or as its IP address (such as, 111.22.3.444). The host name must match the name or IP address that is listed in the Domain Name Server (DNS). When you type the host name in the Talk With field, you should put the @ in front of it. For example, type @camille.netmanage.com or @111.22.3.444.

2. Choose the Talk button to open a conversation window.

3. Type your message in your section of the window, which is the box located near your username.

   You can also enter information by using the Edit menu's Copy and Paste options.

   Talk sends your message to the remote user. If the user responds to your Talk message, the response is displayed in that user's section of your conversation window. You can continue conversing with the user.

4. If you want to save the information displayed in the conversation window, take the following steps:

   - Choose the Save button.
   - A Save Conversation File As... dialog box appears.
   - Specify the file in which you want to save the conversation.
   - Choose the OK button.
   - If the specified file exists, Talk prompts you to append the information. Talk saves your portion of the conversation in one section of the file and the remote user's portion in another section.

5. After you finish conversing, choose the Disconnect button.

Receiving Talk Invitations

You can use Custom to configure your system so that Talk can respond to Invitations without having to keep your Talk application activated.

Note: Steps 1 through 5 apply to Windows 95 and Windows 3.1 users only. If you are using Windows NT, both users must have the Talk application open to talk to each other.

To configure Custom, do the following:

1. Open your Custom application.
2. Choose the INETD Configuration option from the Services menu.
3. Add both Talk servers to the list of installed servers by selecting NTALK or TALK server from the list of servers and clicking on the right arrow button.
    - Check to make sure that the Server name now appears in the list of Installed Servers.
    - Repeat this procedure for the other server.
4. Choose the OK button and close the INETD Configuration dialog box.
5. Restart Windows.

When you receive a Talk invitation, a Can We Talk? dialog box appears. You can set up Talk to also notify you audibly by selecting an option from the Preferences dialog box.

- To converse with the sender, choose the Accept button. Talk displays the sender’s message in the conversation window.
- To refuse the invitation and notify the sender you decline to talk, choose the Decline button.
- To refuse the invitation without notifying the sender you decline to talk, choose the Ignore button.

Ending Conversations

To end a Talk conversation and close Talk, choose the Exit option from the File menu or choose the Disconnect button.

Additional Online Information

The online help file for Talk includes a description of each menu command, plus the following information in the "How to Use" section:

- Creating Conversation Profiles

- Modifying Conversation Profiles
- Deleting Conversation Profiles
- Establishing Conversations
- Receiving Talk Invitations
- Responding to Talk Invitations

Chapter 30. Internet Relay Chat

Internet Relay Chat (IRC) is a multi-user, multi-channel chatting network. It lets people connected to the Internet talk to one another in real-time. Conversations take place in areas called channels. A channel is a separate and distinctive electronic meeting area where people can talk simultaneously. There can be any number of channels in IRC, and any number of people within a channel. Some channels exist all the time, while others come and go as the need arises.

Each channel usually has at least one operator, who is responsible for managing the channel. Channel operators control the channel and have special privileges to maintain it. The initial creator (and the first person) of a channel has channel operator privileges and can give or take away these privileges to others on the channel.

Users are known by a nickname. This nickname must be unique on the network at the time and is limited to nine characters. Nicknames are really not owned because you may find your normal nickname in use the next time you connect to IRC, and will be asked to use another nickname for that session.

While on a channel, all normal text that you enter goes to the channel for all others to see. The one exception to this is if the text starts with a slash. A slash means the text that follows should be interpreted as a server command. You can log the text of a chat session on a specific channel and save it automatically to a text file.

During a chat session, you can also send files to channel members using the DCC (direct client-to-client) protocol. This protocol handles any file transfer type, including text and binary. You can send one or several files to only one nickname per request, but you can send multiple requests at the same time.

If you want to keep a record of whatever you see on a channel, you can use the logging function. You can log each session on a channel separately and save it automatically to a file.

The IRC server that you connect to defines what network you are on. There is one main network often referred to as EFFNET. EFFNET usually hosts about 2500 channels with up to 5000 people at peak times. Other networks have improved the servers in varying ways and have fewer members (examples include UNDERNET and DALNET). Volunteers, usually at colleges and universities, operate and maintain these networks of servers. These networks are not secure; do not assume any form of privacy.

Connecting to IRC

To start the IRC application, choose the IRC icon to display the main IRC window.

There are two types of tabs: a Server tab and a Channel tab. There is only one Server tab, which has two main text areas. The top text area, the Server Information box, displays server output. This area can also contain error messages if local operations fail. The bottom text area, the Channel box, lists the current channels on the connected server. It displays the channel name, the number of users on the channel, and the topic.

There are as many channel tabs as there are channels that you have joined (the maximum is ten). When you join a channel, a new channel tab is created and brought to the top. It contains a small text box on top with the topic and any status information, and a large area for the channel conversation text on the bottom.

To connect, do the following:

1. Select either the Connect! menu option on the menu bar or the Connect button on the Server tab to display the Information not found dialog box. This dialog box explains what information you must enter to connect to an IRC server as a first-time user.

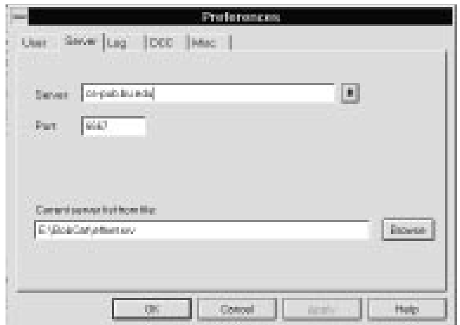
2. Choose the OK button to display the Preferences dialog box, then choose the User tab.

Complete the information in the following fields for preferences listed on the User tab:

Field	**Description**
Default nickname	Under the User tab, enter your nickname. This name can be nine characters or less.
Alternate nickname	Enter another nickname in this check box if you want to automatically try another nickname when your default nickname is in use.
Username	Enter your username if it is blank.
Hostname	Enter your hostname. The local hostname is the actual machine name. The username and local hostname may be filled in if the IRC client can get this information from the local machine.

Field	Description
Description	Enter a single line of text identifying yourself (this is optional).
"Away" text	Enter a single line of text that will be sent to people when you are away from IRC, but still connected and someone sends you a private message. This is also optional.

When you have entered information in all these fields, choose the Server tab to redisplay the Preferences dialog box.



Complete the information in the following fields for preferences listed on the Server tab:

Field	Description
Server	Shows a list of all IRC servers available on a specific network. You can select a server from the list in the drop-down menu or enter one in this field.
Port	Leave 6667 as the port number unless you know a server uses a different port number.
Current server list from file	Shows the default server selected on that network. Choose the Browse button to display the Select server list dialog box. From this field, you can select a different server than the one listed or change the server that you want to connect to for your next IRC session.

Select one of the server networks listed in the File Name list box, then choose the OK button. The Current server list from file field is then updated with the network server that you have selected.

Select the Connect button on the Server tab or the Connect! menu option.

After connecting, the Server Information box shows the following information:

- The number of IRC users

- The number of channels being used
- Other information that the server may send

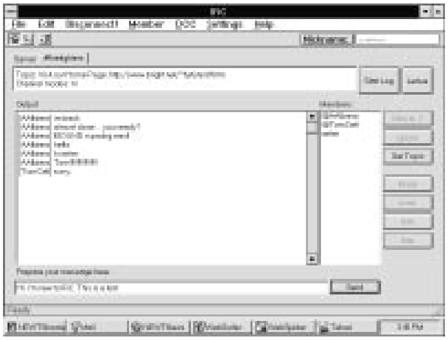
Note: You can change the IRC server selected by choosing the Server icon in the Toolbar, or by choosing the Set Server (shortcut) command from the Settings menu. Choosing this command places you directly in the Preferences dialog box, with the Server tab selected. You can make a change to the server you have selected quickly and easily, then return to your current IRC session. You will connect to the new server you have selected when you start a new IRC session. This feature is only available in Windows 95 and Windows NT.

After connecting, the Channel List box shows the following information:

- The Channel column shows the names of channels in use.
- The Users column shows the number of users on that channel.
- The Topic column describes the subject being discussed and may be blank.

Note: Your currently used nickname now appears in the Nickname: field.

From here, you can browse the channel list, join a channel, or create a new channel.



Additional Online Information

The online help file for Internet Relay Chat includes a description of each menu command, plus the following information in the "How to Use" section:

- Connecting to IRC
- Changing Nicknames
- Creating a Channel
- Getting Current User Information
- Getting Past User Information
- Transferring Files
- Using Filters
- Banning a User
- Entering Text Commands
- Using Configuration Files

Troubleshooting

If you experience difficulties using IRC, refer to the following items:

❑ If you cannot connect to a server:

  - Make sure that a valid server name appears in Preferences on the Server tab. The port number is almost always 6667, unless otherwise instructed.
  - Select the Help menu and choose the Diagnose Connection... command to see if you can ping the server.
  - Wait; sometimes the servers takes several minutes to respond.
  - You or your site has been banned from that server.

❑ If changes do not occur immediately after you make them:

  - Be aware that very busy servers may respond slowly.
  - Wait; you may not see a screen display change until the server confirms the action you performed.

❑ If you made a change and nothing happened:

  - A server may silently ignore an incorrect or redundant request. For example, if you try to set a mode on a channel that already has that mode, nothing happens and you receive no error messages.
  - Check the Server Information box on the Server tab: it may have an error message. Error dialogs do not appear because they may be out of order and be confusing.

- ❑ If you repeatedly get disconnected when you connect to a server:
  - Check the server information box for any messages.
  - Try changing your nickname or try another IRC server or another IRC network.
- ❑ If you cannot join or create a channel, check the Server Information box on the Server tab. Perhaps the channel already exists and is Invite only, needs a keyword, has a user limit, or has banned you.
- ❑ It is possible that the servers listed in the Current server list from file: field are no longer available. If you know the name of a valid IRC server, enter it in this field.

Chapter 31. NEWTView and NEWTSound

NEWTView

NEWTView is a tool for displaying, copying, and printing bit-mapped graphics files produced by various applications. For example, NEWTView lets you view, copy, or print a graphics file received as an attachment from e-mail or downloaded from Gopher or Archie. You can view, copy, or print the graphics file without using or having the application that created it. NEWTView can also be used for displaying, copying, or printing electronic documents transmitted via fax.

Supported File Formats

The NEWTView dialog box contains a list of currently supported file extensions. Supported file formats include the following:

- .BMP
- .GIF
- .JPG
- .XBM
- .TIF
- .PCX
- .DCX
- .DIB
- .WPG (Word Perfect image files)
- .PCT (Mac PICT files)

Additional Online Information

The online help file for NEWTView includes a description of each menu command.

NEWTSound

The NEWTSound application enables you to run a sound file.

Note: You must also have a sound board or a sound-board emulator installed in your computer to use this application.

- When you open the application, a NEWTSound dialog box is displayed, allowing you to select a directory and a sound file.

The NEWTSound application does not contain any additional menus. However, the NEWTSound dialog box does contain a few buttons. To play a sound file, select the file, then choose the Play button. To stop the sound file, choose the Stop button.

Supported File Formats

The NEWTSound dialog box contains a list of currently supported file extensions. The supported sound file formats are:

- .AU
- .MOD
- .SND
- .VOC
- .WAV

Additional Online Information

The online help file for NEWTSound includes a description of each menu command.

Chapter 32. Gopher

Gopher is an Internet access protocol tool. It allows you to browse through large amounts of information residing on the Gopher system called GopherSpace. GopherSpace organizes information into hierarchical menus related to subject matter or the Gopher server's geographical location.

Much like browsing through a remote library's card catalogue and automatically fetching the material you want, Gopher helps you access any of the data it presents.

The resource material in GopherSpace consists of data in a variety of formats such as text, images, sounds, and services such as Telnet connections, index servers, and phone book servers. Each item is labeled by an icon that identifies it as a directory, text file (.TXT), graphics file (.GIF), phone book, or searchable index.

While documents reside on many servers, it does not matter where the resources are located, as long as they are part of GopherSpace.

Setting Up Gopher

The NetManage Gopher client requires no special setup or configuration. However, the network on which Gopher runs must be properly configured, and you need to make sure that the File Manager associations exist between certain file types and their viewers. To see if the proper network connection exists, choose the Diagnose Connection command from the Help menu.

Gopher uses the File Manager file association system to determine the appropriate viewer program for viewing different kinds of files. For example, files with the .TXT extension should be associated with a text viewer such as Notepad or Write. To view GIF images, associate files with a .GIF extension to NEWTView or any other GIF viewer. Associate sound files with NEWTSound.

Starting Gopher

When you first start Gopher, a split window appears. The left half displays a directory tree and the right half displays only a directory of the current server's contents. The directory tree contains the contents of all the Gopher servers visited in the current session, as well as any servers saved in your configuration file.

The Gopher interface resembles a file system like File Manager and tracks two types of items: directories and resources. All resources are accessed through directories. Choose the Symbol Legend... command under the Help menu for a list of all resource types. Directories and individual resources are labeled with their own distinct icons.

Gopher maintains a configuration file (usually GOPHER.CFG) that stores your user preferences and GopherSpace hierarchy information.

Retrieving Resources from the Network

To increase retrieval performance, Gopher stores everything it retrieves in a local directory on your computer. Published Gopher resources do not change often and at times the resources may be difficult to retrieve (servers may not always be up and communication links may be slow). When Gopher cannot retrieve a desired resource from the network, it looks for it in your local directory instead.

Gopher retrieves two types of information:

- Hierarchical information, which appears in the GopherSpace display and is stored in the configuration file.

  You can open a Gopher item by choosing the Retrieve Item option from the Item menu. This option retrieves the selected item directly from the Gopher server and then opens it. When you close the item it is automatically placed in your Gopher cache directory.

- File information, which is stored in the Gopher cache directory (which you set up in Preferences). These files are often named with the letter G, a number, and an extension indicating file type.

  After an item is closed and placed in the Gopher cache directory you can open it again using the Open command from the Item menu. However, this item may not contain the most current update since you did not retrieve and open the item directly from the Gopher server.

The configuration file defaults to the name GOPHER.CFG and the Gopher cache directory is called gopher. By default, both are stored in the same directory in which Gopher was installed.

Navigating Around GopherSpace

The GopherSpace hierarchy is divided into two sections: bookmarks and Gopher servers. Bookmarks are items of specific interest which have been copied from elsewhere in the hierarchy for convenient access using the Set Bookmark command from the Item menu.

Servers are references to the Gopher server machines which act as a starting point for exploring the Gopher hierarchy.

In the left side of the Gopher window, you can select items by clicking on them or using the arrow keys. For example, to view the contents of a directory, double-click on its name. Additional directories and/or files, index searches, or phonebooks may also appear. As you continue to move through the directory in search of items of interest, the file system will expand. To collapse any part of the file structure, double-click the directory you want to close.

You can retrieve items in the right side of window, but you cannot select items from the right window because the right side tracks the left side.

Additional Online Information

The online help file for Gopher includes a description of each menu command, plus the following information in the "How to Use" section:

- Setting Up Gopher
- Starting Gopher
- Retrieving Information
- Navigating Around GopherSpace
- Saving to a Gopher Server

Troubleshooting

If you experience difficulties using the Gopher application, refer to the following items:

❏ If you cannot connect to a specific server during peak hours, try connecting again during off hours. Use the Diagnose Connection command from the Help menu to start Ping and verify that the server is available.

Chapter 33. WhoIs

The WhoIs application is a TCP/IP Internet username directory service that displays information about Internet users who are listed in that directory.

For example: Smith, Mary (SC178) msmith@SENCO.COM (513) 333-232

Information about a user includes:

- full name (for example, Smith, Mary)
- handle **-** a unique identifier, in parentheses. For individuals, the handle is composed of the person's initials and where necessary, a trailing number (for example, (SC178)).
- network mailbox for Defense Data Network (DDN) users (who are registered in the Network Information Center (NIC) database) (for example msmith@SENCO.COM)
- telephone number (for example, (513) 333-232)
- US mailing address

Using WhoIs

The Host field contains the hostname of a WhoIs Server. The default server is rs.internic.net. You can search the WhoIs directory on a different server by selecting or typing the hostname of another WhoIs server in this field.

Note: To view a list of Internet WhoIs servers, select the WhoIs Servers List option from the Help Contents menu command. A list of Internet WhoIs servers is also available using anonymous FTP to sipb.mit.edu, in the PUB/WHOIS directory.

To start a search, do the following:

1. Type or select the name of an alternative WhoIs server or use the default server.
2. Type the username or text you wish to search for in the User field.
3. Choose the WhoIs button to start the search.

Search Options

WhoIs allows you to perform a comprehensive search or a more specific search. In general, searches are conducted in the way outlined in the following paragraphs, but this may vary between WhoIs servers. Most WhoIs servers provide their own help files.

- General search: If you specify a string of alpha characters, WhoIs searches for a name, handle, or mailbox matching that string. For example, if you specify "Smith" in the User field, WhoIs searches for the name Smith, the handle (Smith) and the mailbox Smith@.
- Name-only search: Precede the name with a period. For example, if you specify ".Smith, Jane" in the User field, WhoIs searches only for the name Jane Smith (not for the handle or mailbox).
- Handle only search: Precede the handle with an exclamation point. For example, if you specify "!Smith" in the User field, WhoIs searches only for handle Smith, such as Smith College (Smith).
- Mailbox only search: Type the @ character after the name. For example, if you specify "Smith@" WhoIs searches for the mailbox Smith only, such as Smith@ CVR.COM.
- Group or organization entry search: Type an asterisk as the first character. For example, if you specify *SRI-NIC, WhoIs searches for all users in that group entry.
- Partial search, to find all entries starting with specific text: After the text you want to search for, type a dot or series of dots (for example, Robert... looks for all names preceding with the letters ROBERT, such as Robert, Roberts, Robertson and so on).

Note: If there are a large number of members, it will take longer for the list to display. The WhoIs button remains dimmed until the full list is displayed.

If the User field is blank and the WhoIs button selected, any Help or information files available from the directory specified in the Host field are displayed.

Additional Online Information

The online help file for WhoIs includes a description of each menu command, plus the following information in the "How to Use" section:

- Using WhoIs

Troubleshooting

If you experience difficulties using the WhoIs application, refer to the following items:

❑ Use the Diagnose Connection command from the Help menu to verify connectivity to any remote system to which you want access.

❑ Verify that the Host is a WhoIs server. Note that not all Internet hosts are WhoIs servers. ChameleonNFS for Windows WhoIs provides the major Internet WhoIs server (rs.internic.net) in the Hosts drop-down list by default. A list of Internet WhoIs servers is available using anonymous FTP to sipb.mit.edu, in the PUB/WHOIS directory. Alternatively, select the WhoIs Server list option from the Help Contents menu command.

Chapter 34. Internet Viewer

The Internet Viewer allows you to view several different aspects of the Internet from a single, integrated application. This integrated application includes a News client, a Weather viewer, a Stock quote service, a Web browser, and a link to NetManage's What's Hot home page. In addition to retrieving information interactively (that is, in real-time), the Internet Viewer also provides background retrieval, which lets you define how frequently you want to retrieve updated information. You need to set a timer for background retrieval using the Personalize command from the Settings menu.

You can change the view of the amount of data presented. If you double-click a feature button (such as Forum) or in the feature window, the screen splits in two. When the screen is split, you can see the HTML viewer and the Forum News group list at the same time. To return to full-screen mode, choose the button again or click once within the panel. You can view only the Forum News client and the WebSurfer Web browser in full-screen or in split-screen mode.

Icons for each feature are displayed in the main panel at the bottom, below the ticker tape. When you are accessing information, these icons have a status bar that shows which feature is being updated. Different icons are provided to show active (occurring in the foreground) and passive (occurring in the background) retrieval for each feature, except for retrieval from Web sites, where there is only one. If no processing is occurring, no status bar is displayed.

Each feature is discussed in the following paragraphs.

Note: This application works only on Windows NT and Windows 95.

Forum News Client

The Forum News client allows you to specify a News server to log into, and to download a copy of all the current news groups available. You may double-click individual News groups to subscribe to them. When you have selected a News group, a tab is added that contains the group title. The new tab shows a list with all the items available for the topic. When you choose an article, it is downloaded and displayed in HTML format.

The Forum News client provides background processing for both articles and groups. Each article may appear as one of five colors in the list, depending on its status:

- Yellow: Article has not been downloaded at this time.
- Red: Article was downloaded using background processing and has been added since you subscribed to a selected group.
- White: Article has been viewed.
- Green: Article has been downloaded, but has not been viewed.
- Gray: Invalid article. Article is out of date and has been removed from the server.

When you set the options to allow background news retrieval, unread articles for subscribed groups are automatically retrieved. If new articles have been added to the subscribed group since you originally subscribed, these articles are appended to the list and eventually will also be retrieved using background processing.

When you close the application, all subscribed groups and article status (that is, read, unread, invalid, and so on) are saved, as well as any entries or changes you have made to the application configuration.

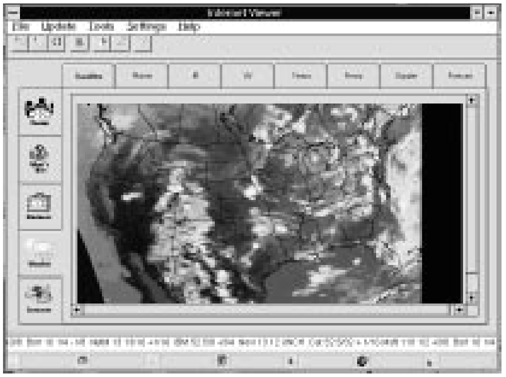
Note: There is no limit to the number of articles or newsgroups that you can download.

To start the Forum News client, choose the Forum button.

Weather Viewer

The Weather Viewer allows you to configure and tailor the type of weather images or Web pages you would like to view.

Note: These images and home pages need not be weather related at all. This feature provides you with a mechanism to download data and images over the Web using active retrieval or passive retrieval (by using background processing).



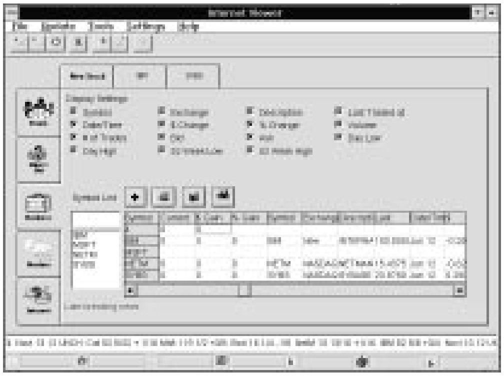
The Internet Viewer comes configured with several links to weather sites, but you may add other sites to the drop-down lists contained in the Personalize dialog box. You add other sites to the list by entering the URL to retrieve, then pressing the Enter key. All your changes will be saved when the you close the application. When you choose the Weather button, eight tabs are displayed that let you select from the following:

- Satellite: Satellite view of the US
- Marine: Coastal marine forecast from the National Weather Service
- IR: Infrared satellite view of the US
- UV: Ultraviolet satellite view of the US
- Temps: Listings for surface temperature, radar, and wind velocity in the US
- Precip: Forecast for rain in the US
- Doppler: Satellite view of the US showing Doppler effect on weather
- Forecast: Projected temperatures shown for various parts of the US

To start the Weather viewer, choose the Weather button.

Stock Quote Service

The Stock Quote service, like the Forum News client, allows both active and passive retrieval of data. When you choose the Stock Quote tab, stock information for a company you have requested previously is retrieved.



The Display Settings box shows a list of 15 items that you can select to display certain stock information. When you select a stock, the settings you have selected are displayed in an easy-to-read table. You can scroll to see information in all the columns using the scroll bar, located below the table.

The Symbol list shows four buttons that you can choose to do the following:

- Choose the Plus button to add a new ticker symbol to the Symbol list box.
- Choose the Printer button to print the stock information shown.
- Choose the Graph button to display the Stock Performance Graph dialog box. You can select from various graph or chart types, including bar, pie, line, log, and Gantt. Select the chart type you want, then choose the OK button.
- Choose the Water Mark button to show the Water Marks dialog box.

  Water marks indicate pre-established stock prices that you enter for notification purposes. Enter the high and low prices in the correct fields. If you want notification when specific stocks reach these prices, you can do either or both of the following in the Notify box:

  - Select the Sound Alarm check box to be notified by a beep on your computer.
  - Select the Tag-It! check box to be notified by a Tag-It! message. When you are done, choose the OK button.

You can receive late-breaking stock news in the shaded text box below the stock information table. Share prices for various stocks are also shown in real-time on a ticker tape at the bottom of the panel.

To start the Stock Quotes viewer, choose the Business button.

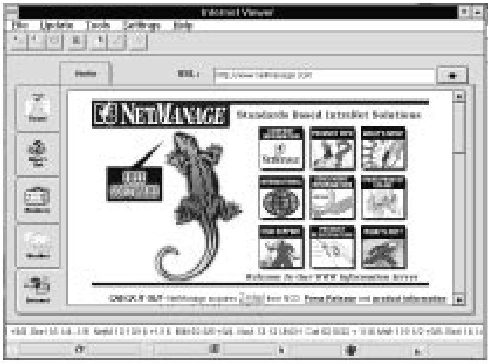
What's Hot

What's Hot is a ready-to-use hot list that contains a single URL to the NetManage home page. Included are news flashes and press releases about NetManage products, provided for your information.

To download the What's Hot home page, choose the What's Hot button.

WebSurfer Web Browser

The WebSurfer Web browser is a fully functional browser that includes a hot list from which you may add or delete URLs. When you choose a URL provided on the list, the home page and data associated with the URL are downloaded automatically. There is no background processing for the Web Browser panel. All URLs you add to your hot list will be saved between sessions.



If you want to access a specific Web site immediately, choose the plus (+) button next to the URL list box or double-click a selected URL.

To start the WebSurfer Web browser, choose the Internet button.

Additional Online Information

The online help file for the Internet Viewer includes a description of each menu command, plus the following information in the "How to Use" section:

- Using the Forum News Client
- Using the Stock Quote Service
- Using the What's Hot Home Page
- Using the Weather Viewer
- Using the WebSurfer Web Browser
- Personalizing Your Internet Viewer
- Defining Retrieval Settings

Troubleshooting

If you experience difficulties using the Internet Viewer application, refer to the following items:

- To prevent your network from getting unnecessarily slow, you should choose timer values for the background processing that seem reasonable (That is, updating the weather every 5 minutes is not very useful since most weather images and reports are only updated twice a day.).
- If you enter URLs that do not have an address cached on the local proxy server, it may cause performance problems. To minimize this occurring, set timers to download data during light-traffic times on the network.
- I f you experience difficulties loading the Internet Viewer after it has been run the first time, delete the .CFG and .DAT files from the root directory and restart the application.

Chapter 35. Automatic Internet

Automatic Internet lets you sign up for a new Internet account and automatically configure your software for connecting to the Internet. Automatic Internet typically only needs to be used once for each account for which you sign up.

Note: You can use this application only if you are running the Windows 95 or Windows 3.1 operating system.

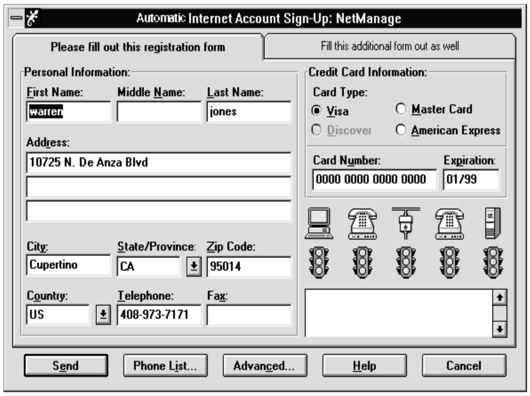
Using Automatic Internet

To use Automatic Internet, do the following:

1. Open Automatic Internet. The Select Internet Provider dialog box appears that includes a tab for each Internet Service Provider and an Others tab. The Others tab allows you to configure your existing Internet Service Provider account.

2. Select the tab of the Internet Service Provider you want to use and then choose the Signup button. You can obtain more information on each Provider by selecting the More Info...button.

An Automatic Internet Account Sign Up: NetManage dialog box appears. This is where you enter your personal registration information to send to the Provider.



3. Type your name, address, and phone numbers in the appropriate fields. Enter your name and address as it appears on your credit card. You can enter information in the State/Province and Country fields by either selecting it or typing it.

   Note: Automatic Internet uses a secure phone call to communicate with the registration server. Your credit card number will not be sent over the Internet.

   You will be presented with various connection options and prices when you actually connect with the Provider’s server. You will have an option to cancel the sign-up process after you have connected with a registration server.

4. Select the type of credit card to which you want to charge the Internet services, and enter that card’s number and expiration date. You must enter the expiration date using the following format: mm/yy.

5. Depending on the Service Provider you choose, there may or may not be additional tabs to supply additional information that the Provider is requesting at signup. Select these tabs if they are present and supply the requested information.

6. Choose the Phone List button on the bottom button bar to open the Phone List dialog box.

The phone numbers listed are used to send in your registration information. Choose the one most local to you. You may edit this number to add special dialing prefixes (such as 9) if required by your phone system.

Note that this is not the access number you use to call your Provider for connecting to the Internet. You will not be communicating your registration information over the Internet. That number will be downloaded from the Provider upon signup.

7. Return to the first registration tab and choose the Send button to sign up for your account. Once you choose the Send button, the status area on the lower right will indicate the progress of your connection and signup. The traffic lights signal the flow of the registration data from your computer through the phone line to the Provider's server and back. Accompanying status messages are displayed in the text box located below the traffic lights.

8. Choose the Advanced button in the bottom bar only if you need to specify a different communications port. The Automatic Internet application will automatically sense your modem on the appropriate COM port. If your modem does not initialize correctly, please verify the COM setting by selecting the Advanced button and choosing the correct COM port.

   In a few minutes, the Provider's server will send back your personalized account information and automatically configure your software. An interface will be created for Custom that defines how you connect to your Provider. This includes your username, password and dialup number. Automatic Internet will also create a Z-Mail® Pro account for you and set WebSurfer, Forum and Gopher defaults according to your Provider's specifications. A welcome message from your Provider will be displayed; you are now ready to explore the Internet.

9. Reboot your system, then run the Custom application to connect to the Internet as described in the following paragraphs.

   Caution: Creating a second account from a new Provider will overwrite the domain name server, Z-Mail® Pro server and gateway, NNTP News Server, and WebSurfer default home page settings. Therefore, before creating a second account, you should write down these settings or back up the .CFG files.

Verifying or Modifying Your Configuration

The first time after you run Automatic Internet, you should verify the settings for port, modem, and dial under Setup in the Custom application. If you are using an ISDN card that supports direct interface with the card using the WinISDN standard, and if your Provider has downloaded an ISDN interface, you will need to select the Hardware Type in the Setup menu. You need to verify these settings only once. Use Automatic Internet only once to sign up for each account.

Note: NetManage is not an Internet Service Provider; you can only register the software with us or configure an existing account. However, it is easier to use Custom to configure an existing account.

Connecting to the Internet

You use Custom to initiate your dial-up Internet connection. After verifying your settings, simply select Connect from the Custom menu. For more information about Custom, refer to the NetManage Configuration Guide.

Additional Online Information

The online help file for Automatic Internet includes a description of each menu command, plus the following information in the "How to Use" section.

- Choosing and Signing Up for an Internet Service Provider
- Configuring an Existing Internet Account
- Connecting to the Internet
- Filling Out Internet Registration Cards
- Verifying Your Automatic Internet Connection

Troubleshooting

If you experience difficulties using Automatic Internet, refer to the following items:

❑ Be sure to fill out all the input fields before you send your registration information in. An account application must be completely filled out to be accepted by the Provider.

❑ Enter your name as it appears on your credit card.

❑ If the Provider's server takes too long to respond, your connection will be timed out. Try to send your registration information later, or increase the Timeout before Disconnecting option in the Custom application (use the Setup menu, Communications command, Dial option).

See Automatic Internet online help for additional troubleshooting tips.

Chapter 36. NEWTToolbar

NEWTToolbar maintains a list of applications that you can start without having to switch to a window. You can make additions, modifications, and deletions to NEWTToolbar's list of applications.

Starting NEWTToolbar

When you install Chameleon ATX, NEWTToolbar is automatically added to your Startup group and the Desktop Management group. If you do not want NEWTToolbar to be started each time you start Windows NT, Windows 95, or Windows 3.1, remove the NEWTToolbar icon from your Startup group. You can then start NEWTToolbar by choosing it from the Desktop Management group.

Note: The NEWTToolbar application is automatically placed in the group called Startup. If you are running non-English Windows, your system has a startup group with a different name that automatically launches applications. Therefore, you need to copy the NEWTToolbar application from the group called Startup to the group in your system that automatically launches applications.

After NEWTToolbar is started, the toolbar appears. The toolbar includes a button for each application that is on the NEWTToolbar application's list. By default, NEWTToolbar includes buttons for the following applications:

- Go IntraNet!
- Z-Mail® Pro
- Forum
- WebSurfer
- WebSpider
- Telnet

For details on an application included in the NEWTToolbar, see the appropriate chapter.

Main Menu

To open the main menu, use the right mouse button to click on either the NEWTToolbar clock or on the outside of any toolbar button.

Note: Be sure to click on the outside of a button and not directly on a button. Otherwise, you will open the button menu.

The main menu includes the following commands:

Main Menu commands	**Description**
Add Toolbar Buttons...	Lets you add an application to the NEWTToolbar list. For details, see the Adding Applications to NEWTToolbar section.
Docking	Controls where you can move the NEWTToolbar window. If this command is disabled, you can move the window anywhere. If this command is enabled, you can move the window only to the top, bottom, left side, right side, or a corner of the screen.
Fit to Screen	Fits the toolbar to the size of your computer screen and fits the toolbar buttons to the size of the NEWTToolbar window. This command is useful if some buttons are not currently displayed.
Always on Top	Lets you control whether the toolbar is always on top of other applications. Select this option if you frequently use Go IntraNet! When you select text to shoot to Z-Mail® Pro or WebSurfer, you must select the desired text first, and then immediately choose the Go IntraNet! button and drag your cursor to the desired menu option.
Display Icon Only	Lets you control whether toolbar buttons include only an application's icon or the application's name and icon.
About NEWTToolbar...	Displays information about the NEWTToolbar application. Choose the Copy button to copy version information to the clipboard. You can then paste this information into any application. Choose the OK button to continue.
Close NEWTToolbar	Exits the NEWTToolbar application

If you enable the Docking, Fit to Screen, Always on Top, or Display Icon Only command, a check mark appears next to the command's name.

Button Menu

To open the button menu, use the right mouse button to click on any toolbar button.

Note: Be sure to click directly on a button and not on the outside of a button. Otherwise, you will open the main menu.

The button menu includes the following commands:

Button Menu Command	Description
Remove Toolbar Button	Removes the application and its associated button from the NEWTToolbar application's list. **Note:** You cannot remove the Go IntraNet! button.
Modify Toolbar Button...	Lets you modify the application's button name and command line information. For details, see the Modifying Toolbar Buttons section.

Adding Applications to NEWTToolbar

Add an application to NEWTToolbar if you want to be able to start that application from NEWTToolbar. To add an application, do the following:

1. Use the right mouse button to click on either the NEWTToolbar clock or on the outside of a toolbar button. The main menu appears.
2. Choose the Add Toolbar Button... command from the main menu.
3. In the dialog box that appears, type the name you want to use for the application you are adding. This name will appear on the toolbar button if the Icons Only command is not enabled.
4. In the command line text box, enter the path for the .EXE file of the application you want to add. If necessary, choose the Browse button to help you locate the desired file.
5. If you want to include any command line options for the application, type those options in the command line text box.

   For example, suppose you do not want the Z-Mail® Pro splash screen (large Z-Mail® Pro box with copyright information) to appear each time you start Z-Mail® Pro from within NEWTToolbar. You can bypass this screen by entering the following in the command line text box:

   ```
   C:\NETMANAG\MAIL.EXE -nologo
   ```
6. Choose the Add button.

Modifying NEWTToolbar Buttons

To change a toolbar button's name or command line information, do the following:

1. Use the right mouse button to click on any toolbar button. The Button menu appears.

2. Choose the Modify Toolbar Button... command from the button menu.

3. Edit the dialog box as desired.

   For details on entering information in the command line text box, see the Adding Applications to NEWTToolbar section.

4. Choose the OK button.

Starting an Application in NEWTToolbar

To start an application in NEWTToolbar, use the left mouse button to click on the button of the application you want to start.

Note: Be sure to click with the left mouse button. Otherwise, you will open the button menu. If you try to start an application that is already running, that application's window becomes the active window.

Exiting NEWTToolbar

To exit NEWTToolbar, use the right mouse button to click on either the NEWTToolbar clock or on the outside of a toolbar button. Choose the Close NEWTToolbar command from the main menu.

Any changes made in NEWTToolbar are saved to the NEWTToolbar application's configuration file.

Chapter 37. Virtual Desktop Manager

Note: This application is only available on the Windows NT and Windows 95 operating systems.

The Virtual Desktop Manager (VDM) enables you to manage windows in a virtual desktop, one much larger than the physical display area. When running the Virtual Desktop Manager each active application is represented as a small window known as the virtual window. The normal application window is known as a real window.

The default dimensions of the virtual screen are three real screens in width by two in height. You can manage virtual windows as you would real windows. This means you can move, scale, minimize or maximize them. You can put a window any place in the virtual or real area and VDM reflects the changes. You can also reposition your windows indirectly using the VDM. Many of the features of the VDM make it a program manager for the virtual screen.

Getting Started with VDM

When you activate the Virtual Desktop Manager for the first time, the virtual windows are displayed as set in the default settings. Assuming there is an active application running, you will see a yellow frame, known as the scope, surrounding a virtual window (assuming the application is not minimized or maximized). The scope and the virtual windows are objects in the Virtual Screen. Fro example, if you have Z-Mail® Pro and WebSurfer running on your screen, the VDM contains a scope and a virtual window for both of these applications.

When first activating the VDM, you will also see a status bar at the bottom of the virtual screen. The right hand corner of the status bar lists the scale for defining the relationship of real to virtual objects. If you place the mouse over one of the virtual windows, the left corner of the status bar will contain the caption of the application running in that window. Otherwise, you will see the word Ready.

Note: If you are restarting VDM, it will automatically restart with the previous configuration file.

Managing Virtual Windows

Every active application that is not minimized or maximized on your screen is represented in the VDM.

You can move both real and virtual windows by dragging them on the screen.

When you move a real window, you see its related virtual window move. When you move a virtual window by dragging it in the VDM, you will see its related real window move.

When you select a virtual window to put it in the foreground, the real window also moves to the foreground. The opposite is also true. When you put a real window on top of all other windows, its related virtual window also pops to the top of all virtual windows.

Resizing the active virtual window by dragging its border, also resizes the real window and vice-versa.

Closing an application, also closes the related virtual window.

Note: Minimized and maximized windows may be displayed on the Minimized or Maximized Bar described in the Displaying Minimized and Maximized Application section of this chapter.

Moving the Scope

To move the scope, grab the scope and move it to the new location. You will know it is ready to be moved when the cursor changes to a hand.

There are two ways of moving the scope: Continuous or Shadow. To set the method:

1. Choose the Preferences... option of the Settings menu. The Preference Dialog appears.

2. Select either Continuous or Shadow. If you select Continuous, all real windows move as you move the scope. If you select Shadow, the scope draws a shadow of itself while you are holding the left mouse button down. After you release the left mouse button, the real windows move.

Note: When moving the scope all virtual windows stay where they are.

Rescaling the Virtual Screen

You can rescale the virtual screen manually or automatically. To manually set the current scaling factor:

1. Choose the Preferences... option from the Settings menu.

2. Enter the new scale in the current scale display area. The default is 8 when the VDM is installed. Otherwise, it is the last setting.

3. Choose the OK button.

Another way to rescale VDM is to grab a corner of the VDM window and drag it to the desired scale. As you drag the screen you will see the scale changing in the status bar. This is the most commonly used method.

To rescale the VDM automatically:

Choose the Rescale VDM option from the View menu.

Note: You might use this method if you are dragging a real window past the limits of the Virtual Desktop area. A virtual window cannot be dragged past the limits of the Virtual Desktop area, however, real windows can be moved anywhere on the screen. Therefore, to reflect this, VDM tries to move all other objects (excluding the window being moved) in the opposite direction, if possible. If it is not possible, it enlarges the manageable virtual area by automatically rescaling the VDM.

Displaying Minimized and Maximized Applications

Special types of status bars known as the Minimized and Maximized Bar let you display the minimized and maximized applications.

Minimized Bar

To display the Minimized Bar:

1. Select the Show Minimized option from the Settings menu. This displays the Minimized Bar with icons of all the minimized application. If a minimized application has no icon, VDM assigns a default icon.

2. Place the mouse over the icon to see its caption in the status bar.

To restore a minimized application to the VDM, double click on its icon in the minimized bar. If you double click on a virtual window, the window is minimized and shown on the minimized bar.

Maximized Bar

To display the Maximized Bar:

1. Select the Show Maximized from the Settings menu. This displays a Maximized Bar with icons of all the maximized windows. If a maximized application has no icon, VDM assigns a default icon.

2. Place the mouse over the icon to see its caption in the status bar.

To restore a maximized application to the VDM, double click on its icon in the maximized bar.

It is possible to activate a menu option of an application or see its menu bar when it is in the maximized bar.

Viewing Application Menu from Virtual Window

It is possible to activate a menu option of an application or see its menu bar using the VDM.

To do this:

1. Hold down the right mouse button. For example, if Z-Mail® Pro is one of the virtual windows, holding down the right mouse button will display a drop down list box with the Z-Mail® Pro Menu bar: File, Message, Folders, Services, Settings, Help.
2. Choose one of the menu commands to list the command options.

You can also activate a menu option of a maximized or minimized virtual window in the same way:

1. Place the cursor over the icon in the Maximized or Minimized bar
2. Hold down the right button.

Using VDM as a Filter

The Virtual Desktop Manager can be used as a filter for viewing only specific application windows. To use VDM as a filter, invoke it from the Command Line with one of the following options:

-c *class types*	displays special classes listed in class types. These are additional classes that are not direct descendants of the desktop.
-f *filename*	uses the settings in the configuration file specified in filename.
-i	activates VDM in iconic mode, that is, VDM starts normally and is immediately minimized.
-o *classes*	displays only those applications listed in classes. For example, if you enter -o NMEditView, VDM will display only the edit windows of a NetManage application. This means it will display only the edit window of a new mail.

Additional Online Information

The online help file for VDM includes a description of each menu command, plus information on How To:

- Trim NonVirtual Area

- Rescale the VDM
- Change Colors
- Set Preferences

Troubleshooting

If you experience difficulties using the VDM application, refer to the following items:

❑ If your windows are not being displayed correctly, first try rescaling. Many problems are a result of improper scaling.

❑ If rescaling does not work, try closing VDM and reactivating it.

Chapter 38. Ping - Network Testing

The Ping application is a network diagnostic tool used to verify connectivity to a particular system on your network. The Ping application transmits any number of echo requests to a particular system and displays the results for each echo reply. This exchange is referred to as pinging. These echo requests and replies are supported by all systems on the network, including those that are currently running one or more Chameleon for Windows applications. This is accomplished through the use of the Internet Control Message Protocol (ICMP). If you encounter a problem in an application, pinging determines if the problem is with the application or due to a non-responding host. You may have multiple instances of the Ping application active simultaneously.

The current options and host name can be saved to a configuration file named PING.CFG (default). The configuration file is loaded when the Ping application is started. If a host name or IP address is specified, the Ping application immediately begins pinging that host. You can also supply a command line option to specify the host name or IP address.

Ping Host

To ping another system, do the following:

1. Select Start. The Host dialog box appears.
2. Type or select a host name or IP address in the Host dialog box.
3. Choose the OK button. The application sends an echo request and waits for the echo reply. If no reply is received within the timeout value (default is 6 seconds), the ping fails.

A single ping produces summary lines in the Ping window. Multiple iterations or continuous pinging produces a single summary line for each ping. After each ping, summary lines are generated, giving the total statistics for this sequence of pings.

Note: When minimized, an animated icon of a ping-pong ball indicates active pinging. The ball moving back and forth indicates successful pings.

Ping Settings

The Ping Settings menu lets you access the Preferences dialog box and the Auto Start option.

The Preferences dialog box contains the following options:

Iteration: Specify the number of echo requests to transmit (1 through 9999 seconds); for an unlimited number of iterations, select Continuous.

Data Length: Specify the length of each echo request (0 through 1,400 bytes). The default is 64 bytes.

Interval: Specify the time the application waits for a response to the ICMP_ECHO request sent (1 through 999 seconds). If there is no response after the specified time the ping returns "0 bytes received".

Timeout: Specify the timeout value for the echo reply (1 through 999 seconds).

To automatically start Ping through your Windows application, do the following:

1. Choose the Auto Start command from the Settings menu.
2. Choose Run from the Start menu and type the following command line:
   `ping<system name>`

Additional Online Information

The online help file for Ping includes a description of each menu command.

Troubleshooting

If you experience difficulties using the Ping application, refer to the following items:

- ❑ Try to Ping by using the IP address (instead of host name) to determine if there is a name resolving problem.
- ❑ The network portion of the IP address must be a locally connected interface, or a default gateway must have been specified to route unknown networks.
- ❑ A local host table or domain server must be specified when pinging by host name. The host name entry must specify the correct IP address.
- ❑ For a Windows system, the NEWT application must be running for Chameleon for Windows 95 or Windows 3.1 to respond to an echo request.
- ❑ Make sure that the Ethernet type is correct: it should be Ethernet/DIX in almost all cases.

Chapter 39. Finger

Finger is an Internet utility that enable you to get information about a user on a specific host. This information includes the person's name, job title, and e-mail address. You can set up Finger to retrieve one or more text files that contain information, such as a resume, that you want made public.

To use Finger, do the following:

1. Start Finger to display the Finger dialog box.
2. Select or type the desired host in the Host field.
3. Type the desired username in the User field.

   If a username is not specified in the User field, Finger displays information on all the users currently logged in to the specified host.
4. Choose the Finger button.

Verbose Option

Choose the Verbose option from the Settings menu.

Verbose displays information provided by the Finger application for each user logged on to the specified host. This information includes:

- login name
- full name
- home directory
- login shell
- time of login (if user is currently logged in)
- time of last login (if user is not currently logged in)
- terminal or host from which the user logged in
- last time the user received and read mail, and idle time
- information of any plan contained in the file .plan and any project described in the file .project, both are located in the user’s home directory

By default, Finger displays one line of information per user including login name, username, TTY, idle time, time of login, and terminal or host from which the user logged in.

Additional Online Information

The online help file for Finger includes a description of each menu command, plus the following information in the "How to Use" section:

- Using Finger

Troubleshooting

If you experience difficulties using the Finger application, refer to the following items:

- ❑ Use the Diagnose Connection from the Help menu to start Ping and verify connectivity to any remote system to which you want access.
- ❑ Verify that the host entered is a valid Finger server.

Chapter 40. Bind - Name Server

The Bind application provides name server functions as a simplified Domain Name Server (DNS). You can specify multiple domains and a list of host names within each domain. Bind provides hostnames to IP address resolution according to the DNS protocol. DNS is used to administer host and domain names from a central location, which is simpler than maintaining a hosts file on each network host

Name Server Setup

To specify a domain:

1. Choose the Add button in the toolbar. The Domain dialog box is displayed.
2. Enter the name of a domain (for example, netmanage.com), then choose the OK button.
3. Choose the Save button to save changes to the list of domains and any hosts defined for each domain. The saved information is in a standard format as a collection of files for each domain.

To add host definitions to a domain:

1. Select the domain and choose the Modify button. A dialog box appears with the domain name as the title. You may add, modify, or delete individual host definitions.
2. Enter the name of the host and its IP address. The name should not specify the full domain. For example, defining a host named howard resolves requests for both howard and howard.netmanage.com.
3. Choose the OK button when you are done.

Additional Online Information

The online help file for Bind includes a description of each menu command, plus the following information in the "How to Use" section:

- Using Bind

Troubleshooting

If you experience difficulties using the Bind application, refer to the following items:

- ❑ The Bind application must be running to respond to name server requests. You can place a copy of Bind in your Startup group to ensure this. You may run Bind minimized, as an icon.
- ❑ Verify that the domain server IP address is entered correctly in the system originating the name lookup request.
- ❑ Verify that the domain name and host name definitions are entered correctly in the Bind application.

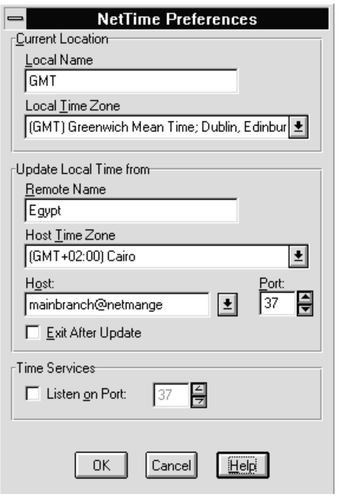
Chapter 41. PC Net-Time

PC Net-Time is a client/server application that allows you to synchronize the time between your PC and any time server. As a client, you can use PC Net-Time to automatically synchronize your PC's time with the server, or to update the time manually by selecting the update button. You can also set your preferences so that your PC Net-Time will act as a server with which other PC Net-Time clients can synchronize their PC's time.

Setting Up PC Net-Time

To set up PC Net-Time, do the following:

1. Start PC Net-Time.
2. Select Preferences... from the Settings menu.



3. Select the desired options in the dialog box. The options are as follows:
   - Current Location: Specify the time zone where your PC is located. Using the drop-down list box, you can select the time zone according to its geographical area or its relationship to GMT (Greenwich Mean Time).
   - Host: Type or select the name of the time server with which you are synchronizing the time of your PC.
   - Port: Type or select the time server's port number. The default port is 37 (as defined in RFC 868).

- Host Location: Specify the time zone where your host is located. Using the drop-down list box, you can select the time zone according to its geographical area or its relationship to GMT.
- Exit after Update: When you start PC Net-Time it automatically updates or synchronizes the time of your PC with the specified host. Select the check box for the application to automatically exit after the update is completed.
- Listen on port: Enables Time Services, which allows your PC to act as a server to PC Net-Time clients. Other PC Net-Time users can identify your PC as the host for synchronizing the time of their PCs.
- Port number: Select or type the time server's port number that should be listened to for incoming time requests (if Time Services is enabled).

4. When you have finished selecting the desired options, choose the OK button.

Updating and Getting Local and Host Times

You can easily update the local time or get the host time by using the Commands menu options or the Update and Get buttons. You should do this after making any changes in the Preferences dialog box.

Changing Between Analog Clock/Digital Clock

To change the clock's screen display choose either the Analog Clock command or the Digital Clock command from the Settings menu.

Changing the Default Time Zones

An advanced user can create another timezone.ini file by copying and renaming the default file, then editing it to include the additional country and time zone. Each listed country is followed by the time difference from GMT. To expand the list, add the country and time difference in the same format. For example:

```
Hongkong=GMT+08:00
```

To change the default time zone file used:

1. Select the Preferences option from the Settings menu.
2. Choose Local Time Zone.
3. Choose the OK button to save the changes to the new configuration file.

Automatic Startup

Select the Run at Startup option from the Settings menu if you want the PC Net-Time application to start automatically when Windows is started.

Additional Online Information

The online help file for PC Net-Time includes a description of each menu command, plus the following information in the "How to Use" section:

- Using PC Net-Time

Troubleshooting

If you experience difficulties using the PC Net-Time application, refer to the following items:

❏ Use the Diagnose Connection option from the Help menu to verify connectivity to any remote system to which you want access.

❏ Verify that the correct time zone has been set.

❏ Verify that a valid host is selected.

❏ Verify the time on the selected host.

❏ Verify the time zone of the selected host.

Chapter 42. Name Resolver

The Name Resolver application enables you easily to retrieve the IP address of a known host or the hostname of an IP address.

To find the IP address of a host:

Enter the host name in the IP Address or Hostname field, then choose the Lookup button or press the Enter key.

To find the host name of an IP address:

Enter the IP address in the IP Address or Hostname field, then choose the Lookup button or press the Enter key.

If Name Resolver is able to locate what you are looking for, both IP address and host name are displayed. Otherwise, a message is displayed stating why the host name or IP address cannot be found.

Additional Online Information

The online help file for Name Resolver includes a description of each menu command.

Troubleshooting

If you experience difficulties using the Name Resolver application, refer to the following items:

- ❏ Use the Ping application to verify connectivity to any remote system to which you want access.
- ❏ Check to make sure that your network DNS servers are set up and correct.

Chapter 43. R-Commands

The R-Commands application lets you enter and execute the rcp (remote copy) and rsh (remote shell) UNIX commands in a Windows environment:

- rcp (remote copy) is a network version of the UNIX cp command. Use this command to copy files from one computer to another system on the network.
- rsh (remote shell) is a network version of the UNIX sh command. Use this command to establish a connection to a remote UNIX system and then execute a command on that system.

Both the rcp and the rsh commands eliminate the need to first log in to a remote system. Passwords are also bypassed, by creating "trusted" Hosts or Usernames, avoiding the need to send passwords over a network. Therefore, you may find these commands more convenient to use than the standard UNIX cp and sh commands.

The commands you enter can be executed immediately or saved for later use. Save commands if you want to add them to a Command Profile or execute them later. Similar to a batch file, a Command Profile is a series of rcp and rsh commands that are executed one after the other. Like the commands, a Command Profile can be executed immediately or saved and then executed later.

Setting Up R-Commands

Note: "Local user" refers to the user entering the R-commands.

"Remote user" refers to the user connected to the UNIX system where the command will be executed.

Entries in UNIX /ETC/HOSTS.EQUIV and .RHOSTS files determine whether the Local user is permitted to access a Remote user's login account. These files define all Local usernames and their host (machine) names that are allowed to access the remote login account. You may edit these files. They are plain text files with the entries in the following format:

```
hostname <space> username
hostname <space> username
```

Where **-**

"hostname" is the name of the UNIX or PC Host from which the rcp/rsh will be performed

"username" is the Local user, who is granted the right to access this account and to perform any rcp/rsh commands using this account.

Adding an entry into the UNIX /ETC/HOSTS.EQUIV file allows the Local user access to all login accounts on that host.

Adding an entry into the .rhosts file allows the Local user access only to this specific account. The file, belongs to the Remote user's login account and resides in the user's home directory, with the other "." (dot) files. The owner of the specific login account may need to create the .rhosts file.

If the Local user's name does not appear in either of these files, access to the remote login account will be denied.

To set up R-Commands, do the following:

1. Start R-Commands.
2. Choose Preferences... from the Settings menu.
3. In the UserName field, type your username.
4. In the Working dir field, specify the directory from which you want to take and/or place files. Choose the Working dir button to browse through the directories.
5. If desired, specify one or both of the Confirm On options:
    - Select the File Replace option to confirm when a file replaces an existing local file.
    - Select the List box selection delete option to confirm the deletion of a command in the command list selection region.

    By default both the Confirm On options are selected.
6. Choose the OK button.

Entering rcp Commands

The rcp command lets you copy files or directories between two systems on the network. Use the rcp command to copy files or directories from:

- your local computer to a remote system
- a remote system to your local computer
- one remote system to another remote system

To enter an rcp command, do the following:

1. Choose the Build... option from the Commands menu.
2. Choose the RCP tab.
3. In the Description field, type a name for the command you are entering.

4. Select a direction from the Transfer direction file drop-down list.

   The Transfer direction selection determines which fields (From and To) must be specified:

   - local to remote transfer: specify only the To fields
   - remote to local transfer: specify only the From fields
   - remote to remote transfer: specify both sets of fields

5. To define the From field, do the following:

   - In the From Name field, select or type your username for the machine from which you plan to copy the file.

     The UserName you entered when setting up R-Commands is listed in the drop-down list.

   - In the From Host field, select or type the name of the host on which the file or directory you want to copy resides.

     If you plan to copy a file or directory from a remote system, include a backslash (\), quotation mark (") or apostrophe (') in the remote host's path. This ensures the metacharacters of the remote system are interpreted remotely.

   - In the From File field, enter the name of the file or directory you want to copy

     For a local PC, the machine on which your working directory is located is the default. You specify your working directory as discussed in "Setting Up R-Commands."

6. To define the To field:

   - In the To Name field, type the username assigned to you on the host to which you want to copy the file or directory.
   - In the To Host field, select or type the name of the host to which you want to copy the file or directory.

   Note: If you plan to copy a file or directory to a remote system, include a backslash (\), quotation mark (") or apostrophe (') in the remote host's path. This ensures the metacharacters of the remote system are interpreted remotely.

   - In the To File field, enter the file or directory to which you want to copy.

     For local connection you can choose the browse button to help you locate and select the desired file.

7. If desired, specify either or both of the rcp command options.

   - Select the **-**r (copy each subtree) check box to copy any subdirectories rooted at the directory of the original file for the remote to local or remote to remote.

- Select the -p (attributes) check box to assign the original file's modification time, access time and modes to the duplicate file.

8. Choose the OK button to save the command to the commands list so that later you can execute it or add it to a Command Profile.

9. Choose the Run button to execute the command immediately without saving it to the commands list.

Entering rsh Commands

The rsh command establishes a remote host connection and executes a specified UNIX command or a batch file containing multiple UNIX commands. Output from rsh is displayed in the R-Commands window's output region unless you redirect the output to a file.

To enter an rsh command, do the following:

1. Choose the Build... option from the Commands menu.

2. Choose the RSH tab.

3. In the Description field, type a name for the command you are entering.

4. In the Shell command field, type the UNIX command you plan to execute on the remote host. For example, the ls UNIX command lists directories. For details on UNIX commands, see your UNIX documentation.

   The From Name, Host and File fields are inapplicable, and therefore dimmed.

5. In the To Name field, type the username assigned to you on the host to which you want to establish a connection.

   In the To Host field, select or type the name of the remote host to which you want to establish a connection.

   Type the file's complete path in the Output field to send output to a file rather than display it in the R-Commands window's output region. Select the local path check box to send an output to a local file. Selecting the local path check box enables the browse button, to help you locate and select the file.

6. If you redirect output to an existing file and want to overwrite that file's contents, select the rewrite an existing file check box.

   Note: If the rsh command causes conflicts between rsh and the shell on the remote host, select the -n option. This redirects the input of rsh to /DEV/NULL.

7. Choose the OK button to save the command to the commands list, so that later you can execute it or add it to a Command Profile.

8. Choose the Run button to execute the command immediately without saving it to the commands list.

Creating Command Profiles

Command Profiles, which are similar to batch files, contain a set of rcp and rsh commands that are executed in a series. Create a Command Profile using the rcp and rsh commands that were previously saved to the commands list.

You can execute a Command Profile immediately after you create it, or you can save it to the commands list, in the main window, and execute it later. The commands are executed in the order they are listed in the profile.

To create a Command Profile, do the following:

1. Choose the Build... option from the Commands menu.
2. Choose the Profile tab.
3. In the Description field, type a name for the Command Profile you are creating.
4. Add commands to the Profile commands box in the order in which you want them executed. To do so, select a command in the List of commands box and a command in the Profile commands box. Then do one of the following:
   - Choose the Before button to add the command before the one currently selected in the Profile commands box.
   - Choose the After button to add the command after the one currently selected in the Profile commands box.

   You can delete a command from the profile by selecting the command in the Profile commands box and then choosing the Delete button.
5. After you have added all desired commands to the profile, perform one of the following operations:
   - Choose the OK button to save the profile to the commands list so you can execute it later.
   - Choose the Run button to execute the profile immediately without saving it to the commands list.

The bottom status bar displays details about the highlighted command.

Executing Commands List Items

To execute commands or Command Profiles previously saved to the commands list do the following:

1. Select the desired command or profile from the R-Commands window's commands list.

2. Choose the Run button, or double click the command.

An executed command or Command Profile's output is displayed in the output region of the R-Commands window. You can copy this information to another application, delete it or print it.

To copy the information to the clipboard, select it and choose the Copy option from the Edit menu. You can then copy the information from the clipboard to another application.

To delete the information displayed in the output region, choose the Clear Buffer option from the Edit menu.

To print the information, select it and then choose the Print... option from the File menu.

Modifying Commands or Command Profiles

You can modify a command or Command Profile by performing either of the following operations:

- Select a command and edit it in the Command field of the R-Commands window.
- Select it from the commands list and then choose the Modify... option from the Commands menu. A Modify dialog box appears. This dialog box contains the same fields as the Build dialog boxes. For details on a command or profile's fields, see the previous section in this chapter that discusses that command or profile.

Deleting Commands or Command Profiles

To delete a command or Command Profile from the R-Commands window's commands list do the following:

1. Select the command or profile from the commands list.

2. Select the Delete option from the Commands menu.

Stopping Command Profiles

You can stop a Command Profile while it is running.

To stop a Command Profile while it is running, select the Abort option from the Commands menu.

Additional Online Information

The online help file for R-Commands includes a description of each menu command, plus the following information in the "How to Use" section:

- Setting up R-Commands
- Entering RCP Commands
- Entering RSH Commands
- Creating Command Profiles
- Executing Commands List Items
- Command Line Options

Command Line Options

You can use command line options to automatically connect to a host, run a command, or run R-commands minimized.

To do this, from the Program Manager, select the File menu, then the Run command and type:

rcmd.exe [option]

You can use the following options:

-c<command name>

Use specified command from the command list.

For example:

c:\netmanag\rcmd.exe-c weekly

Invokes R-Commands application and executes the command weekly, as defined in the command list.

-p<profile name>

Use specified profile from the command list.

For example:

c:\netmanag\rcmd.exe-p backup

Invokes R-Commands application and executes all the commands listed under the backup profile, as defined in the command list.

-i

Runs the R-Command application minimized.

For example:

```
c:\netmanag\rcmd.exe-i
```

Troubleshooting

If you experience difficulties using the R-Commands application, refer to the following items:

- ❏ Verify that the installation and setup steps have been successfully completed.
- ❏ Use the Diagnose Connection option in the Help menu to verify connectivity to any remote system to which you want access if a failure to connect is reported.
- ❏ Interactive programs such as word processors and spread sheet programs should not be run in R shell.
- ❏ Make sure you only copy individual files and not entire directories from Local to Remote Hosts (including use of the -r or *.* commands).

Chapter 44. Modem Server

The Modem Server lets you use one of your PCs as a modem server for other PCs on the same LAN. Rather than configuring separate modems on each of your PCs, you can centralize your physical access to the modem to one PC, which has a modem and is configured with Modem Server. The other PCs can connect to the Modem Server PC through any Telnet client.

Note: You may connect to the Modem Server through any Telnet client on any platform, including UNIX, Macintosh, and other.

In this way, the other PCs connect to a modem that is available through the Modem Server PC and dial through that modem to external hosts (such as BBS). If you have access to a Modem Server over the Internet, you can save costs on international calls by using the Modem Server for those connections.

Preparing Your Machine to Act as a Modem Server

To make your machine available as a Modem Server for other users, do the following:

1. Connect 1-4 modems to the PC that will be functioning as the Modem Server.
2. Open your Modem Server application.
3. Select a Listening Port to be used for the connection by choosing the Listen command from the Connection menu, or use the default port setting, port 23.
4. Specify desired settings by choosing commands from the Settings menu (such as, specifying the baud rate for your COM ports by using the Asynchronous Preferences command).
5. After you have completed your settings, you can make your Modem Server COM port(s) available to be accessed by other users by choosing the COM Port Connecting command from the Connection menu. Users cannot connect to a port until you make it available by using this command.

Note: If you change the settings for a port that is currently busy, the changes will take effect only after the port has been released.

6. Notify potential users of your machine's IP address or host name, and listening port.
7. You may minimize the application after opening it.

Connecting to a Modem Server

To make a connection to a Modem Server, do the following:

1. Using the Telnet and ASCII application, specify the IP address or host name of the Modem Server as the host to which you wish to connect.

2. Select the appropriate port (usually 23).

3. When you have connected to the Modem Server host, a short menu of commands will be displayed.

Note: When you first connect to the Modem Server host, you are not actually connected to a port. You cannot dial out or connect to another host through the Modem Server, until you have used one of the connect or dial commands listed in the Welcome menu.

4. Type the letter of the command you wish to use. You should use the "l" command before trying to make a connection, so that you can find out what ports and modems are currently available for use.

Additional Online Information

The online help file for the Modem Server includes a description of each menu command, plus further information in the "How to Use" section.

Troubleshooting

If you experience difficulties using Modem Server, refer to the following items:

- ❑ If you get disconnected without notice when trying to connect, check that you have opened the port for use in Modem Server and connected to the modem from the Preferences dialog.
- ❑ If you receive a NO CARRIER message when trying to dial out, check that the line is connected to the modem. Do not press the Enter key after choosing d for dialing. Type only the telephone number followed by the Enter key.
- ❑ Increase the size of the Port Receive Buffer in the Asynchronous Communications Settings dialog, if you are getting too many general errors when connected.

Chapter 45. Serial to Telnet

Note: This application is only available for Windows 3.1 Operating System.

The Serial to Telnet application lets you use any serial terminal emulation application (such as, the Windows Terminal application) in your system to communicate over Telnet (TCP/IP). The Serial to Telnet application includes the following main features:

- Lets you redirect specific COM ports to Telnet
- Emulates a Hayes™-compatible modem.
- Can be used to access network modems.
- Can provide Telnet connections on up to 8 COM ports (COM1-8).

Redirecting COM Ports to Telnet

The Serial to Telnet application has only one main dialog box, which lets you select and configure the COM ports that you want to redirect to Telnet.

1. Select one or more COM ports (COM 1-8) to redirect to Telnet.
2. Select the desired Terminal Type emulation for the port(s) you have chosen.
3. If you would like to assign additional, optional characteristics to a selected port, choose the Advanced button. For a description of the Advanced options, see the table in the following section, "Advanced Configuration Options."
4. When you have completed the configuration of the ports, choose the OK button in the Serial to Telnet Configuration dialog box and close the application.
5. A prompt will be displayed, asking you if you wish to restart Windows at this time. When Windows restarts, the COM ports will be redirected as you specified.
6. To stop redirecting the ports, you should uninstall the configuration by choosing the Uninstall button from the Configuration dialog box.

Advanced Configuration Options

You may assign the advanced configuration options to your redirected COM ports.

Selecting COM Port Numbers

It is recommended that you select all port numbers which you are not using, so that you can have the maximum number of simultaneous Telnet connections. However, note the following guidelines:

- Do not select ports that do have devices attached (such as a modem or a serial mouse) or else you be unable to access that port while the Serial to Telnet Configuration is installed.
- Because of a limitation in how Windows operates in 386-Enhanced mode, most machines will only be able to use COM1-COM4. You may be able to use the higher port numbers if you have a multi-port board that comes with a 386 virtual device driver, which supports port numbers higher than 4.

Configuring Your Communications Program for Serial to Telnet

To interface between your serial communications program and the Serial to Telnet application, do the following:

- Instruct your communications program to use one of the COM ports that you selected in the Serial to Telnet Configuration.
- Configure your communications program to use one of the Hayes™-compatible modems (Hayes Smartmodem, OPTIMA, or ULTRA).
- Set the communications parameters in your communications program to any baud rate, with 8 bits, no parity, and 1 stop bit.
- Instead of a phone number, you specify either the IP address or host name of the host to which you wish to connect.

Removing the Redirection of COM Ports

To stop redirecting the ports, you should uninstall the configuration as follows:

1. Open the Serial to Telnet application.
2. Choose the Uninstall button from the Serial to Telnet Configuration dialog box.
3. Close and restart Windows.

Additional Online Information

The online help file for Serial to Telnet includes a description of each menu command, plus further information in the "How to Use" section.

Troubleshooting

If you experience difficulties using the Serial to Telnet application, refer to the following items:

❑ Because of a limitation in how Windows operates in 386-Enhanced mode, most machines will only be able to use COM1-COM4. You may be able to use the higher port numbers if you have a multi-port board that comes with a 386 virtual device driver, which supports port numbers higher than 4.

❑ On some machines, you may run into problems when redirecting COM3 and COMr. Certain terminal programs may either give unnecessary warnings or refuse to access the port. This may be caused by the terminal program’s attempts to detect and avoid possible hardware conflicts. To overcome this type of problem, add the following lines to the [386Enh] section of the SYSTEM.INI file:

```
COM3IRQ=11
COM3BASE=3E8
COM4IRQ=15
COM4BASE=2E8
```

Caution: If any of the above lines already exist in the [386Enh] section of the SYSTEM.INI file, especially if you have real hardware attached to COM3 and/or COM4, do not replace them with the above lines. Doing so may cause you to be unable to access the physical port.

Note: If adding the above lines does not solve your problem, you may find it necessary to adjust the numbers provided in these lines.

❑ Trouble with host name: You should instruct your communications program to use the IP address or host name of the host to which you wish to connect (instead of specifying a telephone number as you would for a serial connection). If you specify a hostname, such as ftp.netmanage.com, some terminal programs may try to substitute the Touch-Tone (digit) equivalents for each letter in the host name. In this case, you will need to use the IP address.

Also, if your host name begins with "t" or "p", you may need to insert either a space or a comma in front of the name in order for it to work properly.

❑ Trouble with IP address: Some terminal programs strip out the period in an IP address, which results in incorrect dialing. If this happens, remove the periods from the IP address and expand each portion of the IP address to 3 digits. For example, the address 11.222.44.1 would be changed to 011222044001.

If you want to connect to a port on the remote host other than the standard Telnet port (port number 23), add the port number in brackets after the IP address. For example, specifying the number 11.222.44.1[25] would connect you to the SMTP server (port 25) on that host.

Chapter 46. NISLookup

The NISLookup application, which uses the Network Information Service (NIS) protocol, displays information retrieved from NIS including map entries, domain and server names.

Note: You can use this application only if you are running the Windows 3.1 operating system.

NIS maintains centralized network administrative information in tables or maps. You can select a map name from the map list or enter the name manually. Some of the more common NIS map names, such as host and password maps, are included in the list.

NIS domains consist of NIS clients and servers sharing common information. NIS clients request information from a domain, rather than from a server. This allows server configuration to occur dynamically, which is completely transparent to the clients.

Looking Up Map Entries

To look up an NIS map entry, you must specify the map name in the Map field and key name in the Key field and choose the Lookup button. The corresponding map entry is then retrieved from NIS and shown in the display window. For example, a host name and IP address can be retrieved.

Retrieving Entire NIS Maps

You can retrieve a host map using NIS. To retrieve an entire NIS map:

Specify the map name in the Map field and choose the Retrieve button. The entire map is retrieved from NIS and displayed.

Retrieving NIS Domain Names

To retrieve the NIS domain name:

Select the Domain option from the Settings menu.

To configure the NIS domain name, use the Custom application. Refer to the NetManage Configuration Guide for additional information on configuring the NIS client.

Retrieving the Domain NIS Server Name

You can retrieve the NIS server name for the domain by selecting the Server command from the Settings menu. Although an NIS domain may contain multiple NIS servers, only one server, the first to respond, is used.

Additional Online Information

The online help file for NISLookup includes a description of each menu command, plus the following information in the "How to Use" section:

- Looking Up Map Entries
- Retrieving Entire NIS Maps
- Retrieving NIS Domain Names
- Retrieving the Domain NIS Server Name

Troubleshooting

If you have problems using the NISLookup application, refer to the following items:

❑ Verify that an NIS domain has been specified in Custom. Contact your System Administrator for the NIS domain name. From a UNIX host, you can retrieve the domain name using the Domain Name command.

❑ Verify that an NIS server is serving the specified NIS domain. From a UNIX host, you can retrieve the server using the ypwhich command. You can use the Portmapper application to determine if the ypserv program is running on the NIS server.

❑ Verify that you can connect to the NIS server using the Ping application.

❑ Verify that the NIS server resides on the same subnet as your machine.

Appendix A. MultiLAN Driver

Note: Read this appendix if you are running NFS in Windows 3.1 and you want to switch between different networks. The information in this appendix does not apply to Windows NT or Windows 95 users.

Chameleon ATX lets you switch dynamically between different networks installed in Windows without having to exit Windows.

The Windows Setup program lets you select only one network that you can connect to Windows. To switch between Novell and NFS, you have to run Setup again and restart Windows for the change to take affect.

With NetManage's MultiLAN driver, you can add and remove networks dynamically without having to exit Windows to initialize your selection. MultiLAN currently supports NetManage's Chameleon ATX, Novell's NetWare, Banyan Vines, Windows for Workgroups, and LAN Manager networks.

Note: We recommend that you install all required networks, such as Novell's NetWare, through the Windows Setup program, prior to the installation of Chameleon ATX. Adding networks after the installation of Chameleon ATX will necessitate re-installing the Chameleon ATX product.

Because certain Windows operations depend on the type of network installed, select one of the networks installed to be your default network. To define a default network, do the following:

1. Start the Control Panel.
2. Choose the Network icon. If the Microsoft Windows Network dialog box appears, choose the Other Net button.

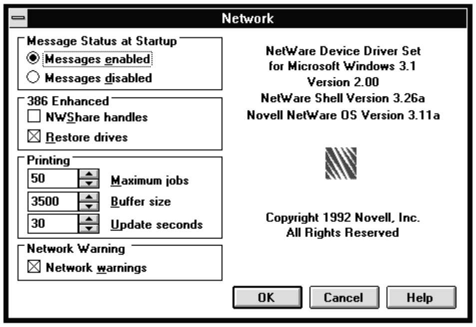
   The Networks dialog box appears.
3. Select the network you want to set as your default network from the Installed Networks box.
4. Choose the Set As Default Network button.

   The name of the network will appear in the Default Network box.

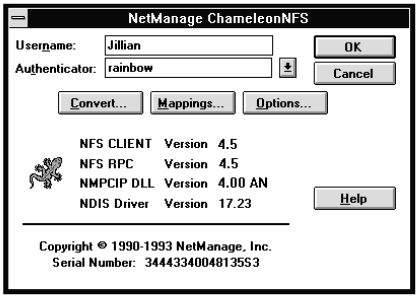
   Operations such as Browse inside File Manager, will use the appropriate functions as supplied by your Default Network.

5. Choose the Network button... to display a dialog box that describes different facilities of the network selected in the Installed Networks box. The dialog box is designed by the vendor of that specific network.

   The following figure is an example of the dialog box provided by Novell for the NetWare Driver:



   The following figure is an example of the dialog box provided by NetManage for the Chameleon ATX network:



Adding Networks

Note: Before adding a network, you should install the drivers for your new network through the standard Windows Setup program.

To add networks, do the following:

1. Start the Control Panel.

2. Choose the Network icon. If the Microsoft Windows Network dialog box appears, choose the Other Net button.

   The Networks dialog box appears.

3. From the Networks dialog box, choose the Add>> button.

The dialog box expands to include a list of available networks from which to choose.

4. Select a name from the list, then choose the Install... button.

   The new network name will be added to the list of Installed Networks.

Removing Networks

To remove networks, do the following:

1. Start the Control Panel.
2. Choose the Network icon. If the Microsoft Windows Network dialog box appears, choose the Other Net button.

   The Networks dialog box appears.

3. From the Networks dialog box, select the network you want to remove from the list of installed networks and choose the Remove button.
4. Choose the OK button.

Appendix B. Removing Your Product

How you remove your NetManage product depends on the environment in which NEWT is installed. This appendix describes how to remove a NetManage product from Windows NT, Windows 95, or Windows 3.1.

Removing a NetManage Product from Windows NT

You can use the NetManage Uninstaller program to remove part or all of your NetManage product from the Windows NT environment.

To remove your NetManage product from the Windows NT environment, do the following:

1. If the NetManage product's folder (directory) contains files that you added after installing that product, back up those files.
2. Start Uninstaller by doing the following:
   a) In Program Manager or File Manager, choose the Run command from the File menu and enter the command that starts the Uninstaller program (UNINST.EXE).

      UNINST.EXE is located in the directory in which you installed your NetManage product. For example, if the product is installed in NETMANAG on your C drive, start Uninstaller by entering the following command:

      ```
      C:\NETMANAG\UNINST.EXE
      ```

   b) Choose the OK button
3. Choose the Remove button.

   The Remove Option dialog box appears.
4. Do one of the following:
   - To remove only specific applications, choose the Custom option button and then the Next button. Select the check boxes of the applications you want to remove and choose the Next button.
   - To remove all NetManage applications currently installed on your system, choose the Complete option button and then the Next button. When prompted to verify that you want to delete all your NetManage product's files, choose the Yes button.

Caution: Choosing the Complete button deletes the folder in which you installed your NetManage product and all of that folder's contents.

If an application's files are not in the directory in which your product is installed, the application's command will not be removed from the appropriate menu and some items (such as an application's files and shortcuts) are not removed.

5. If you removed all of the NetManage applications, remove the NetManage directory from the path specified in your NT registry.

Removing a NetManage Product in Windows 95

You can use the NetManage Uninstaller program to remove part or all of your NetManage product from the Windows 95 environment.

Note: If you used the Windows 95 registry editor to change or delete the Path key for the folder containing your NetManage product, you will not be able to run the Uninstaller program.

To remove your NetManage product, do the following:

1. If the NetManage product's folder (directory) contains files that you added after installing that product, back up those files.
2. Start Uninstaller by doing the following:
   a) Open the Control Panel and choose the Add/Remove Programs icon.
   b) From the Install/Uninstall tab, select NetManage. Then choose the Add/Remove button.
3. Choose the Remove button.

   The Remove Option dialog box appears.
4. Do one of the following:
   - To remove only specific applications, choose the Custom option button and then the Next button. Select the check boxes of the applications you want to remove and choose the Next button.
   - To remove all NetManage applications currently installed on your system, choose the Complete option button and then the Next button. When prompted to verify that you want to delete all your NetManage product's files, choose the Yes button.

   Caution: Choosing the Complete button deletes the folder in which you installed your NetManage product and all of that folder's contents.

If an application's files are not in the directory in which your product is installed, the application's command will not be removed from the appropriate menu and some items (such as an application's files and shortcuts) are not removed.

5. If you were using the NetManage TCP/IP stack (NEWT) and want to remove it, delete the lines from the TCPIP section of your WIN.INI file.

6. If you removed all of the NetManage applications, remove the NetManage directory from the path specified in your AUTOEXEC.BAT file.

Removing a NetManage Product from Windows 3.1

To remove your NetManage product from the Windows 3.1 environment, do the following:

1. Edit the files that were modified during the installation process so that those files reflect that the NetManage product will no longer be available.

   SETUP.EXE modifies some files when you install a product. The files it modifies and the changes it makes to those files depends on whether you installed the product in an NDIS, ODI, or Windows for Workgroups environment.

   For information about the changes made in the NDIS environment, see the Files Modified in the Windows 3.1 NDIS Environment section of this appendix.

   For information about the changes in the Windows 3.1 ODI or Windows for Workgroups environment, see the appropriate appendix in your NetManage Configuration Guide.

2. Reboot your system.

3. Delete the NETMANAG directory and all of its contents.

Files Modified in the Windows 3.1 NDIS Environment

When you install a NetManage product in the Windows 3.1 NDIS environment, changes are made to the following files: AUTOEXEC.BAT, CONFIG.SYS, WIN.INI, SYSTEM.INI, and PROTOCOL.INI. If you want to remove your NetManage product from the Windows 3.1 environment, modify these files so they reflect that the NetManage product has been removed.

The following examples for an NDIS installation assume that the C:\NETMANAG directory was specified and a 3COM EtherLinkII (3C503) network adapter was selected during the installation process. Your files may differ slightly, depending on the network adapter you actually select.

AUTOEXEC.BAT Changes

- The following line is inserted at the beginning of your PATH variable in your AUTOEXEC.BAT file:

  ```
  PATH=C:\NETMANAG;
  ```

- The following line is added to your AUTOEXEC.BAT file:

  ```
  C:\NETMANAG\NETBIND
  ```

Note that the NETBIND.COM program is not unique to your NetManage product; it is actually the standard Microsoft program used in LAN Manager and other applications.

CONFIG.SYS Changes

The following lines are added to your CONFIG.SYS file:

```
DEVICE=C:\NETMANAG\PROTMAN.DOS /I:C:\NETMANAG
DEVICE=C:\NETMANAG\ELNKII.DOS
DEVICE=C:\NETMANAG\NETMANAG.DOS
```

Note that the Protocol Manager (PROTMAN.DOS) and the NDIS driver (ELNKII.DOS) are not unique to your NetManage product; they are actually the standard Microsoft driver for LAN Manager and the NDIS driver from your network adapter vendor.

The Protocol Manager driver entry must use the /I: parameter to point to the location of the PROTOCOL.INI file.

WIN.INI Changes

The following lines are added to your WIN.INI file. The WIN.INI file’s TCP/IP section may also include information about NetManage applications you started.

```
 [TCPIP]
FILE=C:\NETMANAG\TCPIP.CFG
ID=XXXX
```

Caution: Back up your TCP/IP section in the WIN.INI file because it contains vital information about your NetManage product.

SYSTEM.INI Changes

- The following lines are added to your SYSTEM.INI file:

  ```
   [BOOT]
  NETWORK.DRV=C:\NETMANAG\mult46.DRV

  [BOOT DESCRIPTION]
  NETWORK.DRV=NETMANAGE CHAMELEON ATX

  [386Enh]
  DEVICE=C:\NETMANAG\NMTCPIP.386
  DEVICE=C:\NETMANAG\NMREDIR.386
  ```

- The following directory is created:
  `C:\NETMANAG`
- The following files are copied to your C:\NETMANAG directory:

```
PROTOCOL.INI
PROTMAN.DOS
NETMANAG.DOS
NETBIND.COM
Applications and Help files
NDIS driver for your specific network internet card
```

PROTOCOL.INI Changes

The following lines are added to your PROTOCOL.INI file:

```
 [EtherLinkII]
DRIVERNAME=ELNKII$
INTERRUPT=3
TRANSCEIVER=OnBoard
IOADDRESS=0x300

[NETMANAGE]
DRIVERNAME=NETMNG$
BINDINGS=EtherLinkII
```

The EtherLinkII section describes values for your network adapter. These values must match those configured for your network adapter either using hardware jumpers, or a software setup program supplied with the network adapter.

The NETMANAGE section describes a network protocol for NEWT. The BINDINGS statement binds the protocol to your network adapter. You may specify additional bindings for a protocol if you have more than one network adapter section.

Glossary

access	Entry to or communication with a particular object, such as an operating system, specific files, or accounts.
account	An entity which is established as an authorized user of the system.
address mask	A bit mask used to select bits from an IP address for subnet addressing. The mask is 32 bits long and selects the network portion of the IP address and one or more bits of the local portion.
ANSI	The American National Standards Institute sets standards for the US computer industry. ANSI participates in defining network protocol standards.
API	Application Program Interface. A standard interface that allows upper-layer applications to work with different communication protocol stacks. Some of the most commonly used ones include NetBIOS, Berkeley Sockets, and Named Pipes.
application	An application is a computer program that performs a certain task. FTP, Telnet, TN3270 are some of the applications provided by NetManage.
ARP	Address Resolution Protocol. The TCP/IP protocol used to dynamically bind a high-level IP address to a low-level physical hardware address. ARP is only across a single physical network and is limited to networks that support hardware broadcast.
ASCII	The American Standard Code for Information Interchange, widely accepted code for representing alphanumeric information.
authenticator	A password or code that verifies the identity of individual system or network users.
AUTOEXEC.BAT	A file that resides on all PCs and contains a series of commands that are executed when the computer starts up. This file includes the path command that points to the NETMANAG directory where the applications reside.

backup — The process of preserving copies of files on a different drive, directory, or media to protect against the destruction or loss of the original files.

Bind — The Bind application provides name server functions, such as a Domain Name Server (DNS). You can specify multiple domains and a list of host names within each domain. (See also Domain Name System.)

buffer — A temporary storage area for data during the transfer of that data between the computer and a peripheral, or between parts of a computer, to prevent loss of information.

client — A computer system that uses resources provided by another machine on the network. Most of NetManage's applications can run as both client and server.

command line — The entire command string, including the command and any parameters or qualifiers that it may have. A command is an instruction or request for the system to perform a particular action.

CONFIG.SYS file — This file resides on all PCs and defines which device drivers to install.

daemon — An agent program that runs continuously on a server system in a UNIX environment and provides resources to client systems on the network.

DLL — Dynamic Link Library. Windows automatically loads the applications into memory when required and unloads it when space is needed for other applications. The Chameleon for Windows 95 applications are 100% DLL.

default — A value supplied by the system when a user does not specify a required command, parameter, or qualifier.

device name — Identification of a physical device: for example, LPT1 for a printer. Can also be a logical name that is equated to a physical device name.

DNS — Domain Name System. An online distributed database that maps machine names into IP addresses. (See also Bind.)

domain — A named group of machines on the network. A domain name consists of a sequence of names (labels) separated by periods (dots).

driver A software module that controls an input/output port or external device such as a keyboard or a monitor. TCP/IP uses a driver to control the network interface cards.

Ethernet Ethernet is a type of network that supports high-speed communication among systems.

Ethernet address A six-part hexadecimal number in which a colon separates each part (for example, 8:0:20:1:2f:0). This number identifies the Ethernet communications board installed in a PC and is used to identify the PC as a member of the network.

export The process that makes a file available so that other systems can access it.

file access Allows users to work with a remote file as if the file were local.

file server A process running on a computer that provides access to files on that computer to programs running on remote machines.

Finger A standard protocol that lists who is currently logged in on another host.

FTP File Transfer Protocol. The FTP application is used to provide file transfer services across a wide variety of systems. Usually implemented as application-level programs, FTP uses the Telnet and TCP protocols. The server side requires a client to supply a login identifier and password before it will honor requests.

group Id A unique number associated with each group name on the server.

HLLAPI High-Level Language Application Program Interface, which you can use to program in high-level languages such as COBOL, Pascal, BASIC, or C.

host Any end-user computer system that connects to a network. Hosts range in size from personal computers to supercomputers.

host table ASCII text file in which each line is an entry consisting of one numeric address and one or more names associated with that address.

HTML Hypertext Markup Language is the document formatting language used by World Wide Web browsers. It enables text formatting, embedded pictures, and hypertext links to other documents and different locations within documents.

ICMP Internet Control Message Protocol. The ICMP delivers error and control messages from hosts to the requesters. An ICMP test can determine whether a destination is reachable and responding.

IMAP Internet Message Access Protocol. Defines a way for mail programs to access mailboxes on remote computers as if they were local. Includes operations for creating, deleting, and renaming mailboxes; checking for new messages, and permanently removing messages. It also allows users to share remote folders (mailboxes), to connect from multiple locations, and see a consistent mailbox structure and content.

Internet When capitalized, the world-wide network of networks connected to each other using the IP and other similar protocols. The Internet provides file transfer, remote login, electronic mail and other services. When not capitalized, any collection of distinct networks working together as one.

Intranet A private enterprise network that uses TCP/IP standards-based networking technologies for host access, workgroup collaboration, desktop and network resources management, and developer tools for custom applications to maximize the enterprise’s productivity. For example, Web began as an Internet application and has now been incorporated into internal company applications.

IP The TCP/IP standard protocol defined as a unit of information passed across the Internet, providing the basis for packet delivery service. IP includes the ICMP control and error message protocol as an integral part. The entire protocol suite is often referred to as TCP/IP because TCP and IP are the two most fundamental protocols.

IP address Internet Protocol address. This is a 32-bit address assigned to host on a TCP/IP Internet. The IP address has a host component and a network component.

IPX/link The IPX/link application for NetWare connects your PC Novell NetWare LAN through the Network Device Interface Specification (NDIS) developed by Microsoft and 3COM.

Java A programming language. It can also be a program that can be included in an HTML page on the Web.

log in To perform a sequence of actions at a terminal that establish a user's communication with the operating system and sets up default characteristics for the user's terminal session.

log out To terminate interactive communication with the operating system, and end a terminal session.

LPR/LPD Line Printer Remote/ Line Printer Daemon. An application that allows you to print to network printers.

Mail server A host and its associated software that offer electronic mail reception and forwarding service. Users may send messages to, and receive messages from, any other user in the system.

MIB Management Information Base. The set of variables that a gateway running SNMP maintains.

MIB-II An extended management database that contains variables not shared by SNMP.

MIME Multi-purpose Internet Mail Extensions. MIME enables: files to be attached to mail messages; multiple objects in a single message; an unlimited line length or overall length for text; character sets other than ASCII; multi-font messages, binary or application specific files; images, audio, video and multi-media messages.

mount NFS user command that makes remote file systems and resources available to the local PC.

NDIS Network Device Interface Specification. NDIS is used for all communication with network adapters. The specification was developed by Microsoft and 3COM to provide a common programming interface for MAC drivers and transport drivers. NDIS works primarily with LAN Manager and allows multiple protocol stacks to share a single network interface card.

NetBIOS Network Basic Input/Output system. Provides a Session Layer interface between network applications running on a PC and the underlying protocol software of the Transport and Network Layers.

NETBUI The NetBOIS Extended User Interface. This is the transport layer driver frequently used by LAN Manager.

NetWare	A network operating system developed by Novell.
NetWare IP	Feature offered by Novell to allow Windows clients to access Novell servers using the Internet Protocol (IP). Once NetWare IP is installed, you can access all standard NetWare server features normally available from Windows.
network address	A unique number associated with a host that identifies it to other hosts during network transactions.
network printing	Printing to a shared printer locally attached to one of the PCs on the network.
NEWT	NetManage TCP/IP communication stack for Microsoft Windows. NEWT provides users with a degree of network access previously only available to workstation and mainframe users.
NFS	Network File System. A protocol developed by Sun Microsystems that uses IP to allow a set of computers to access each other’s file systems as if they were local. Originally designed for UNIX systems, this protocol has been implemented on many other operating systems, including DOS and Windows.
NIC	Network Information Center. The NIC at SRI in Menlo Park, California, assigns IP addresses and network numbers by request. The number assigned is appropriate to the number of host devices on the network.
OfficeVision	An electronic mail application that runs on an IBM mainframe. Z-Mail® Pro supports OfficeVision transport that allows you to exchange electronic mail messages and calendar information between your PC and the IBM mainframe.
PCNSD authenticator	A program that runs continuously and authenticates a user name and password before attempting to mount a network drive, access files, or print on the network.
Ping	The Packet Internet Groper is a program that is useful for testing and debugging networks. It sends an echo packet to the specified host, and waits for a response. It reports success or failure and statistics about its operation.
Plug-in	A third-party embedded application that you can launch from within its "host" application.

POP Post Office Protocol. This protocol is used by mail applications to retrieve electronic mail services from the Internet.

PROFS An electronic mail application that runs on an IBM mainframe. Z-Mail® Pro supports PROFS transport that allows you to exchange electronic mail messages and calendar information between your PC and the IBM mainframe.

prompt Word or words used by the system to assist a user’s response. Such messages generally ask the user to respond by typing some information in a supplied field.

RAM RAM is Random Access Memory.

remote Files, devices, and users not attached to your local machine.

remote host The computer receiving a network command.

remote printer In LPR/LPD, a printer with a special network card, or a PC or workstation.

RFC Request For Comment. The RFC documents describe all aspects and issues associated with the Internet protocols.

router A router has two or more network interfaces to different networks. The primary function of a router is to direct packets between these networks, delivering them to their final destination or to another router. When used with TCP/IP, the term refers to an IP gateway that routes data using IP destination addresses.

RPC Remote Procedure Call. A mechanism defined by Sun Microsystems that provides a standard for initiating and controlling processes on remote or distributed computer systems.

script A sequence of ASCII text lines stored in a file.

server A computer that provides services to a network.

SMTP Simple Mail Transfer Protocol. A protocol used by mail applications to send and retrieve electronic mail messages from the Internet.

SNMP Simple Network Monitoring Protocol. A standard protocol used to monitor network activity on "agent" nodes from management stations.

subnet	A field used by routers and hosts for routing packages on the network.
subnet address	An extension for the IP addressing that allows a site to use a single IP network address for multiple physical networks.
subnet mask	Identifies the subnet field of a network address. The subnet mask is a 32-bit Internet address written in dotted-decimal notation with all the 1s in the network and subnet portions of the address.
TCP/IP	Transmission Control Protocol/Internet Protocol. TCP allows a process on one machine to send data to a process on another machine using the IP protocol. TCP can be used as a full duplex or one-way simplex connection.
Telnet	Provides virtual terminal services for a wide variety of remote systems using the Telnet protocol. The application allows a user at one site to interact with a remote system at another site as if the user's terminal were connected to the remote machine.
terminal emulator	A program that makes a PC screen and keyboard act like a video display terminal of another computer.
TFTP	Trivial File Transfer Protocol. A standard TCP/IP protocol that allows simple file transfer to and from a remote system without directory or file listing. TFTP is used where FTP is not available.
token-ring	A type of ring-shaped network that supports high-speed communications between computers. A distinguishing packet, called a "token," is transferred from machine to machine. Only the machine that holds the token can transmit the packet.
TSR	Terminate-and Stay-Resident. A DOS program that is loaded into memory before Windows and stays in memory until the machine is rebooted.
user id	A unique number, created by your system, that is associated with each user name on a server system.
user name	A character string, usually assigned by the system administrator that identifies a user on the system.
utility	A command or operation that works at the level of the operating system.

Weighted Fair Queuing (WFQ)

ChameleonNFS feature that can guarantee bandwidth to specified IP traffic. WFQ is particularly useful for applications that require real-time performance capabilities across routed networks. Applications for which WFQ is useful include video-conferencing programs (such as NetManage's InPerson), video servers, internet telephony, real-time simulations, and other performance critical networking task software.

VRML

Virtual Reality Modeling Language. Uses a 3D rendering engine to render the image progressively in pieces. Supports the .GIF, .JPEG, .BMP, and SFImage file formats, as well as extensions such as background color.

Index

—D—

—K—

—L—

—M—

—U—

—V—

—W—

—Z—

Reader's Comment Form

Chameleon™ ATX for All Windows® Platforms User's Guide, Version 6.0

Part No. 3000-60-0696

NetManage, Inc. welcomes your comments and suggestions on the quality and usefulness of this publication. Your input is an important part of the information used for revision.

- Did you find any errors?
- Is the information clearly presented ?
- Do you need more information? if so, where?

If you find any errors or have any suggestions for improvement, please indicate the topic, chapter, and page number.

Please mail or fax your comments to:

Technical Publications, NetManage, Inc.

10725 N. De Anza Blvd., Cupertino, CA 95014, USA

Phone: (408) 973-7171 Fax: (408) 973-8272

e-mail: doc@netmanage.com

If you would like a reply, please give your name, company name, address, and telephone number below. We are committed to responding promptly to your suggestions.

We are grateful for your assistance.