

BrightWorks Help Index

[Introduction to BrightWorks](#)

[Metering](#)

[Inventory](#)

[Distribution](#)

[BrightWorks Error Messages](#)

[Using Brequest](#)

[Btrieve Error Messages](#)

[NMS Smart-Launch Support](#)

Introduction to BrightWorks

Introduction

[About BrightWorks](#)

[The Problem and the McAfee Solution](#)

[BrightWorks Features](#)

[BrightWorks Economic Benefits](#)

[Technical Support](#)

Getting Started

[Accessing BrightWorks](#)

[Exiting BrightWorks](#)

[BrightWorks Menu Bar](#)

[BrightWorks Tool Bar](#)

[Using the Keyboard](#)

[BrightWorks Help Facility](#)

[BWORKS.INI](#)

[Printer Setup and Administration](#)

[Error Handling](#)

About BrightWorks

BrightWorks empowers network administrators with the tools necessary to manage network software effectively. By offering software metering, asset management and software distribution capabilities, BrightWorks increases the effectiveness of your LAN support efforts.

BrightWorks modular design enables network administrators to structure the solution that best fits their network management needs. In addition to offering three integrated functions, BrightWorks also allows access to other McAfee management solutions. And BrightWorks open framework facilitates integration with many management systems and products.

Only McAfee has the networking experience to deliver an integrated solution as comprehensive and valuable as BrightWorks.

See Also:

[The Problem and the McAfee Solution](#)

[BrightWorks Features](#)

[BrightWorks Economic Benefits](#)

The Problem and the McAfee Solution

As LAN technology expands, networks grow rapidly and users demand additional capabilities. With this growth come expanded network software support issues. Software license compliance, driver updates, software upgrades, hardware purchases, and network security are but a few of the network software issues in question. Dealing with all of these issues dramatically increases the network administrator's workload. Unfortunately, even as networks grow and the number of users continues to expand, the number of skilled network administrators typically remains constant.

In addressing these issues, network administrators are raising questions such as:

- * How do I keep network support from becoming an overwhelming task?
- * How do I make changes to files on large numbers of workstations quickly and without downtime?
- * How can I maintain crucial network security as the network continues to expand?

Clearly, for network administrators, intelligent network software management is mission critical!

The McAfee Solution

The solution to these problems is BrightWorks! By offering integrated software management capabilities for networks, BrightWorks performs:

- * Software usage tracking and license enforcement
- * Asset inventory of hardware and software
- * Software and data distribution
- * Software security
- * Management reporting

With these software management tools, network administrators can maximize resource productivity:

- * Effective allocation of resources, software and hardware, based on actual usage
- * Increased support effectiveness enabled by active configurations
- * Accurate software purchasing based on actual usage

BrightWorks flexibility lets you tailor network management to your specific needs.

See Also:

[About BrightWorks](#)

[BrightWorks Features](#)

[BrightWorks Economic Benefits](#)

[Technical Support](#)

BrightWorks Features

BrightWorks is full of features that aid you in managing your network. The following lists the features available with each of BrightWorks main capabilities:

Software Metering

- * Flexible metering methods to give you a choice between NLM and TSR-based metering-an agent is not required to meter your DOS, Windows, or OS/2 software.
- * Easy and simple installation of the metering methods to decrease administrative burden
- * Option to prevent users from using local drives with optional workstation TSR to help you maintain network integrity
- * Real time trustee rights granting tied to application usage (masking) to control access to sensitive or critical network applications
- * Metering for DOS, Windows and OS/2 programs to ensure compliance with software license agreements
- * Real time graphical display of software usage to help you make purchase decisions
- * Queue back for metered applications to ensure license compliance and maximize the effectiveness of available resources
- * Suite metering for accurate enforcement of concurrent license agreements for suite application, such as Microsoft Office
- * Virus protection to prevent costly downtime due to corrupted files

Asset Management

- * Recording and detecting of changes in software and hardware configurations (file servers, PCs & Macs) to eliminate the need for manual inventory
- * Inventorying for multiple sites to maximize resource usage
- * Alerting of changes in software and hardware configurations via cc:Mail, MHS, e-mail and paging notification to enable timely network support and provide an added level of security
- * Auto-learning of new software to reduce the time required to input new applications
- * Detailed vendor and warranty data tracking to keep records up-to-date and to inform purchase decisions
- * Inventory information import and export capabilities to preserve compatibility with other products and databases

Software Distribution

- * Automated distribution of system files, data files and software applications to eliminate "sneaker-ware" for these functions
- * Flexible and powerful scripting language that allows you to customize distribution of system files, data files and software applications
- * Ability to edit system files (e.g., CONFIG.SYS, AUTOEXEC.BAT) to enable global replacement and workstation-specific changes without having to visit each workstation

See Also:

[About BrightWorks](#)

[The Problem and the McAfee Solution](#)

[BrightWorks Economic Benefits](#)

BrightWorks Economic Benefits

By using structured software management, you can realize the following economic benefits:

1. Reduce user support costs by

- * reducing the time required to troubleshoot user and network problems with an accurate, up-to-date hardware and software inventory,
- * minimizing potential software incompatibilities by preventing unauthorized software from running,
- * allocating training dollars to minimize support calls with detailed application usage reports by user, department or application, and
- * reducing downtime by pinpointing outdated equipment and drivers quickly and easily.

2. Reduce network maintenance costs by

- * minimizing incompatibilities from non-standard configurations or tampering users by standardizing and centrally enforcing PC software configurations,
- * eliminating the need to walk from PC to PC to update workstation software configurations (i.e., network drivers, versions of DOS, CONFIG.SYS, AUTOEXEC.BAT, etc.),
- * eliminating the need to walk from PC to PC to get an accurate hardware and software inventory,
- * minimizing downtime for users with automatic detection and reporting of changes to network hardware and software configurations,
- * minimizing theft by automatically detecting missing hardware,
- * eliminating wasted time spent trying to determine what equipment and software requires upgrades,
- * configuring new workstations identically to others in an automated fashion with pre-configured distribution packages, and
- * recovering quickly from individual PC hard drive disaster by automating the distribution of a new configuration based on records of previous distributions.

3. Cut software and hardware purchase costs by

- * purchasing software based on actual simultaneous usage rather than on counting the number of workstations attached to the network,
- * purchasing software upgrades based on actual simultaneous usage rather than on the number of packages owned, and
- * using detailed management reports to allocate workstation equipment based on actual usage (e.g. giving a 486 to a power user and a 386 to a mail user).

4. Enforce software license compliance by

- * ensuring compliance with corporate and SPA guidelines, and
- * proving software compliance with SPA-approved reports.

All of these benefits result in significant time and money savings!

See Also:

[About BrightWorks](#)

[The Problem and the McAfee Solution](#)

[BrightWorks Features](#)

Technical Support

Please take this opportunity to mail in your Product Registration Card. Mailing in your card enables us to notify you about upgrades, fixes, enhancements and new products.

If you have a problem with BrightWorks, call the McAfee Technical Support number found in the front of the BrightWorks manual.

Before requesting support, please thoroughly consult the BrightWorks manual, the READ.ME file, and any product update information included in your BrightWorks package.

When requesting support be prepared with:

- * The version and serial number of your BrightWorks product from the distribution diskette label
- * The type of network hardware and operating system software you are using
- * The version of DOS you are using
- * Any TSRs loaded, including network drivers and memory managers on your workstation
- * A clear and concise statement of your problem

If you have not mailed in your Product Registration Card you will be required to answer some questions over the telephone so our Technical Support Staff can register your product.

NOTE:

Support cannot be provided for problems related to the installation or operation of your network.

Accessing BrightWorks

After successfully installing BrightWorks, a McAfee Program Manager group and a BrightWorks program icon are created on your Windows desktop.

Use the following procedure to launch BrightWorks.

1. Load Brequest.

Either server-based or client-based Btrieve can be used with BrightWorks. Server-based Btrieve is strongly recommended due to its increased database access speed.

Upon installation, BrightWorks is configured to run with Brequest. If you are running client-based Btrieve, you must run the USEBTR.BAT file in the BWORKS directory before launching BrightWorks. This batch file configures BrightWorks to run with local Btrieve. For example, from within the BWORKS directory, issue the following command:

USEBTR <ENTER>

2. Run Windows, and double click on the BrightWorks program icon.

If you are using Novell's local Btrieve, a message displays recommending that you use BREQUEST for increased database access speed. This message will only display when BrightWorks cannot detect Brequest. It will also display when BrightWorks is configured to run with client-based Btrieve (i.e., by running the USEBTR.BAT file). To disable the warning message under all circumstances, place a checkmark in the "Disable warning message when Brequest isn't running" field.

(To re-enable the warning message, the BWORKS.INI file must be edited. Refer to [BWORKS.INI](#) for more information on the BWORKS.INI file parameters.)

Choose the OK button to continue the BrightWorks program launch. The BrightWorks application window displays.

The application window consists of the following items:

- * [BrightWorks Menu Bar](#)
- * [BrightWorks Tool Bar](#)
- * Access to BrightWorks on-line [Help Facility](#)

See Also:

[Exiting BrightWorks](#)

[Using the Keyboard](#)

[Printer Setup and Administration](#)

[Error Handling](#)

Exiting BrightWorks

Use the following procedure to end a BrightWorks session.

1. Choose the Exit command from the File menu.

A dialog box displays prompting you to confirm the exit action

2. To save your monitor configuration, check the 'Save monitor configuration' option.

Checking this option will save the configuration of any metering windows that are currently open in your BrightWorks application window. All open metering windows will be automatically restored upon starting your next BrightWorks session.

3. Choose the OK button to close the BrightWorks application.

See Also:

[Accessing BrightWorks](#)

[BrightWorks Menu Bar](#)

[BrightWorks Tool Bar](#)

[Using the Keyboard](#)

[BrightWorks Help Facility](#)

BrightWorks Menu Bar

To choose a menu, point to the menu name and click the left mouse button. The menu displays.

The general purpose of each menu item is defined below:

- * [File](#)
- * [Edit](#)
- * [Administration](#)
- * [Tools](#)
- * [Reports](#)
- * [Window](#)
- * [Help](#)

NOTE:

Holding down the left mouse button over a menu command causes the function of the command to display in the BrightWorks title bar at the top of the BrightWorks application window.

See Also:

[Accessing BrightWorks](#)

[Exiting BrightWorks](#)

[BrightWorks Tool Bar](#)

[Using the Keyboard](#)

[BrightWorks Help Facility](#)

BrightWorks Tool Bar

When using BrightWorks with a mouse, BrightWorks tool bar buttons provide an alternative for accessing the most frequently used BrightWorks functions.

Instead of choosing commands from the drop-down menus, you can choose the tool bar buttons to perform the same tasks. For example, to define applications to be metered, you can either choose Define Metered Applications from the Metering command on the Administration menu, or you can simply choose the Metering tool bar button. Both actions result in displaying the Define Metered Applications dialog box.

The function of each tool bar button is described below:

- * [Metering](#)
- * [Security](#)
- * [Monitor](#)
- * [Inventory](#)
- * [Distribute](#)
- * [Remote](#)
- * [Tickets](#)
- * [Alerting](#)
- * [Reports](#)

NOTE:

Holding down the left mouse button over a tool bar button causes the function of the button to display in the BrightWorks title bar at the top of the BrightWorks application window.

See Also:

[Accessing BrightWorks](#)

[Exiting BrightWorks](#)

[BrightWorks Menu Bar](#)

[Using the Keyboard](#)

[BrightWorks Help Facility](#)

Using the Keyboard

To use BrightWorks without a mouse, perform the standard Windows keyboard actions to navigate throughout the program.

Each menu item on the BrightWorks menu bar has a keyboard mnemonic. Press the <ALT> key in combination with the keyboard mnemonic key to choose a menu and cause the menu to drop down. For example, press the <ALT><F> keys to choose the File menu and display its commands.

Each command also has a keyboard mnemonic. Once the menu is displayed (i.e., "dropped down"), press the keyboard mnemonic of the command you want to choose. For example, from the File menu, press <P> to choose the Printer Setup command. You can also use the <up/down arrow> keys to move the highlight to a desired command and press <ENTER> to select the command.

For detailed information on using a Windows application with the keyboard, refer to your Windows documentation.

NOTE:

Some BrightWorks features require the use of a mouse and cannot be accessed with the keyboard.

BrightWorks Help Facility

BrightWorks Help facility provides on-line assistance for using the BrightWorks software. To get information quickly about a BrightWorks feature or procedure, choose the Help Index command from the Help menu.

Choosing the Help Index command causes an index list of topics to display. Choose the topic for which you require assistance.

BrightWorks Help system is written in a standard Windows hypertext format. This means that you can jump from one topic to another by simply choosing topic names from a list. Several buttons display across the top of the Help dialog box that allow you to search for topics and also to view a list of the topics you have visited.

For detailed information on using a Windows Help facility, refer to your Windows documentation.

See Also:

[Accessing BrightWorks](#)

[Exiting BrightWorks](#)

[BrightWorks Menu Bar](#)

[BrightWorks Tool Bar](#)

[Using the Keyboard](#)

[BWORKS.INI](#)

[Printer Setup and Administration](#)

[Error Handling](#)

BWORKS.INI File

Upon installation of the BrightWorks software, the BWORKS.INI file is created and placed in the Windows directory of the local workstation. The file can consist of the following sections:

- * **[ShowBrequestWarning]** - This section indicates the status of the Brequest warning that displays upon launching BrightWorks when BrightWorks does not detect the presence of the Btrieve NLM or VAP. This section contains an 'Init=' line that indicates whether or not the warning is disabled (i.e., 'Init=No' when warning is disabled; 'Init=Yes' when warning is enabled).

The warning message can be disabled from within BrightWorks by checking the "Disable message when Brequest isn't running" field when launching the application. The only way to re-enable the warning message after it has been disabled is to edit the BRIGHTWORKS.INI file, and enter 'Init=Yes' in this section.

- * **[Report Preference]** - This section indicates the reporting module that will be accessed upon choosing the BrightWorks Reports tool bar button. Choosing the Reports tool bar button accesses either the Inventory/Distribution reports or the Metering/Security reports, depending on the command that was last selected from the Reports menu. For example, the following section will be included in the BWORKS.INI file if the Inventory and Distribution command was last selected from the Reports menu:

```
[Report Preference]
Module=Inventory
```

- * **[ShowBanner]** - This section can be manually added to the INI file in order to disable the About BrightWorks dialog box that displays upon launching BrightWorks. To disable the About dialog box, enter the following:

```
[ShowBanner]
Init=No
```

To re-enable the warning message, either delete this section or enter 'Init=Yes.'

- * **[DisableExitPrompt]** - This section can be manually added to the INI file in order to disable the prompt that displays upon exiting BrightWorks. This prompt allows you to save the configuration of any open metering windows. (Refer to [Exiting BrightWorks](#).) To disable the exit prompt, add the following section to the BWORKS.INI file:

```
[DisableExitPrompt]
AutoSave=Yes
```

Note that an 'AutoSave=Yes' setting will inhibit the exit prompt and save the configuration of the open *metering windows*. An 'AutoSave=No' setting will also inhibit the exit prompt but will not save the open metering window settings.

- * **[Alternate EXE]** - This section can be manually added to the INI file in order to change the applications that are launched when the BrightWorks Remote or Tickets tool bar buttons are pressed. For example, to run the Magic Solution help desk program when the Tickets tool bar button is chosen, add the following section to the BWORKS.INI file:

```
[Alternate EXE]
ticket=magic.exe
```

To launch another program when the BrightWorks Remote tool bar button is chosen, add the line 'remote=' to this section and indicate the alternative application's executable file name.

Printer Set-up and Administration

Before printing BrightWorks reports, you should review the global print parameters to be sure they reflect the printer settings that you require.

Printer settings include:

- * Printer destination
- * Page orientation (portrait / landscape)
- * Paper size and source
- * Graphics resolution

NOTE:

Please refer to your Windows manual for detailed procedures on modifying the Windows print settings.

Use the following procedure to review and change your print settings.

1. Choose the Printer Setup command from the File menu.

NOTE:

The Orientation section of this dialog box does not appear when you choose the Printer Setup command while the metering reports window is open.

2. Select the printer you want to use for printing BrightWorks reports.

The printer selected from your Windows printer control is selected as the default. To use another printer, select a Specific Printer from the drop-down list associated with this field.

NOTE:

Choosing a specific printer does not permanently change your printer setting.

3. Select the desired orientation and paper parameters.

Choose either the Portrait (long) or Landscape (wide) Orientation setting. Use the drop-down lists to define the Paper Size and Paper Source settings.

4. To make additional changes to the selected printer configuration, choose the Options button.

Additional settings include dithering and intensity control.

5. Choose the OK button in the Print Setup dialog box to save the print settings.

See Also:

[Accessing BrightWorks](#)

[Exiting BrightWorks](#)

[BrightWorks Menu Bar](#)

[BrightWorks Tool Bar](#)

Using the Keyboard

BrightWorks Help Facility

BWORKS.INI

Error Handling

Error Handling

If you encounter an error while using BrightWorks, a message box displays with a description of the error. Choosing the OK button returns you to either the previous screen to select another choice or to the BrightWorks application window if no other choices are available.

Refer to [BrightWorks Error Messages](#) for a list and description all BrightWorks errors.

BrightWorks Metering Help Index

About BrightWorks Metering

[What is BrightWorks Metering Capability?](#)

[BrightWorks Metering Methods](#)

[BrightWorks Metering Features](#)

[BrightWorks Metering Components](#)

Setting Up Applications for Metering

[How BrightWorks Metering Works](#)

[Access to Metering Setup Functions](#)

[What is a Metered Application?](#)

[Registering Applications for Software Metering](#)

[Granting Trustee Rights](#)

[Modifying an Application's Metering Configuration](#)

[Modifying Trustee Rights](#)

[Deleting Trustee Rights](#)

[Deleting Metered Applications](#)

[Attaching to and Detaching from File Servers](#)

[Attaching to a File Server](#)

[Detaching from a File Server](#)

Using the Security Features

[About the Security Features](#)

[Access to Security Functions](#)

[File Integrity Scanning](#)

[Adding Authorized Files](#)

[Reprotecting Authorized Files](#)

[Deleting Authorized Files](#)

[Running Unauthorized Files](#)

[Specifying the File Scan Interval](#)

[Specifying the Security Scan Interval](#)

[Disabling Local Drives](#)

[Restricting Local Execution](#)

[Specifying Security Exceptions](#)

Using the Administration Features

[About the Metering Administration Features](#)

[Access to Metering's Administration Functions](#)

[Viewing Metering Status](#)

[Viewing Application Usage](#)
[Viewing Application Information](#)
[Administering Current Users](#)
[Sending a Message to a Current User](#)
[Viewing Current Users' Information](#)
[Releasing a Current User from a Metered Application](#)
[Launching another BrightWorks Capability](#)
[Administering Queued Users](#)
[Sending a Message to a Queued User](#)
[Editing the Queued User List](#)
[Viewing Queued Users' Information](#)
[Launching another BrightWorks Capability](#)
[Changing the Number of Maximum Concurrent Users](#)
[Changing the Usage Scale](#)
[Changing the Colors Used in the Status Bar](#)
[Hiding or Showing the Status Bar](#)
[Performing Queries](#)
[Configuration Options](#)
[Purge Usage Information](#)
[Purge Security Information](#)

Using the Reporting Functions

[What is the Reporting Module?](#)
[How Does It Work?](#)
[Accessing the Reporting Module](#)
[Exporting Files](#)
[The Report Screen](#)
[Printing From the Reporting Module](#)
[Report Types](#)
[Metering Definitions Report](#)
[Application Summary Report](#)
[Application with User Detail Report](#)
[Application with User Summary Report](#)
[User Summary Report](#)
[User with Application Detail Report](#)
[User with Application Summary Report](#)
[File Integrity Activity Report](#)
[SPA Compliance Report](#)
[Software Purchase Forecast Report](#)
[Upgrade Purchase Forecast Report](#)

Defined Terms

Glossary

What is BrightWorks Metering Capability?

BrightWorks metering features control your LAN software--so you stay legal. And it helps you reduce your software expenditures.

In addition to offering industry--standard metering methods, BrightWorks metering also includes an NLM-based process that both eliminates workstation administration and improves security. Now you can meter your DOS, Windows and OS/2 software without an agent!

Plus BrightWorks provides comprehensive reports with *all* the management information you need.

Because McAfee has been perfecting software metering since 1988, you are getting the most comprehensive package available. Take, for example, the ability to meter suites of applications (e.g., Microsoft Office). With BrightWorks, you can ensure accurate license compliance with groups of files that are governed by a single license agreement.

The Purpose of Software Metering

Software metering puts control of all your network applications at your fingertips. With software metering you can:

- * Reduce software expenditures
- * Reduce training and administrative expenses
- * Enforce software license compliance
- * Track software usage

See Also:

[BrightWorks Metering Methods](#)

[BrightWorks Metering Features](#)

[BrightWorks Metering Components](#)

BrightWorks Metering Methods

By providing these options, you can choose the method best suited to your network needs. Multiple methods can be loaded on the same network simultaneously.

File Server Agent Method for DOS

By using a file server agent to meter your DOS, Windows, and OS/2 based network software, **no workstation software is required**. Because software usage is tracked at the system level, you gain:

- * Compatibility with NetWare 3.x
- * Tracking of all DOS software at the file server via the NLM
- * Metering without workstation software

This method involves the Proxy NLM directly. To use this option you must be running NetWare 3.x. Before a user fully loads an application, the Proxy NLM intercepts the request and checks to verify that the application is available. If the NLM grants permission, the user is permitted to use the application.

Using the Proxy NLM does not consume any additional memory on the workstation and is the least intrusive method of providing software metering.

Workstation Security Agent

In addition to multiple methods of metering, the metering capability also provides an optional workstation security agent. This method:

- * Is compatible with NetWare 2.x and 3.x
- * Tracks all DOS and Windows software usage via the workstation TSR and file server NLM or VAP
- * Can disable local drives
- * Can prevent execution of local files

The metering's workstation security agent is Swatcher and is a DOS TSR that requires 5K. Swatcher is included in the package but should be loaded only if the above capabilities are required. You can also load Swatcher for metering purposes if you choose not to use the file server agent method for metering DOS applications.

See Also:

[BrightWorks Metering Features](#)

[BrightWorks Metering Components](#)

BrightWorks Metering Features

BrightWorks offers a comprehensive set of features and a wide range of capabilities:

- * Metering for individual DOS, Windows and OS/2 programs (or suites of programs) via fourth generation agent based metering
- * Optional workstation security agent
- * Real time trustee rights tied to applications to protect sensitive files
- * Real time graphical display of application usage
- * Valuable application usage summaries and reports
- * File protection against possible virus infection
- * Automatic notification for users when applications become available
- * Windows compatibility and native Windows console
- * Security for your network with optional password protection and VAP/NLM technology

These features allow you to:

- * Assure compliance with software license agreements
- * Prevent users from using local drives (with optional security agent)
- * Determine software license requirements based on actual usage
- * Prevent costly downtime caused by virus infection
- * Increase productivity by keeping your users informed about application availability
- * Monitor software usage on several servers simultaneously
- * Control your network by keeping it secure

See Also:

[BrightWorks Metering Methods](#)

[BrightWorks Metering Components](#)

BrightWorks Metering Components

The License Server NLM/VAP

SITEMETR.NLM/SITEMETR.VAP is the module responsible for granting or denying permission to execute applications. It resides on the file server.

Using this module with one of the metering methods, you can:

- * Control all access to metered applications
- * Track software usage on your network

The File Server Agent NLM for DOS

SMRPROXY.NLM is the file server agent method of software metering and file protection. As users request permission to use applications, the File Server Agent NLM communicates with the License Server NLM/VAP.

Using SMRPROXY.NLM, you can:

- * Determine if the applications are available to run
- * Meter software without any workstation software on DOS workstations for NetWare 3.x file servers

The Workstation Security Agent

SWATCHER.COM is the workstation security agent method for software metering and file protection. It acts as a messenger between the application and the file server.

Using SWATCHER.COM, you can:

- * Automatically log out users who have tried to circumvent BrightWorks metering capability
- * Restrict local execution
- * Disable local drives

The Usage Monitor

USAGE.EXE is a BrightWorks metering utility that you can make available to your network users. Using this Windows utility, the users can determine who is using a metered application in real time.

With USAGE.EXE, the users can:

- * View a list of both current and queued users of an application
- * Send current or queued users a NetWare Send Message

The View Users Utility

SLOOK.EXE is a DOS program that you can make available to your users to enable them to determine who is using a metered application in real time.

Using SLOOK.EXE the users can:

- * View a list of current users of an application
- * Send current users a NetWare Send Message

Using DOS Slook, a user with SUPERVISOR rights can:

- * View a list of queued users for an application
- * View a list of current users of an application

The Report Utility

SREPORT.EXE generates a variety of reports with useful information about software usage on your network.

Using this Windows application, you can:

- * Determine the number of concurrent, queued and peak users for an application
- * Identify the number of licenses needed to accommodate current need, as well as a 10% increase and a 20% increase
- * Access information about the activity on virus secured files and applications

See Also:

[BrightWorks Metering Methods](#)

[BrightWorks Metering Features](#)

How BrightWorks Metering Works

Think of the metering capability as a public library. You go to the library to check out a copy of a book (i.e., software). The library only has two copies of this book, and they are both checked out.

The library places your name on a waiting list (i.e., queue); you then have the option to check out the book once it becomes available.

Once the copy is available, the first person on the waiting list is notified. The library holds this book exclusively for this person for a predetermined amount of time (i.e., queue-back time).

If after that time the person has not checked out the book, the next person on the list is notified of the book's availability. If no one else is waiting for the book, it is returned to the shelf for anyone's use.

BrightWorks metering works in much the same way. User requests to run applications go through the license server. This program checks for the application's availability; if it is available, the user is allowed to run the program.

Once the maximum number of users is reached (as set by the network administrator), any further attempts to access the software are prevented. The metering capability then places all other potential users on a waiting list (queue), unless otherwise specified by the network administrator.

Users that are placed in the queue are notified when a free copy of the application is available. The application is held for the exclusive use of the notified user for a specified number of minutes. If the user does not access the application within this time period, it is offered to the next user in the queue.

See Also:

[Access to Metering Setup Functions](#)

[What is a Metered Application?](#)

[Registering Applications for Software Metering](#)

Access to Metering Setup Functions

The functions needed to setup applications for metering are accessed in two ways:

- * by choosing the Metering button from the tool bar, or
- * by choosing the Metering command from the Administration menu.

See Also:

[What is a Metered Application?](#)

[Registering Applications for Software Metering](#)

What is a Metered Application?

A metered application is a software application(s) that has been registered with BrightWorks for software metering. When registering a file(s), you need to gather the following information for each application:

- * File(s) to Meter
- * Metered Application Name
- * Full Name
- * Maximum Number of Concurrent Users.
- * Password
- * Queue Back Time
- * Directories in which Trustee Rights are Granted During Execution

See Also:

Registering Applications for Software Metering

Registering Applications for Software Metering

To control the number of simultaneous users of an application, you must register this application with BrightWorks. When registering a product, you need the information described in [What is a Metered Application?](#). Once a file is registered, BrightWorks ensures that only the specified number of concurrent users is using that application simultaneously.

Use the following procedure to register an application for metering.

1. **Choose the Metering command from the Administration menu. From the sub-menu that displays, choose the Define Metered Applications command.**

This dialog box displays all files currently registered to be metered. (If you have not registered any files for metering, the list will be blank.)

This dialog box offers the following options:

- * **Add** allows you to register an application for metering.
- * **Modify** allows you to change the information already entered for a metered application.
- * **Delete** allows you to remove an application from metering.
- * **Attach to/Detach from File Servers** allows you to attach to or detach from different file servers while in BrightWorks. See [Attaching to and Detaching from File Servers](#).

2. **Choose the Add button to register a metered application.**

The Add Metered Application dialog box displays.

3. **Choose the Add button.**

The Browse for Files to Meter dialog box displays.

4. **Select the appropriate filename (and directory, which is optional) from the list and choose the OK button to insert that filename in the File(s) to Meter text box. You can select multiple files for suite metering.**

NOTE:

To include the file's entire path, select the Include Path option. The entire path and file name will be inserted in the File(s) to Meter text box.

You are returned to the Add Metered Application dialog box.

5. **Select the Metered Application Name text box and enter the metered application name.**

For example: WordPerfect. You cannot enter spaces in the metered application name.

6. **Select the Full Name text box and type the full name of the application.**

You can also enter descriptive information about the application you are metering. For example: WordPerfect Version 5.1 for DOS.

7. **Select the password text box and enter a password for this Metered Application (optional).**

Entering a password is optional. If you assign a password to this application or suite, you must provide this password before reconfiguring this metered application (or suite of applications).

The password does not appear in the Password text box.

NOTE:

Assigning a password does not require users to enter that password before running the application. Instead it protects the metered application information you entered from any unauthorized changes.

8. **Select the Maximum Number of Concurrent Users text box and enter the number of licenses you have purchased for this application.**

One is the default.

9. **Select the Queue Back Time text box and enter an appropriate length of time.**

Entering a Queue Back Time is also optional. If no time is specified, the queue-back feature is disabled.

The default time is five minutes.

10. **If you want to grant trustee rights, see Granting Trustee Rights. Otherwise, choose the OK button to exit this dialog box.**

If you choose the OK button, you have finished registering this application for software metering.

NOTE:

The information that you entered for the metered application can be changed if necessary.

See Also:

Modifying an Application's Metering Configuration

Deleting Metered Applications

Granting Trustee Rights

This option allows you to grant temporary rights (trustee assignments) to users while an application is running.

This option is only in effect for users running NetWare 3.x. As the network administrator, you always have access to this feature, but only users running NetWare 3.x will be granted trustee assignments.

For example, you can configure metering so that a user has rights to the ACCOUNTS directory only while he or she is running the Accounts Receivable package. This prevents users from copying/viewing/deleting financial data files from outside the application.

Please note the following when using this option:

- * **Rights for Named File Only** - Secured Directory rights are available only for the file whose execution is being tracked. For example, if the application is WordPerfect (WP.EXE), you have Secured Directory rights while running this application. However, if you issue the DOS Shell command and exit into DOS, all rights are revoked until you EXIT back to WordPerfect.
- * **Use Group Inheritance for Continuous Rights Inheritance** if you need a continuous "base" of rights in a directory affected by a Secured Directory definition, you **MUST** define those rights via SYSCON's Group Inheritance.
- * **Supervisor Equivalent** if a user is a Supervisor or has Supervisor Equivalence, the Directory Security Mask feature does not affect that user.
- * **Spawned Applications** if you define Directory Security for an application and that application calls another application (e.g., a menu system), the called application will not automatically receive Directory Security rights. If you wish the called application to have Directory Security rights, you must meter and grant rights to each called application.

Use the following procedure to grant trustee rights.

1. From the Add Metered Applications dialog box, choose the Directories button.

The Select a Drive/Directory dialog box displays. This dialog box allows you to traverse all directories.

2. Double click on the desired drive.

A list of directories on that drive displays.

3. From the list, double click on the desired directory.

The new drive/directory displays as your Current Directory. This is where you will grant trustee rights.

4. Choose the OK button to grant rights.

The Select Rights Mask dialog box displays.

5. To assign rights, select the desired right from the Rights Available to Grant list and then choose the Include button.

The right displays in the Rights Granted list.

To include all rights, choose the Include All button.

If you wish to remove a right you have already assigned, select the right from the Rights Granted list and then choose the Remove button. To remove all rights, choose the Remove All button.

6. Choose the OK button to exit the Select Rights Mask dialog box.

In the Add Metered Applications dialog box, the rights you assigned and the directory in which they were assigned are shown in the text box.

7. Repeat steps 1 through 6 for all directories where you wish to grant rights.

8. Choose the OK button to exit the Add Metered Applications dialog box.

Notice that the directories in which you granted rights are shown in the Directories in Which Trustee Rights are Granted list.

This completes registering this metered application.

See Also:

[Modifying an Application's Metering Configuration](#)

[Modifying Trustee Rights](#)

[Deleting Trustee Rights](#)

Modifying an Application's Metering Configuration

You can modify any of the metered application information you provided when registering the software for metering.

Use the following procedure to change metered application information.

1. **Choose the Metering command from the Administration menu. From the sub-menu that displays, choose the Define Metered Application command.**

The Define Metered Application dialog box displays.

2. **Select the application you wish to modify.**

3. **Choose the Modify button.**

If the application does not have a password associated with it, the Modify Metered Application dialog box displays with all the information you provided when registering this application for metering.

If the application (or suite) has a password assigned to it, a dialog box displays prompting you to enter that password. Enter either the Metered Application or the Supervisor Password and then choose the OK button.

4. **Select the file(s) you wish to modify from the File(s) to meter list box.**

This list box allows you to meter a suite of applications that share a license agreement. For example, if you have one license agreement that covers three applications, only one person should be able to access any of these three applications at one time (as opposed to three different users using the three different applications at one time). By selecting all the files belonging to a suite of applications, you can modify the name, password, maximum number of concurrent users, queue back time and trustee rights for all files under a single license.

5. **Make all necessary changes to the information shown in the window by selecting the text box and typing the new data.**

NOTE:

The changes you make do not take effect until you choose the OK button. The file server console reflects the changes made by displaying "Refreshing Filename Table for Metered Applications."

6. **Choose the OK button to exit the Modify Metered Application dialog box and save your changes.**

You are returned to the Define Metered Application dialog box where you can either modify other applications or exit this dialog box by choosing the Close button.

See Also:

[Modifying Trustee Rights](#)

[Deleting Trustee Rights](#)

[Deleting Metered Applications](#)

Modifying Trustee Rights

In addition to changing the metered application information described in [Modifying an Application's Metering Configuration](#), you can also reassign or delete the trustee rights you had previously granted.

Use the following procedure to modify trustee rights.

- 1. From the Modify Metered Application dialog box, select the directory whose rights you wish to modify.**
- 2. Choose the Modify Mask button.**
The Select Rights Mask dialog box displays.
- 3. Include additional rights by selecting the desired right from the Rights Available to Grant list and choosing the Include button.**
You can also include all available rights by choosing the Include All button.
- 4. Remove existing rights by selecting the desired right from the Rights Granted list and choosing the Remove button.**
You can remove all the granted rights by choosing the Remove All button.
- 5. Choose the OK button to exit the Select Rights Mask dialog box.**

The Modify Metered Application dialog box displays with the changes you made to the granted rights.

See Also:

[Modifying an Application's Metering Configuration](#)

[Deleting Trustee Rights](#)

[Deleting Metered Applications](#)

Deleting Trustee Rights

You can also delete all the rights you assigned from the Modify Metered Application dialog box.

NOTE:

Deleting rights does not delete any of those rights which users may have permanently assigned to them through Novell's system.

Use the following procedure to delete trustee rights.

- 1. Select the desired directory from the Directory In Which Trustee Rights Are Granted list.**
- 2. Choose the Delete button.**

The rights are removed from this directory.

See Also:

[Modifying Trustee Rights](#)

[Deleting Metered Applications](#)

Deleting Metered Applications

You can remove an application from metering. This procedure does not remove the application from your network.

Use the following procedure to delete applications from BrightWorks metering capability.

1. **Choose the Metering command from the Administration menu. From the sub-menu that displays, choose the Define Metered Applications command.**

The Define Metered Application dialog box displays.

2. **Select the application to delete.**
3. **Choose the Delete button.**
If the application has a password associated with it, you are prompted to enter that password.
4. **From the prompt that displays, choose the Yes button to delete the metered application.**
5. **Choose the Close button to exit this window.**

Attaching to and Detaching from File Servers

BrightWorks metering offers you flexibility in managing your network applications. Many of the Metering dialog boxes allow you to attach to and detach from file servers in a multiserver environment on your LAN.

Using the Attach and Detach buttons, you can change file servers quickly and easily. In doing so, you can control metering on any file server you wish. This feature allows you to log in and out of file servers while using BrightWorks.

This tool will be particularly useful when you register applications for software metering.

See [Configuration Options](#) for instructions about both the home directory and logging historical data.

See Also:

[Attaching to a File Server](#)

[Detaching from a File Server](#)

Attaching to a File Server

The steps outlined below describe attaching to and detaching from file servers from the Configuration Options dialog box. Many of BrightWorks metering dialog boxes have Attach and Detach buttons.

NOTE:

If you choose these buttons from a dialog box, you can skip step 1 of these two procedures.

Use the following procedure to attach to a file server.

1. **Choose the Metering command from the Administration menu. From the sub-menu that displays, choose the Configuration Options command.**

The Configuration Options dialog box displays. Your current server automatically displays; you can choose another server from this list box.

2. **Choose the Attach button.**

The Attach to Server dialog box displays,.

3. **From the list box, select the file server to which you wish to attach.**
4. **Enter your user name and password to attach to that file server.**
5. **Choose the OK button to exit this dialog box.**
6. **Choose the OK button to exit the Configuration Options dialog box.**

See Also:

[Detaching from a File Server](#)

Detaching from a File Server

Use the following procedure to detach from the current file server.

1. **Choose the Metering command from the Administration menu. From the sub-menu that displays, choose the Configuration Options command.**

The Configuration dialog box displays. Your current server automatically displays; you can select a different file server from this list box.

2. **Choose the Detach button.**

The Detach from Server dialog box displays.

3. **Select the desired file server from the drop-down list box.**
4. **Choose the OK button to detach from this file server.**

NOTES:

a - You cannot detach from the file server from which the BrightWorks was launched.

b - You cannot detach from a file server if it is the only file server to which you are attached or if it is your current file server.

5. **Choose the OK button to exit the Configuration Options dialog box.**

See Also:

[Attaching to a File Server](#)

About the Security Features

BrightWorks metering offers a number of features that secure and protect your network software by preventing viruses and unauthorized access to local drives.

The file integrity scanning feature guards your LAN against potential infection from viruses. A virus is an additional process that attaches itself to or maliciously alters an executable file. A virus can cause many problems on your network, such as:

- * Rename or destroy data
- * Create a program that can cause the workstation to hang or possibly crash
- * Cause a program to run in a continuous loop
- * Make a program consume more memory than is necessary

With file integrity scanning you reduce the risk of virus infection.

BrightWorks provides other security features related to local drive use. You can control which users have access to their local drives to prevent unauthorized software and program installation. With BrightWorks metering, you can define rights to local execution for all users or for specific users.

See Also:

[Access to Security Functions](#)

[File Integrity Scanning](#)

[Adding Authorized Files](#)

[Reprotecting Authorized Files](#)

[Deleting Authorized Files](#)

[Running Unauthorized Files](#)

[Specifying the File Scan Interval](#)

[Specifying the Security Scan Interval](#)

[Disabling Local Drives](#)

[Restricting Local Execution](#)

[Specifying Security Exceptions](#)

Access to Security Functions

The security functions are accessed in two ways:

- * by choosing the Security button from the tool bar, or
- * by choosing the Security command from the Administration menu.

See Also:

[File Integrity Scanning](#)

[Adding Authorized Files](#)

[Reprotecting Authorized Files](#)

[Deleting Authorized Files](#)

[Running Unauthorized Files](#)

[Specifying the File Scan Interval](#)

[Specifying the Security Scan Interval](#)

[Disabling Local Drives](#)

[Restricting Local Execution](#)

[Specifying Security Exceptions](#)

File Integrity Scanning

To prevent virus infection, the BrightWorks checks files for changes before allowing them to execute. The first step is registering (or authorizing) your files for scanning. Once you register your software, only files that have a fingerprint matching the fingerprint registered are allowed to run.

Every time the file is run (or at specified intervals), the fingerprint value is recalculated and compared to the value that had been originally registered. If the two do NOT match, the file is not allowed to run.

With this method of file protection it is not necessary to recognize a particular virus strain. Any byte change is detected and treated as a potential virus.

See Also:

[Adding Authorized Files](#)

[Running Unauthorized Files](#)

[Specifying the File Scan Interval](#)

[Specifying the Security Scan Interval](#)

[Disabling Local Drives](#)

[Restricting Local Execution](#)

[Specifying Security Exceptions](#)

Adding Authorized Files

To register a file for virus protection, you need to add it to the list of [authorized files](#).

Use the following procedure to add a file to the authorized files list.

1. **Choose the Security command from the Administration menu. From the sub-menu that displays, choose the Define Authorized Files command.**

The Define Authorized Files dialog box displays.

From this dialog box you can:

- * **Add** files to the Currently Authorized Files list.
- * **Reprotect** files that are already authorized.
- * **Delete** files that have been authorized.

2. **Choose the Add button.**

The Browse for Files to Authorize dialog box displays.

3. **Select the desired drive and directory.**

4. **Select a file from the File Name list.**

You can select all the files in this directory by pointing to the first file in the File Name list, holding down the left mouse button and dragging the cursor down. This highlights all the files that will be authorized.

5. **Choose the OK button.**

Any files you just chose now appear in the Currently Authorized Files list.

See Also:

[File Integrity Scanning](#)

[Reprotecting Authorized Files](#)

[Deleting Authorized Files](#)

[Running Unauthorized Files](#)

[Specifying the File Scan Interval](#)

Reprotecting Authorized Files

You can reprotect a previously authorized file. This should be done for applications that have been upgraded. Reprotecting a file recalculates the file's checksum value.

NOTE:

If you are using Swatcher to meter and file protect your network, make sure Swatcher is not loaded when reprotecting a file.

Use the following procedure to reprotect a file.

1. **Choose the Security command from the Administration menu. From the sub-menu that displays, choose the Define Authorized Files command.**

The Define Authorized Files dialog box displays.

2. **Choose the file you wish to reprotect.**
3. **Choose the Reprotect button.**

A message displays at the bottom of the window indicating that the file is being updated for protection.

NOTE:

You can also reprotect a file by double clicking on the appropriate filename in the Currently Authorized Files list.

See Also:

[File Integrity Scanning](#)

[Adding Authorized Files](#)

[Deleting Authorized Files](#)

[Running Unauthorized Files](#)

Deleting Authorized Files

You can remove authorization from a file. This does not remove the file from the network; it merely removes the BrightWorks security protection features for the file. When you do this, the file will be allowed to execute regardless of any changes made to the file. (It will not, however, be allowed to run at all when the unauthorized files option is enabled.)

Use the following procedure to remove an authorized file from registration for file integrity scanning.

1. **Choose the Security command from the Administration menu. From the sub-menu that displays, choose the Define Authorized Files command.**

The Define Authorized Files dialog box displays.

2. **Select the file to be deleted.**
3. **Choose the Delete button.**

A prompt displays, asking you to verify your choice to delete the file protection from this file.

4. **Choose the Yes button if you wish to remove the file protection.**

See Also:

[File Integrity Scanning](#)

[Adding Authorized Files](#)

[Reprotecting Authorized Files](#)

[Running Unauthorized Files](#)

Running Unauthorized Files

Use the following procedure to use the run unauthorized files option.

NOTE:

Make sure BWORKS.EXE is always an authorized file. If BWORKS.EXE is not an authorized file and you choose to use the unauthorized files option, you will NOT be able to run BrightWorks. Also make sure LOGIN.EXE is always an authorized file. If LOGIN.EXE is not an authorized file and you choose to use the unauthorized files option, users will be unable to log in to the network.

1. **Choose the Security command from the Administration menu. From the sub-menu that displays, choose the Specify Policy command.**

The Specify Policy dialog box displays.

This dialog box allows you specify on which file servers you allow unauthorized files to run.

The Current Server list box automatically displays your current server. You can attach to or detach from other file servers using the Attach and Detach buttons.

NOTE:

The default is to have the option enabled. Step 2 disables this option.

2. **If you do not wish to allow unauthorized files to be executed, select the "Allow unauthorized files to be executed" option.**

The "x" disappears from the box, indicating that you do not allow files that are not authorized to run on the network.

3. **Choose the OK button to save your change and exit the dialog box.**

See Also:

[File Integrity Scanning](#)

[Specifying the File Scan Interval](#)

[Specifying the Security Scan Interval](#)

[Disabling Local Drives](#)

[Restricting Local Execution](#)

[Specifying Security Exceptions](#)

Specifying the File Scan Interval

Use the following procedure to specify the [file scan interval](#).

1. **Choose the Security command from the Administration menu. From the sub-menu that displays, choose the Specify File Scan Interval command.**

The Specify File Scan Interval dialog box displays.

From this dialog box you can:

- * Set the scan interval
- * Attach to/Detach from a file server

2. **Use one of the following methods to set the interval and specify how frequently the metering capability checks the executable:**

- * Click on the slide bar arrows to increment/decrement the value in one minute intervals,
- * Slide the slide bar to the appropriate value, or
- * Click on either side of the slide bar to increment/decrement the value by 10 minute intervals.

3. **Once you have selected the appropriate time, choose the OK button.**

See Also:

[File Integrity Scanning](#)

[Specifying the Security Scan Interval](#)

[Disabling Local Drives](#)

[Restricting Local Execution](#)

[Specifying Security Exceptions](#)

Specifying the Security Scan Interval

Use the following procedure to specify the security scan interval.

NOTE:

Trying to load Swatcher after receiving the 30 second warning will not prevent the user from being logged out.

*Setting the value to 0 minutes informs metering not to check if Swatcher is loaded. **This is mandatory if you are NOT using the Swatcher TSR as your choice to meter and file protect your network.***

1. **Choose the Security command from the Administration menu. From the sub-menu that displays, choose the Specify Security Scan Interval command.**

The Specify Security Scan Interval dialog box displays.

From this dialog box you can:

- * Set the scan interval
- * Attach to/Detach from file servers (See Attaching to and Detaching from File Servers.)

2. **Use one of the following methods to set the interval and specify how often BrightWorks metering capability checks users for the Swatcher TSR:**

- * Click the slide bar arrows to increment/decrement the value in one minute intervals,
- * Slide the slide bar to the appropriate value, or
- * Click on either side of the slide bar to increment/decrement the value in 10 minute intervals.

NOTE:

*Setting the value to 0 minutes informs metering not to check if Swatcher is loaded. **This is mandatory if you are NOT using the Swatcher TSR as your choice to meter and file protect your network.***

3. **Once you have selected the appropriate time, choose the OK button.**

See Also:

[File Integrity Scanning](#)

[Disabling Local Drives](#)

[Restricting Local Execution](#)

[Specifying Security Exceptions](#)

Disabling Local Drives

To use the disable local drives option you must specify a Novell Group in which users will not have access to their local disk drives. For example, you may want to create a group named NODRIVE.

This NODRIVE group must be unique; the group you select for Disable Local Drives cannot also be used for Restrict Local Execution.

NOTE:

For information on creating these groups, refer to your Novell User Manuals.

Use the following procedure to disable local drives.

- 1. Choose the Security command from the Administration menu. From the sub-menu that displays, choose the Disable Local Drives command.**

The Disable Local Drives dialog box displays.

From this dialog box you can:

- * Include groups
- * Remove groups
- * Attach to/Detach from file servers (See Attaching to and Detaching from File Servers)

- 2. If you wish to add a group to the Groups With Disabled Local Drives list, select the desired group from the Available Groups list and choose the Include button.**

The group is then moved from the Available Groups list to the Groups With Disabled Local Drives list.

You can include all the available groups by choosing the Include All button.

The groups you included now do not have access to their local drives.

- 3. If you wish to remove a group from the Groups With Disabled Local Drives list, select the desired group from this list and choose the Remove button.**

The group is then moved from the Groups With Disabled Local Drives list to the Available Groups list.

You can remove all groups from the disabled drives list by choosing the Remove All button.

The groups you removed now have access to their local drives.

- 4. When you have completed moving groups, choose the OK button to save your changes and exit.**

See Also:

File Integrity Scanning

Restricting Local Execution

Specifying Security Exceptions

Restricting Local Execution

To use the restrict local execution option, you must specify a Novell Group in which users will not be able to execute files from their local drives. For example, you may want to name this group NOEXEC.

NOTE:

For information on creating these groups, refer to your Novell User Manuals.

Use the following procedure to restrict local execution.

NOTE:

If you decide to use this option please refer to the section entitled "DSW" in Chapter 11 of your BrightWorks manual.

- 1. Choose the Security command from the Administration menu. From the sub-menu that displays, choose the Restrict Local Execution command.**
The Restrict Local Execution dialog box displays.
From this dialog box you can:
 - * Include groups
 - * Remove groups
 - * Attach to/Detach from file servers (See Attaching to and Detaching from File Servers)
- 2. To add a group to the Groups With Restricted Execution list, select the desired group from the Available Groups list and choose the Include button.**
The group is then moved from the Available Groups list to the Groups With Restricted Execution list.
You can include all the available groups by choosing the Include All button.
The groups you included now cannot execute applications from their local drives.
- 3. To remove a group from the Groups With Restricted Execution list, select the desired group from this list and choose the Remove button.**
The group is then moved from the Groups With Restricted Execution list to the Available Groups list.
You can remove all groups from the restricted list by choosing the Remove All button.
The groups you removed now can execute all applications from their local drives.
- 4. When you have completed moving groups, choose the OK button to save your changes and exit.**

See Also:

File Integrity Scanning

Disabling Local Drives

Specifying Security Exceptions

Specifying Security Exceptions

Use the following procedure to define security exceptions.

NOTE:

If you decide to use this option please refer to the section entitled "DSW" in Chapter 11 of your Fusion manual.

1. **Choose the Security command from the Administration menu. From the sub-menu that displays, choose the Specify Security Exceptions command.**

The Specify Security Exceptions dialog box displays.

From this dialog box you can:

- * Include users
- * Remove users
- * Attach to/Detach from file servers (See Attaching to and Detaching from File Servers)

2. **To add a user to the Security Exceptions list, select the desired user from the Secure Users list and choose the Include button.**

The user is then moved from the Secure Users list to the Security Exceptions list.

You can include all the Secure Users by choosing the Include All button.

The users you included now are not required to load Swatcher when using the network.

3. **To remove a user from the Security Exceptions list, select the desired user from this list and choose the Remove button.**

The user is then moved from the Security Exceptions list to the Secure Users list.

You can remove all users from the security exceptions list by choosing the Remove All button.

The users you removed now are required to load Swatcher.

4. **When you have completed moving users, choose the OK button to save your changes and exit.**

See Also:

Disabling Local Drives

Restricting Local Execution

Specifying Security Exceptions

About the Metering Administration Features

BrightWorks metering gives you the tools you need to administer your metered applications and monitor your network software usage.

The metering capability is extremely flexible, giving you maximum control in administering your network. You can administer metering on different file servers and even launch another BrightWorks capability.

Most of the metering administration functions can be accessed from the application usage graph, giving you maximum flexibility and control over your network applications. You can use all of the following features to monitor your metered applications:

- * **View the current metering settings.** See [Viewing Metering Settings](#).
- * **View a list of all the metered applications for a specified file server in addition to the number of current, peak and queued users and the number of licenses.** See [Viewing Application Usage](#).
- * **View information about and send messages to both current and queued users, as well as release them from metering.** See [Viewing Current Users' Information](#), [Viewing Queued Users' Information](#), [Sending a Message to a Current User](#), and [Sending a Message to a Queued User](#).
- * **Perform a query to update the application usage information throughout the network.** See [Performing Queries](#).
- * **Set the high value for the usage scale.** See [Changing the Usage Scale](#).
- * **Conceal the status bar located at the bottom of the Application Usage dialog box.** See [Hiding or Showing the Status Bar](#).
- * **Purge stored data.** See [Purge Usage Information](#) and [Purge Security Information](#).

Access to Metering's Administration Functions

The metering administration functions are accessed in several ways:

- * by choosing the Monitor button from the tool bar,
- * by choosing the Monitoring command from the Administration menu,
- * by choosing the Security command from the Administration menu, or
- * by choosing the Hide/Show command from the File menu.

See Also:

[Viewing Metering Status](#)

[Viewing Application Usage](#)

[Viewing Application Information](#)

[Administering Current Users](#)

[Administering Queued Users](#)

[Changing the Number of Maximum Concurrent Users](#)

[Changing the Usage Scale](#)

[Changing the Colors Used in the Status Bar](#)

[Hiding or Showing the Status Bar](#)

[Performing Queries](#)

[Configuration Options](#)

[Purge Usage Information](#)

[Purge Security Information](#)

Viewing Metering Settings

BrightWorks metering lets you view all the metering settings for a file server.

With this feature, you can access the following important information:

- * [Current Server](#)
- * [SiteMeter NLM](#)
- * [SiteMeter Proxy NLM](#)
- * [Unauthorized Files](#)
- * [File Integrity Status](#)
- * [Current Home Directory](#)
- * [Swatcher Users](#)
- * [Status of Swatcher](#)

Use the following procedure to view the current metering settings.

- 1. Choose the Metering command from the Administration menu. From the sub-menu that displays, choose the View Metering Status command.**

The View Metering Status dialog box displays.

If you are not attached to the desired file server, use the Attach button.

The current server displays automatically. You can view the settings for other file servers by choosing the desired file server from the list box.

- 2. Choose the Close button to exit this dialog box.**

See Also:

[Viewing Metering Status](#)

[Viewing Application Usage](#)

[Viewing Application Information](#)

Viewing Application Usage

The metering capability gives you the ability to view application usage on a specified file server. Using this feature, you can determine which applications are being used on the network, which applications have queued users, what the peak usage for an application is, and more.

Use the following procedure to access the application usage graph.

1. Choose the Monitor button from the tool bar.

The View Application Usage dialog box displays.

2. Select the desired file server.

If you are not currently attached to the desired file server, choose the Attach button and then supply your user name and password for that file server.

3. Choose the OK button.

The Application Usage window for the file server you have selected displays.

From this window you can view the following information in graph form:

Number of:

- * **Current users** of all metered applications
- * **Queued users** of all metered applications
- * **Peak users** of all metered applications (the total number of current users plus the total number of queued users)
- * **Maximum Number of Concurrent Users** of all metered applications (the total number of licenses purchased for this application)

See Also:

[Viewing Application Information](#)

Viewing Application Information

You can view the following application information from this window:

- * Number of Current Users
- * Number of Queued Users
- * Number of Peak Users
- * Number of Licenses

This information can be viewed either on the graph itself or with pop-up boxes.

Use the following procedure to view this information with the pop-up boxes.

- 1. Select the desired application from the list along the left hand side of the window.**
When the cursor is in this area, it changes to a magnifying glass.
- 2. Hold down the left mouse button to display an information box that contains the above information.**

See Also:

[Administering Current Users](#)

[Administering Queued Users](#)

Administering Current Users

From the application usage graph, you can view the names of the individual users using an application, as reflected by the graph bars. This is particularly helpful if other users are trying to access an application. Once you find out which users are using the application currently, you can determine when a copy will be available by asking those users on the list how much longer they will be working in the desired application.

Once you access this list, you have additional administrative options.

Use the following procedure to view the names of the users who are currently using an application.

- 1. In the application usage graph, position the cursor to the area to the right of the application name within the graphic display.**

A small menu box should be attached to the bottom right of the arrow. If there is no menu box, the cursor is not pointing to a place where information is available. Information is available where the number of users (queued, peak and current) is displayed with graph bars.

- 2. Click here to display a popup menu.**

When the pop-up menu displays, the application in question is outlined in the list along the left hand side of the window.

From this menu, you can:

- * **View Current Users** the list of users currently using this application
- * **View Queued Users** the list of users who are currently waiting to use this application
- * **Edit Queued User List** list of queued users who you can add or delete with this option

NOTE:

If the Queue Back Time is set to zero (0), the View Queued Users and Edit Queued User List options are disabled.

- 3. Choose the View Current Users command.**

The Current Users window displays.

This window lists all the users who are currently using this application.

From this window you can:

- * Send a Message to a Current User
- * View Current Users' Information
- * Release a Current User from a Metered Application
- * Launch another BrightWorks Capability

Sending a Message to a Current User

You can send messages to current users from the Current Users window. This is useful if you want to know when a user will be finished using an application.

Use the following procedure to send a NetWare Send message to a user who is currently using an application.

- 1. From the Current Users window, choose the user to whom you wish to send the message.**
A popup menu displays.
- 2. Choose the Send Message command.**
The Send Message To User dialog box displays.
- 3. Enter the message in the text box provided.**
- 4. Choose the OK button to send the message.**

See Also:

[Viewing Current Users' Information](#)

[Releasing a Current User from a Metered Application](#)

[Launching another BrightWorks Capability](#)

Viewing Current Users' Information

From the Current Users window, you can find out important information about users using applications on your network.

The following information can be displayed about a selected current user:

- * Login Name
- * Full Name
- * Server
- * Logical Station
- * Network
- * Station Address
- * Time into Network
- * Application
- * Time into Application

Use the following procedure to view this information.

- 1. From the Current Users window, select the desired user.**
A popup menu displays.
- 2. Choose the User Information command.**
The Current User Information dialog box displays.
- 3. Choose the Close button to exit this dialog box.**

See Also:

[Sending a Message to a Current User](#)

[Releasing a Current User from a Metered Application](#)

[Launching another BrightWorks Capability](#)

Releasing a Current User from a Metered Application

The Release command lets you release a current user from being metered in an application. This is helpful in the following scenario:

John is using the last available copy of WordPerfect. He goes into a meeting with his workstation still running the application. Meanwhile, Karen needs to use a copy of WordPerfect. The network administrator can release John's copy of WordPerfect from metering, freeing up a licensed copy for Karen.

When John returns from his meeting, he can still use the copy of WordPerfect running on his workstation (using the release feature does not exit that user from the application). Once he exits WordPerfect, however, he cannot get back in until another copy of the software frees up on the network.

NOTE:

If you use the Release feature, you may be in violation of your software vendor's license agreement.

Use the following procedure to release a current user from metering.

- 1. From the Current Users window, choose the user whom you wish to release.**
A pop-up menu displays.
- 2. Choose the Release command.**
- 3. If the metered application has a password associated with it, you are prompted to enter the password.**
Enter the appropriate password and choose the OK button.
- 4. At the prompt, choose the Yes button to verify your choice to release this user from metering.**
Choose the No button to exit the message box without releasing the user.
If you chose the Yes button, the selected user is released from metering. The copy of the application he or she is using is released for use by another user.

See Also:

[Sending a Message to a Current User](#)

[Viewing Current Users' Information](#)

[Launching another BrightWorks Capability](#)

Launching another BrightWorks Capability

From the menu, you can also launch another BrightWorks capability (NETremote+ or LAN Support Center) in a context-sensitive fashion. When launching the capability, it automatically configures it to the current user you selected. For example, if you selected Jane and then launched the remote capability, you would have control of Jane's PC.

NOTE:

Users must have the executables for these programs in their search path.

Use the following procedure to launch another BrightWorks capability.

1. **From the Current Users window, select the desired user.**
2. **Choose the desired capability (either remote or tickets) from the drop-down menu.**

NOTE:

If you do not have either of these applications (NETremote+ or LAN Support Center) loaded, an error box displays notifying you that the executable could not be found. The executables for these applications should be placed in a search path mapping.

See Also:

[Sending a Message to a Current User](#)

[Viewing Current Users' Information](#)

[Releasing a Current User from a Metered Application](#)

Administering Queued Users

From the application usage graph, you can view the names of the individual users waiting to use an application, as reflected by the graph bars.

Once you access this list, you have several administrative options.

Use the following procedure to view who the queued users are.

- 1. In the application usage graph, position the cursor in the area to the right of the application name within the graphic display.**

A small menu box should be attached to the bottom right of the arrow. If there is no menu box, the cursor is not pointing to a place where information is available. This is the area where the number of users (queued, peak and current) displays with graph bars.

- 2. Click here to display a popup menu.**

When the pop-up menu displays, the application in question is outlined in the list along the left hand side of the window.

- 3. Choose the View Queued Users command.**

The Queued Users window displays.

This window lists all the users who are waiting to use this application.

From this window you can:

- * Send a Message to a Queued User
- * Edit Queued User List
- * View Queued Users' Information
- * Launch another BrightWorks capability

Sending a Message to a Queued User

If there are users waiting for an application you can send them a message. This is particularly useful if you want to notify them what number they are in the list, which user may be finishing with the application soon, etc.

Use the following procedure to send a NetWare Send message.

- 1. From the Queued Users window, choose the user to whom you wish to send the message.**

A popup menu displays.

- 2. Choose the Send Message command.**

The Send Message To User dialog box displays.

- 3. Enter the message in the text box provided.**

- 4. Choose the OK button to send the message.**

See Also:

[Administering Queued Users](#)

[Editing the Queued User List](#)

[Viewing Queued Users' Information](#)

[Launching another BrightWorks Capability](#)

Editing the Queued User List

You can add and remove users to and from the queued user list. This is particularly useful if a user does not want to be queued for an application.

Use the following procedure to edit the queued user list.

1. From the Queued Users window, choose the list of Queued Users.

The Edit Queued Users List dialog box displays.

This dialog box displays the following information:

- * List of nonqueued and noncurrent users
- * List of queued users

2. If you want to add users to the queue, select the desired users from the Non-Queued/Non-Current Users list and choose the Include button.

The user is moved to the Queued Users list. Use the Include All button to include up to eight Non-Queued/Non-Current Users.

3. If you want to remove a user from the queue, select the desired users from the Queued User list and then choose the Remove button.

The user is moved to the Non-Queued/Non-Current Users list. Use the Remove All button to remove up to eight non-queued/non-current users at once.

4. Choose the OK button to save your changes and exit.

See Also:

[Administering Queued Users](#)

[Sending a Message to a Queued User](#)

[Viewing Queued Users' Information](#)

[Launching another BrightWorks Capability](#)

Viewing Queued Users' Information

The following information can be displayed about a selected queued user:

- * Login Name
- * Full Name
- * Server
- * Logical Station
- * Network
- * Station Address
- * Time into Network
- * Application
- * Time into Application

Use the following procedure to view this information.

- 1. From the Queued Users window, select the desired user.**

A popup menu displays.

- 2. Choose the User Information command.**

The Queued User Information dialog box displays.

- 3. Choose the Close button to exit this dialog box.**

See Also:

[Administering Queued Users](#)

[Sending a Message to a Queued User](#)

[Editing the Queued User List](#)

[Launching another BrightWorks Capability](#)

Launching another BrightWorks Capability

From the menu, you can also launch another BrightWorks capability (NETremote+ or LAN Support Center) in a context-sensitive fashion. When launching the capability, it automatically configures it to the current user you selected. For example, if you selected Jane and then launched the remote capability, you would have control of Jane's PC.

NOTE:

Users must have the executables for these programs in their search path.

Use the following procedure to launch another BrightWorks capability.

1. **From the Queued Users window, select the desired user.**
2. **Choose the desired capability (either remote or tickets) from the drop-down menu.**

NOTE:

If you do not have either of these applications (NETremote+ or LAN Support Center) loaded, an error box displays notifying you that the executable could not be found. The executables for these applications should be placed in a search path mapping.

See Also:

[Administering Queued Users](#)

[Sending a Message to a Queued User](#)

[Editing the Queued User List](#)

[Viewing Queued Users' Information](#)

Changing the Number of Maximum Concurrent Users

By changing the number of maximum concurrent users, you can increase and decrease the number of licenses on your network as is necessary.

Use the following procedure to change this value.

1. **In the application usage window, position the cursor at the end of the Number Licensed line (i.e., point to the end of the vertical line).**

When you position the cursor on this line, the cursor should change from an arrow to a horizontal black doublearrow.

2. **Click and hold down the left mouse button.**

3. **Drag the line to the desired value.**

The Count box in the status bar at the bottom of the window displays the number of licenses as you move the cursor to select a new value.

NOTE:

You can also modify the Maximum Number of Concurrent Users value by choosing the Define Metered Application option of the Administration Metering Menu. Select the metered application you wish to modify and then choose the Modify button. You can now edit the value.

If you wish to change the value from the Modify Metered Application dialog box, double click on the application name in the left hand side of the application usage graph.

4. **Once you change the value, the Edit License Maximum dialog box displays to confirm your change.**

This dialog box displays the following information:

- * **Server** the current file server
- * **Application** the name of the metered application
- * **Current Maximum** the current maximum number of licenses for this application
- * **New Maximum** the new value you dragged to for this application

5. **If you wish to accept the new Maximum Number of Concurrent Users value, choose the OK button.**

If this value is not the desired number of licenses, you can also edit the value by simply typing over it in the text box.

See Also:

[Changing the Usage Scale](#)

[Changing the Colors Used in the Status Bar](#)

[Hiding or Showing the Status Bar](#)

[Performing Queries](#)

Changing the Usage Scale

The Usage Scale is the scale displayed across the top of the Application Usage window. By changing the high value for the graph, you can increase or decrease the number of users shown. For example, if you have 250 workstations on your network, you can set the high value to 250 to allow the bars to indicate usage by all your network users. If your network is large, but the number of licenses and concurrent users is small, you can set the value lower. With this flexibility, you can change the graph to be best suit your viewing needs.

Use the following procedure to change the usage scale.

1. **While displaying the usage graph, position the cursor in the scale bar directly beneath the title bar.**

The cursor itself should change to a scale.

2. **Double click to display the Edit View Usage Scale dialog box.**

NOTE:

You can also display this dialog box by choosing the Monitoring command from the Administration menu and then the Edit View Usage Scale from the sub-menu that displays.

3. **Choose either Auto-scaled or User-defined.**

- * **Auto-scaled** allows the metering Administration program to determine the maximum value of the scale.
- * **User-defined** allows you to determine the maximum value of the scale.

4. **If you chose User-defined and wish to enter a high value, select the High text box and type the value you wish to use.**

NOTE:

This value cannot exceed the maximum number of users for the file server.

5. **Choose the OK button to save your changes and exit this dialog box.**

The usage scale beneath the title bar now reflects the new maximum you specified.

See Also:

[Changing the Number of Maximum Concurrent Users](#)

[Changing the Colors Used in the Status Bar](#)

[Hiding or Showing the Status Bar](#)

[Performing Queries](#)

Changing the Colors Used in the Status Bar

You can change the colors used in the status bar of the application usage window that reflect the number of current, queued, peak and licensed users. Change these colors to best suit your viewing needs.

Use the following procedure to change these colors.

- 1. In the application usage graph, position the cursor to one of the colors in the status bar.**

The cursor changes to a transparent arrow.

- 2. Double click at this point.**

The standard Windows color palette for that key displays.

- 3. Select a new color.**

- 4. Choose the OK button.**

The new color is shown on the graph bars as chosen.

The colors can be changed for each item in the status bar.

See Also:

[Changing the Number of Maximum Concurrent Users](#)

[Changing the Usage Scale](#)

[Hiding or Showing the Status Bar](#)

[Performing Queries](#)

Hiding or Showing the Status Bar

You can hide or show the status bar on the usage graph. Showing the status bar lets you know which colors indicate current users, queued users, etc., as well as the count when you are changing the number of maximum concurrent users.

Hiding the status bar gives you more room to display the usage graph.

Use the following procedure to conceal the status bar.

From the File menu, choose the Hide Status Bar command.

This toggles the status bar on and off. When the status bar is hidden, this command in the File menu changes to Show Status Bar. By choosing the Show command, you can display the status bar again.

See Also:

[Changing the Number of Maximum Concurrent Users](#)

[Changing the Usage Scale](#)

[Changing the Colors Used in the Status Bar](#)

[Performing Queries](#)

Performing Queries

Every time a query is issued, either manually or automatically, the metering capability updates the usage information on the graph. The Query Indicator is the last item on the status bar. The indicator displays for either manual or automatic queries.

The **Query Now** option allows you to update the application usage information.

Use the following procedure to instruct the metering capability to perform a query at that point.

1. **Select the Monitoring command from the Administration menu. From the sub-menu that displays, choose the Query Now command.**

The Metering function performs a query and updates the Application Usage graph accordingly. The status bar at the right bottom of the window reflects the query's progress.

NOTE:

You can also access this feature by pressing <CTRL><Q>.

The **Set Timer** option allows you to determine how often the metering capability checks with the file server for application usage information.

Use the following procedure to set the timer.

1. **Choose the Monitoring command from the Administration menu. From the sub-menu that displays, choose the Set Monitor Timer command.**

The Set Monitor Timer dialog box displays.

2. **Enter the desired value.**

You can enter a value in seconds from 0 to 9999.

3. **Choose the OK button.**

NOTE:

You can also access this option by pressing <CTRL><T>.

See Also:

[Changing the Number of Maximum Concurrent Users](#)

[Changing the Usage Scale](#)

[Changing the Colors Used in the Status Bar](#)

[Hiding or Showing the Status Bar](#)

Configuration Options

Configuration Options let you attach to and detach from file servers, specify the home directory for data files, and specify whether or not to log historical usage and security data.

Use the following procedure to use this option.

1. **Choose the Metering command from the Administration menu** From the sub-menu that displays, choose the **Configuration Options** command.

The Configuration Options dialog box displays.

You can attach to and detach from file servers using this option.

2. **Select the Home Directory for Data Files text box.**
3. **Type the appropriate directory (using the full path).**
4. **If you wish to track software usage and security information, select the box next to "Log historical usage and security information."**

When the box is checked, the metering capability logs the usage and file protection information used in the reporting function.

The default is to have this box checked.

5. **Choose the OK button.**

See Also:

[Purge Usage Information](#)

[Purge Security Information](#)

Purge Usage Information

This option removes the SITEDATA file from the home directory.

NOTE:

If metering needs to access the file after it is deleted, a new file is automatically created.

Use the following procedure to purge usage information.

1. **Choose the Monitoring command from the Administration menu. From the sub-menu that displays, choose the Purge Usage Information command.**

The Purge Usage Information dialog box displays.

The list box displays your current server.

2. **From the Current Server list box, select the file server from which you wish to purge the usage information.**

If you are not attached to the desired file server, you can use the Attach button.

3. **If you are sure you want to purge all usage information on the specified file server, choose the OK button.**

See Also:

[Purge Security Information](#)

Purge Security Information

This option removes the VIRUSDTA from the home directory.

NOTE:

If metering needs to access the file after it is deleted, a new file is created automatically.

Use the following procedure to purge security information.

1. **Choose the Security command from the Administration menu. From the sub-menu that displays, choose the Purge Security Information command.**

The Purge Security Information dialog box displays.

The list box displays your current server.

2. **From the Current Server list box, select the file server from which you wish to purge the security information.**

If you are not attached to the desired file server, you can use the Attach button.

3. **If you are sure you want to purge all security information on the specified file server, choose the OK button.**

See Also:

[Purge Usage Information](#)

BrightWorks Inventory Help Index

About Inventory

[Inventory Capabilities](#)

[How Does an Inventory Work?](#)

[Inventory Features](#)

[Inventory Modules](#)

[Inventory Configuration Options](#)

How To

[Define Inventory Parameters](#)

[Collect Inventory](#)

[Conduct an Audit](#)

[Review Audit Results](#)

[Maintain the Baseline Inventory](#)

[Generate Inventory Reports](#)

BrightWorks Inventory Capabilities

BrightWorks inventory features assist LAN managers in collecting and maintaining information about the hardware and software components of their LAN. The capabilities focus on performing, recording and reporting the inventory of your standalone and networked PCs and MACs.

The benefits realized from using BrightWorks inventory capabilities include:

- * enabling access to a vast amount of valuable information about your LAN
- * reducing and controlling LAN management and maintenance costs
- * performing inventories quickly, without leaving your desk
- * automatically updating your LAN inventory after each audit
- * gaining control over users changing their own PC settings
- * tracking detailed vendor, warranty and service information on any hardware or software component
- * eliminating the costs of carrying unnecessary or defective equipment on your LAN

How Does a BrightWorks Inventory Work?

BrightWorks inventory capabilities access several modules that collect and maintain information about your network configuration. The collected inventory data includes information regarding all file servers, and networked and stand-alone PC/MAC workstations.

During each inventory, BrightWorks detects the changes made to the network components and updates its database. BrightWorks can be configured to alert you of the inventory changes. The detailed hardware and software records can include serial numbers, purchasing information, warranty and maintenance data. BrightWorks helps you perform your network inventory and audit quickly and easily, giving you complete control over your LAN assets.

BrightWorks Inventory Features

The key features listed below help you maintain a complete asset management system for your LAN.

BrightWorks inventory features include:

- * Recording and updating detailed hardware and software inventory information on networked and stand-alone PCs and MACs, file servers, and remote LANs
- * Automatically integrating remote configurations into the LAN equipment baseline
- * Multiple site support
- * Detecting application software changes and hardware configuration changes
- * Tracking software versions
- * Enabling the viewing of system files
- * Tracking detailed vendor and warranty information
- * Audit scheduling
- * Extensive alerting capabilities
- * Extensive import and export capabilities
- * Pre-defined and custom report generation

BrightWorks Inventory Modules

The BrightWorks inventory capabilities interact with four major functional modules:

- * BrightWorks console and administrative functions
- * PC inventory collection programs
- * MAC inventory collection programs
- * Inventory databases

BrightWorks Console and Administrative Program

BWORKS.EXE is the BrightWorks console and administrative program. This program provides access to all BrightWorks capabilities. This main module is a Windows-based program and is intended to be used by the LAN network manager to perform all asset management functions.

The inventory functions available from the BrightWorks console include:

- * Inventory auditing
- * Site and database administration
- * Inventory viewing and editing
- * Audit notification
- * Pre-defined and custom inventory report generation

PC Inventory Collection Programs

BrightWorks provides two programs for collecting PC inventory:

- * **EQUIP.EXE** - The program used to collect the hardware and software information for networked PCs. EQUIP's command line options allow you to specify how often inventory collection should be performed and what type of item should be inventoried. For example, you may choose to collect hardware information once per day or software information once per week. Inventory collection for networked PCs can also be initiated at login time.
- * **SA_EQUIP.EXE** - The program used to collect the inventory information on stand-alone PCs. Stand-alone PC data is saved to a "collector diskette" and later merged with the LAN inventory information. Using a unique identification method, multiple stand-alone inventories can be conducted without duplicating records in a central database.

Both programs are DOS-based and are executed from the machine on which you want to perform an inventory.

All data collected by the PC inventory collection programs is written to central transaction files and later accessed by BrightWorks administrative functions.

MAC Inventory Collection Programs

BrightWorks provides two programs for collecting MAC inventory. Both programs are used to collect inventory information on stand-alone and networked Macintosh computers; however, their execution is slightly different:

- * **MACEQUIP 3.1** - Performs inventory collection and provides a menu from which you can choose to view the collected inventory and/or close the program.
- * **MACEQUIP AUTOMATIC 3.1** - Automatically closes the program when inventory collection is complete.

Both MACEQUIP programs are MAC-based and are executed from the machine on which you want to perform an inventory.

Macintosh machines on NetWare systems do not execute login scripts; however, the MACEQUIP program can be added as a Startup Item in the System Folder.

Database Access

Every BrightWorks inventory module accesses at least one database. Novell's Btrieve is used to manage the database records.

BrightWorks inventory databases consist of the following:

- * **Transaction inventory files** maintain inventory information detected by EQUIP, SA_EQUIP and MACEQUIP.
- * **Transactions log file** - maintains inventory changes detected during collection by EQUIP, SA_EQUIP and MACEQUIP.
- * **Baseline inventory files** maintain all inventory information following an audit. All information viewed in the main console is obtained from the baseline inventory files. Conducting an audit transfers information from the Transaction Inventory files to the Baseline Inventory files and enables the inventory data to be viewed from the console.
- * **Qualification lists** act as reference lists which aid in data entry and maintain the integrity of the data files.

See Also:

[Baseline Inventory Export/Import](#)

[Maintaining the Baseline Inventory](#)

[Qualification List Set-up and Administration](#)

Inventory Configuration Options

Before you begin to inventory your LAN components, you must verify that the appropriate rights have been granted to the BrightWorks program directory.

Users of NetWare 4.X, 3.X or 2.2 systems need READ, WRITE, FILESCAN, CREATE, and ERASE rights to the directory from which the EQUIP file will be executed. These rights are required in order to update the transaction and log files upon executing the EQUIP.EXE program.

Users of NetWare 2.1X systems need READ, OPEN, SEARCH, WRITE, CREATE, and DELETE rights to the directory from which the EQUIP file will be executed.

Upon the installation of BrightWorks, EQUIP.EXE and all transaction and log files reside in the BWORKS directory.

See Also

[Establishing a Separate Transaction Directory](#)

[Improving Database Performance](#)

[Placing EQUIP in the Login Script](#)

Establishing a Separate Transaction Directory

If granting extensive rights to a program directory (i.e., the Fusion directory) is a concern, then follow the steps below to set up a separate "transaction directory" from which your users can execute EQUIP. *By doing so, your users will no longer need any rights to the Fusion directory.*

Another reason for creating separate transaction directories is for maintaining individual LAN sites. Defining sites is useful for categorizing your inventory data. By establishing a separate transaction directory, you are defining a directory into which a site's inventory data will be collected. For example, you might want to maintain a separate inventory for your Sales Department. To do so, you must define a site for the Sales Department (e.g., the SALES site). You also must establish a separate transaction directory for the SALES site. The collected inventory data for all workstations included in the SALES site will be maintained in the new sales transaction directory.

NOTE:

The BrightWorks program directory must always be defined as a site because it maintains the file server, MAC and stand-alone inventory data.

Perform the following steps to establish a transaction directory in which EQUIP can collect inventory data:

1. Create a transaction directory.

To allow all your network users access to the EQUIP inventory program, create the new directory on the file server. For example, create a directory named F:\FUSION\SALES.

2. Grant rights to the transaction directory.

Users of NetWare 3.X or 2.2 systems need READ, WRITE, FILESCAN, CREATE, and ERASE rights to the directory from which the EQUIP file will be executed. These rights are required in order to update the transaction and log files upon executing the EQUIP.EXE program.

Users of NetWare 2.1X systems need READ, OPEN, SEARCH, WRITE, CREATE, and DELETE rights to the directory from which the EQUIP file will be executed.

3. Copy the required EQUIP files into the new transaction directory.

The files required to execute EQUIP are:

- * EQUIP.EXE
- * BTRIEVE.EXE or BREQUEST.EXE (depending on the Btrieve method you are using)
- * ENDBTRV.EXE
- * WPCSLIST.DAT
- * NRCALL.PID

Several transaction files also need to be in the transaction directory. The first time EQUIP is run in a new transaction directory, you are prompted to instruct EQUIP to create the files.

- * WEQTRN.DAT
- * WEQLOG.DAT
- * WTRSOFTT.DAT
- * WSYSFTRN.DAT
- * WSYSFLOG.DAT

NOTE:

In order for EQUIP to properly execute, you cannot manually copy the transaction files.

4. Copy the required administration files into the new transaction directory.

Several administration files must be copied into the transaction directory; however, *the files must first be configured* using the BrightWorks console program.

- a - To configure the administration files, choose the Software Options command from the Administration menu to display the Software Options dialog box.
- b - Select the Workstation software option and define the unknown file extensions and system files to be identified.
- c - Choose the OK button to accept the configuration changes, and exit BrightWorks.
- d - Copy the following administration files into the new transaction directory:
 - * WAUDCFG.DAT
 - * WFIDPC.DAT
 - * WIDPCSFT.DAT
 - * WSYSFLST.DAT

5. Define the transaction directory as a Site.

This can be accomplished in one of two ways:

- a - From the BrightWorks menu bar, choose the Inventory command from the Administration menu, and then choose the Define Sites command to define the new site and its path, or
- b - When EQUIP first executes in the new transaction directory, you will be prompted to enter a Site name (if the Site has not yet been defined). You must then also define the site and its path from within BrightWorks by choosing the Inventory command from the Administration menu, and then choosing the Define Sites command.

At this point, the new transaction directory is established. To include the transaction site data in your baseline (i.e., include the data in an audit), choose the Inventory command from the Administration menu, and then choose the Scope of Audit command and include the transaction site in the audit scope.

IMPORTANT:

- a - *From within the BrightWorks console, any changes made to the PC Software List, the "System Files to Inventory on Local Site" option, the "Unknown Files to Identify on Local Site" option or the Audit Parameters window options will update the files in the BrightWorks program directory. (The file names are WPCSLIST.DAT, WSYSFLST.DAT, WFIDPC.DAT and WAUDCFG.DAT, respectively.) To maintain the changes, you must copy the updated files into the transaction directory before running EQUIP again.*
- b - *Before you perform an audit, the WIDPCSFT.DAT file (the list of unidentified software) in the transaction directory needs to be appended to the same file name in the Fusion directory. Use a third party utility to append one file to another file. For example, Novell's BUTIL.EXE or Magic Solution's BU.EXE can be used to do this. (BU.EXE is shipped with BrightWorks. It is located in the self-extracting file named TOOLS.EXE in the BrightWorks program directory.)*

For example, the procedures for using BUTIL.EXE to append the files are as follows:

1. *Load Btrieve. (Either BTRIEVE.EXE or BREQUEST.EXE can be used.)*
2. *At the DOS prompt, enter the BUTIL -copy command, which has the following syntax: BUTIL -COPY <source file> <target file>. The command you enter may look similar to the following:*


```
BUTIL -COPY f:\transdir\widpcsft.dat f:\fusion\widpcsft.dat
```

3. Use the ENDBTRV command to unload Btrieve.

Note also that the BU.EXE COPY syntax is exactly the same as the BUTIL syntax; simply replace the BUTIL command with BU.

Consider Improving BrightWorks Database Performance

BrightWorks uses the Novell Btrieve Record Manager as its record manager. Btrieve is integrated with NetWare and offers an extremely high performance mechanism for storing information. In addition, Btrieve is the basis for Novell's Network Management System (NMS), which allows McAfee to integrate with this important management platform more easily.

NOTE:

Refer to "[NMS Smart-Launch Support](#)" for instructions on enabling BrightWorks smart-launch within NMS.

Btrieve must be loaded before running the EQUIP program. There are two methods of implementing Btrieve:

- * **Serverbased** All data processing is done at the file server by the Brequestor (the Btrieve NLM or VAP). Each workstation communicates with the NLM or VAP by loading BREQUEST.EXE in the local PC's memory.
Brequest uses 31-45KB of RAM, depending on the options specified. (Version 6.10 or greater is required when using Brequest with BrightWorks.) It is much faster than the local Btrieve; however, it requires that the NLM or VAP be loaded on the file server.
- * **Clientbased** Workstations load an executable version of the record manager (BTRIEVE.EXE) and perform all data processing locally.
Btrieve uses approximately 85KB of RAM and is much slower than Brequest.

NOTES:

- a - Refer to "[Using Brequest](#)" for general instructions for configuring the Btrieve NLM. Refer to your Novell documentation for details on configuring Btrieve.*
- b - When running the Brequestor, BSPXCOM and BROUTER must also be loaded. Refer to your Novell documentation for details on loading these programs.*
- c - Verify that you are running the latest versions of the Btrieve files. Updated Btrieve files can be found on CompuServe in the Novell Libraries (GO NOVLIB).*

EQUIP is fully compatible with both methods of access. BrightWorks is shipped with BTRIEVE.EXE; however, it is highly recommended that you use the serverbased method while running EQUIP, as this will improve the performance of data collection by at least 50% and as much as 500%.

BrightWorks provides two batch files for loading Btrieve:

- * **BTR.BAT** - loads local Btrieve (BTRIEVE.EXE)
- * **BRQ.BAT** - loads server-based Btrieve (BREQUEST.EXE)

All batch files provided with BrightWorks are placed in the BrightWorks program directory upon installation.

Consider Placing EQUIP in the Login Script

To ensure that EQUIP is executed on a regular basis, the EQUIP command can be placed in the system login script. The following example illustrates how EQUIP can be executed from within a system login script.

Note that BTRIEVE or BREQUEST must be loaded before EQUIP is run and unloaded after EQUIP has completed gathering the inventory.

```
....  
MAP F:=FS/SYS:FUSION  
DRIVE F:  
#BREQUEST /D:17000  
#EQUIP /H /S  
#ENDBTRV  
....
```

where F:=FS/SYS:FUSION is the drive ID and complete path where the BrightWorks files are stored.

NOTE:

When mapping to the Fusion directory, make sure that you use a straight logical map. MAP ROOTS and MAP INS are not allowed.

See Also

[EQUIP Command Line Reference](#)

Defining Inventory Parameters

Access to Inventory Administrative Functions

Inventory administrative functions are accessed by choosing the Inventory command from the Administration menu. From the sub-menu that displays, choose the desired command.

Inventory Set-up and Administration

Refer to the following topics for instructions on defining inventory parameters:

- Audit Set-up - Audit parameters refer to the settings and options that indicate where, how and when to run the inventory audits for your network(s). Before running an audit, BrightWorks automatically refers to the defined options that can be changed as needed.
- Alerting Set-up - BrightWorks can be configured to notify you of any inventory changes upon completion of an audit. Notification options allow you to receive notices via printed output, cc:Mail, or various other electronic communications methods.
- Inventory Qualification Lists - BrightWorks provides several databases that contain standard types of equipment, software and contracts that commonly apply to a network. BrightWorks uses these databases, or "qualification lists," for data validation and for compiling equipment inventory records. You can add to, modify and remove entries from these databases to reflect your own environment.
- Inventory Data Import/Export - BrightWorks baseline inventory data can be exported for use in other applications (e.g., a spreadsheet or database application). Likewise, a spreadsheet or database file can be imported into BrightWorks.

Audit Set-up and Administration

After the inventory collection programs have been run on your networked and stand-alone PCs and MACs (i.e., EQUIP, SA_EQUIP and MACEQUIP), the inventory data is ready to be audited. Audits should be performed on a periodic basis to keep your inventory accessible and current.

The first time an audit is performed, a "baseline inventory" list is generated. Every subsequent audit updates the baseline inventory to reflect the most recently collected inventory data.

Refer to the following topics for the procedures that should be performed before running an audit:

Defining LAN Sites

Setting the Scope of the Audit

Defining Audit Parameters

Identifying the Software to be Inventoried

NOTE:

To verify that the current parameters accurately reflect your intended audit, the audit settings should be reviewed before running your audit. Audit settings can be adjusted as needed.

Defining LAN Sites

Defining sites is useful for categorizing your inventory data. For example, a company that has a LAN with two file servers might want to define two sites. Assuming that the Marketing Department uses one file server and the Sales Department uses the other file server, the two sites might be named MARKETING and SALES. Defined in this way, an audit can be performed which compiles an inventory list for one or both departments. The hardware and software components for the individual departments (or sites) can be separately maintained.

A site is actually a directory (referred to as a "transaction directory") from which EQUIP is run. It contains, at the very minimum, the inventory transaction files. In the example above, all workstations in the SALES site execute EQUIP from within the SALES site directory. As a result, the collected inventory data for the SALES site workstations is maintained in the SALES transaction directory.

NOTES:

a - Refer to Establishing a Separate Transaction Directory for procedures on defining a transaction directory.

b - The BrightWorks program directory must always be defined as a site because it maintains the file server, MAC and stand-alone inventory data.

Use the following procedure to define a LAN site, change site information, or remove a site from inventory consideration.

- 1. Choose the Inventory command from the Administration menu. From the sub-menu that displays, choose the Define Sites command.**

The Define Sites dialog box displays.

This dialog box lists all of the sites that are defined in your BrightWorks configuration. It also indicates the directory in which the site data is maintained.

NOTE:

The Local Site (i.e., the site name defined for the BrightWorks program directory) is displayed in red.

- 2. Choose one of the following:**

- * **To add a site**, choose the Add button. The Add Site dialog box displays. Go to Step 3.
- * **To change the transaction directory of an existing site**, highlight the site to be changed and choose the Edit button. The Edit Site dialog box displays. Go to Step 4.
- * **To remove a site**, highlight the site to be removed and choose the Delete button. A dialog box displays the prompt "Do you really want to delete this item?" Choose the Yes button to remove the site. Go to Step 5.

NOTE:

The Local Site cannot be deleted.

- * To exit the Define Sites dialog box, choose the Close button.

- 3. To add a site in the Add Site dialog box, enter the site name in the Site Name field.**

Site names can be up to 25 characters (including spaces).

4. Specify the drive and directory combination which identifies the site.

To define or modify the site's path, click on the entries in the Directories and Drives lists.

NOTE:

The Directories and Drives lists only display the information for the file server volumes and directories to which you are mapped.

5. Choose the OK button to save the site information and return to the Define Sites dialog box.

6. Choose the Close button to save the new or changed site information and exit the Define Sites dialog box.

NOTE:

Sites are not included in BrightWorks audits unless they are specified using the Scope of Audit dialog box. See the next procedure, "Setting the Audit Scope."

See Also:

[Setting the Scope of the Audit](#)

[Defining Audit Parameters](#)

[Identifying the Software to be Inventoried](#)

Setting the Audit Scope

The scope of an audit is defined by selecting the sites and file servers to be included in the audit.

For example, if you administer a LAN on which you have defined two sites, you might want to perform an audit that includes only SITE1's inventory data. (This might be the case if you have not collected the updated inventory data for SITE2.) To do so, you would include only SITE1 in the audit scope, and then run the audit.

Use the following procedure to define the scope of an audit.

1. **Choose the Inventory command from the Administration menu. From the sub-menu that displays, choose the Scope of Audit command.**

The Scope of Audit dialog box displays. Sites included in the audit are listed on the top right side of the dialog box; sites excluded from the audit are listed on the top left side of the dialog box.

File servers included in the audit at the local site (as identified during the BrightWorks installation and set-up) are listed on the bottom right side of the dialog box; file servers excluded from the audit at the local site are listed on the bottom left side of the dialog box.

NOTE:

All file servers on the network are listed in the Scope of Audit dialog box; however, only those file servers on which BrightWorks is installed can be audited, even if the file server name is "included" in the Scope of Audit.

2. **Choose one of the following:**

- * **To include a site or file server in the audit**, click on the item name and choose the Include button. The selected item moves from the left side of the dialog box to the right side.
- * **To remove a site or file server from the audit**, click on the item name you want to remove and choose the Remove button. The selected item moves from the right side of the dialog box to the left side.

3. **Choose OK to save the site(s) and server(s) specified.**

The Scope of Audit dialog box is closed.

NOTE:

When including file servers in the audit scope, a Server dialog box displays prompting you to enter your login name and password. Enter a login name which has supervisor or equivalent rights. Press the <Tab> key to move to the Password field, and enter your password. Choose the OK button to continue.

See Also:

[Defining LAN Sites](#)

[Defining Audit Parameters](#)

[Identifying the Software to be Inventoried](#)

Setting the Audit Software Options

The software that is identified during an audit or workstation inventory depends on the applications listed in the PC and MAC Software Lists, as well as the "software options" that you have defined for the audit. The process of setting software options instructs the audit to identify specific files. Nonsystem file extensions can be listed for identification on file servers or PC workstations (e.g., application-specific software that you want to monitor). System files also can be tagged for identification. Files that have extensions other than those listed will not be identified in an audit.

For example, all PC-based files with the extension .EXE and .COM are executable files and, by default, are included in an audit. However, if you also want to include all files having the extension .BAT, you must list the .BAT extension as a software option.

NOTE:

Unidentified software is only collected if the "Unknown Files to Identify on Local Site" options for Fileserver, Workstation and/or Macintosh are checked in the Software Options dialog box. If none of these options are checked, then only the applications listed in the PC or MAC Software Lists are included in the inventory and audit.

Use the following procedure to identify the software to be audited on your file servers and workstations.

- 1. Choose the Inventory command from the Administration menu. From the sub-menu that displays, choose the Software Options command.**

The Software Options dialog box displays.

The top portion of the Software Options dialog box allows you to instruct BrightWorks to identify specific file extensions during an audit. The bottom portion of the dialog box allows you to instruct BrightWorks to identify specific system files to be considered for inventory/audit purposes at the local site.

- 2. To specify or manage the file extensions identified during an audit, click on either Fileserver or Workstation, depending on where the file(s) will be inventoried. Then choose the Options button.**

The File Extensions to Identify dialog box displays.

Follow the steps below to do the following:

To add another file extension for a local workstation or file server:

- a - Choose the Add button. The Add File Extension to Identify text box displays.
- b - Type the file extension to be inventoried/audited (e.g., BAT), and choose OK to save the new extension. Choose the Close button to continue, and proceed to Step 4.

To edit a non-system file extension for a local workstation or file server:

- a - Highlight the desired file extension, and choose the Edit button. The Edit File Extension to Identify text box displays with the selected extension.
- b - Change the file extension, and choose OK to save the change. Choose the Close button to continue, and proceed to Step 4.

To delete a nonsystem file extension from a local workstation or file server for inventory/audit purposes only:

- a - Highlight the desired file extension, and choose the Delete button. A dialog box displays with the prompt "Do You Want to Proceed?"

- b - Choose Yes to remove the extension. Choose the Close button to continue, and proceed to Step 4.

NOTE:

Checking the Macintosh option in the Software Options dialog box has the effect of including all Macintosh applications in the MAC inventory (i.e., those applications that are not listed in the Macintosh Software List). All MAC applications will be included in the equipment inventory and audit.

3. To add, change or remove a system file in the audit consideration, follow the steps below:

To add a local system file for inventory/audit consideration:

- a - In the bottom portion of the Software Options dialog box, choose the Add button. The Add System File to Inventory text box displays.
- b - Enter only the name of the system file you want to add (not the path), and choose OK. Proceed to Step 4.

To edit a local system file for inventory/audit consideration:

- a - Highlight the desired system file, and choose the Edit button. The Edit System File to Inventory text box displays with the selected file name.
- b - Change the file name, and choose OK to save the change. Proceed to Step 4.

To delete a local system file from inventory/audit consideration:

- a - Highlight the desired file and choose the Delete button. A dialog box displays with the prompt "Do You Want to Proceed?"
- b - Choose Yes to remove the file. Proceed to Step 4.

NOTE:

All system files must be in the workstation's search path.

4. Choose the OK button in the Software Options dialog box to save your selections.

The Software Options dialog box closes.

See Also:

Defining LAN Sites

Setting the Scope of the Audit

Defining Audit Parameters

Defining the Audit Parameters

Audit parameters identify the following:

- * the software and hardware to be audited
- * the audit schedule
- * whether the BrightWorks baseline inventory file should be backed up before the audit
- * whether or not an alert should be made if changes are detected during an audit

Use the following procedure to define the audit parameters:

- 1. Choose the Inventory command from the Administration menu. From the sub-menu which displays, choose the Audit Parameters command.**

The Audit Parameters dialog box displays.

Define the audit settings by selecting any/all of the following:

- a - To define the components to be audited**, select either Software or Hardware or both (as applicable) for each of the following categories: PCs, MACs, and Fileservers. An 'x' appears in the box associated with each selected option.
- b - To identify when BrightWorks audits should occur**, choose one of the following options:
 - * To run the audit automatically, click on Once Per Hour, Once Per Day, Once Per Week, or Once Per Month as applicable. Then click on the box to the right of your choice and enter the desired time or day of the audit.
 - * To run the audit only on request, click on When Requested.
- c - To back up the BrightWorks baseline before each audit**, select the Backup Baseline setting in the Additional Options area. The Settings button is enabled and, when chosen, displays the Backup Baseline Settings dialog box listing the four BrightWorks inventory files to be backed up.
Enter the backup file names, and specify the drive, and directory (and subdirectory as applicable) to which the baseline files should be saved.

NOTE:

The backup file names must be explicitly entered; wildcards are not allowed.

Choose OK to save the baseline settings.

- d - To be notified of certain conditions and changes detected during the audit**, select the Alert if Changes Occur setting in the Additional Options area.

NOTE:

The conditions and changes for which you want to be alerted are defined in the Alerting Options dialog box. Refer to "[Defining Alerting Criteria](#)."

- 2. Choose the OK button to save the audit parameters.**

To close the dialog box without saving your changes.

NOTE:

When an audit has been scheduled, a [Timer icon](#) displays in the BrightWorks application window.

See Also:

Defining LAN Sites

Setting the Scope of the Audit

Identifying the Software to be Inventoried

Alerting Set-up and Administration

BrightWorks can be configured to automatically alert you of inventory changes that are detected during an audit.

An "alert" consists of one or more change criteria for which you want to be notified, the method of notification and the alert schedule. Therefore, the following three steps are required to define an alert:

1. **Define alerting criteria.** For example, you might want to be notified if a new node has been detected during the audit or if a system file has been deleted.
2. **Select the method of notification.** Notification of the changes in inventory can be made via numeric pager, MHS message, cc:Mail message, BrightWorks report, NetWare broadcast message or special program.
3. **Set the alert schedule.** Alerting can be performed in one of several intervals. For example, you might want to be notified of the changes in inventory only once per month, or you might want to be notified once per hour.

NOTE:

When all three steps above are performed to set the alert, a Timer icon displays in the BrightWorks application window.

See Also:

The Audit Timer Icon

Defining Alerting Criteria

Use the following procedure to define the inventory changes to which you want to be alerted.

1. **Choose the Inventory command from the Administration menu. From the sub-menu which displays, choose the Alerting Options command.**

You can also choose the Alerting tool bar button to display the Alerting Options dialog box.

The first time you establish alert parameters, no values appear in either the Criteria or Change columns. Once you have defined alert parameters, the Criteria column displays specific classes and descriptions or file names for the alert, and the Change column displays the reason for the alert.

2. **To remove an alerting criteria item, highlight the criteria to be removed and choose the Remove button.**

The specified Criteria/Change line is removed.

3. **To add an alerting criteria item, choose the Add button.**

The Alerting Criteria dialog box displays.

4. **In the Item Changed category, select the item to be monitored.**

You can be alerted to changes in either equipment or system files. For example, if you want to be alerted to additions to the hardware inventory, then select the Equipment option in the Item Changed category. If you want to be alerted to changes to system files, then select the System File option.

5. **In the Source Database category, select the database to be monitored.**

You can be alerted to changes in either the transaction database (i.e., inventory before the audit) or the baseline database (i.e., inventory after the audit). For example, if you want to be alerted to changes made to the inventory data compiled after an audit, then select the Baseline option in the Source Database category. If you want to be alerted to changes in the inventory data before the audit is performed, then select the Transaction option.

6. **In the Filters area, define the filtered data to be monitored.**

From the drop-down lists, select the Category, Class and/or Description of the item to be monitored. For example, if you want to be alerted to changes in the base memory of all file servers on your network, then enter "File Server Hardware" in the Category field and "Base Memory" in the Class field.

NOTE:

When monitoring System Files, filter criteria applies to the system file names. Note that either one specific system file name or <All> system file names that are listed in the Software Options dialog box can be selected for monitoring.

7. **In the Change category, select the reason for the alert.**

For example, if you are monitoring Equipment and you want to be alerted of the removal of any 386 PCs, then select the Removed option. (In this example, 386 would be specified as the Class filter.) If you are monitoring System Files and you want to be alerted of any additions, then select the Added option.

The following Change options also are available:

- * **Changed** - alert on any changes made to the inventory.
- * **New Node** - alert on any new nodes added to the inventory.

8. **Choose the OK button to save your changes and exit the Alerting Criteria dialog box.**

When the Alerting Criteria dialog box is closed, the Alerting Options dialog box becomes active. If you have not already done so, you can:

- * Choose the Notifications button to review or change the method by which you want to be notified.
- * Choose the When button to review or change the notification schedule.
- * Exit the Alerting Options dialog box by choosing the OK button to save your changes.

See Also:

Selecting the Method of Notification.

Setting the Alert Schedule.

The Audit Timer Icon

Setting the Method of Notification

Use the following procedure to select the method by which you want to be notified of inventory/audit changes.

1. From within the Alerting Options dialog box, choose the Notifications button.

The Notification Options dialog box displays. The *selected* methods of notification display on the right side of the dialog box; the *excluded* methods of notification display on the left side of the dialog box.

2. Define the notification method.

To select a method of notification, click on the method listed in the left side of the dialog box, and choose the Include-> button. A dialog box displays allowing you to enter details regarding the selected method. Refer to Step 3.

To remove a method of notification, click on the method listed in the right side of the dialog box, and choose the <-Remove button. Your selection moves from the right side of the dialog box to the left side.

3. Enter details associated with the selected method of notification, and choose the OK button.

For example, if you choose the Run Fusion Report notification method, you have the opportunity to select a specific report type to be printed. If you choose the cc:Mail Message option as the method of notification, a dialog box displays prompting you to specify your cc:Mail name and password, as well as the user name or group name to whom you want to send the message.

The available notification options and the information required for their use are as follows:

- * **NetWare Message** - Select a Server and Login Name from the drop-down list associated with each field.
- * **cc:Mail Message** - Enter your cc:Mail user name in the From field and your cc:Mail Password; enter the Mail Drive Letter and cc:Mail user name of the person or group to receive the notification.
- * **Run a Program** - Enter the name of the program to be run as the alert. The program can be either a DOS or Windows program with the extensions .EXE, .COM or .BAT, and it must be in the current path.
- * **MHS Message** - Enter the MHS Mail Drive, the MHS user name in the From field and the associated Host; enter the MHS user name of the person to receive the notification in the To field and the associated Host; enter the Application and Routing Information.
- * **Numeric Pager** - Enter the Name of the person to be paged, their Phone Number, Baud Rate, Port and Delay criteria.
- * **Run Report** - Select a report to be printed from the list of available BrightWorks inventory reports.

Upon choosing OK, you are returned to the Notification Options dialog box, and the selected method moves from the left side of the dialog box to the right side.

4. Choose the Save button to save your changes and exit the Notification Options dialog box.

You are returned to the Alerting Options dialog box.

See Also:

[Defining Alerting Criteria.](#)

[Setting the Alert Schedule.](#)

The Audit Timer Icon

Setting the Alert Schedule

Use the following procedure to define the frequency of notification.

1. From within the Alerting Options dialog box, choose the When button.

The Alert Scheduling dialog box displays.

2. Define the notification schedule.

Click in the radio button that corresponds to the desired notification schedule. For example, to be notified of the inventory changes on a weekly basis, click in the radio button to the left of the Once Per Week option.

If you choose the Once Per Hour, Once Per Day, Once Per Week or Once Per Month options, additional information is required. For these options, you must specify the exact time of day and/or the number of the day that you want to be notified, as appropriate. For example, if you want to be notified once per month, you must specify which day of the month you want to be notified, as well as specify the time of day (e.g., Day No.1 at 10:00 AM to be notified at 10:00 AM on the first day of each month).

To be notified immediately, click on the Right Now option.

To stop notification via electronic communications, click on the Stop Alerting option.

3. Choose the OK button to save the alert schedule and exit the Alert Scheduling dialog box.

You are returned to the Alerting Options dialog box.

See Also:

[Defining Alerting Criteria.](#)

[Selecting the Method of Notification.](#)

The Audit Timer Icon

The Audit Timer Icon

When an alert has been set and/or when an automatic audit has been scheduled, a Timer icon displays in the BrightWorks application window.

NOTE:

Only one alert can be defined at any time; however, the alert can consist of multiple change criteria (e.g., the alert might be triggered if a system file has changed, a node has been added or equipment has been removed). Similarly, only one audit can be scheduled at any time.

Use the following procedure to view the timers that have been set.

1. Double click on the Timer icon.

The Active Timers dialog box displays.

This dialog box lists the Timer Name and indicates the next time the "event" should occur. (An "event" is either a scheduled audit or an alert.)

2. To delete a timed event, select the event to be deleted and choose the Delete button.

3. To modify the configuration of an event, select the event and choose the Configure button.

If the selected event is an alert, then the Alerting Options dialog box displays. Modify the information, and choose the OK button to return to the Active Timers dialog box.

If the selected event is a scheduled audit, then the Audit parameters dialog box displays. Modify the audit schedule, and choose the OK button to return to the Active Timers dialog box.

4. Choose the Close button to close the Active Timers dialog box.

See Also:

[Defining Alerting Criteria.](#)

[Selecting the Method of Notification.](#)

[Setting the Alert Schedule.](#)

Qualification List Set-up and Administration

The inventory databases provided with BrightWorks are referred to as "qualification lists." As you collect inventory and perform audits, BrightWorks maintains your inventory information in several categorized databases. This enables you to access the information easily. BrightWorks automatically updates and maintains the qualification lists; however, you can customize the data to reflect your network environment.

The following topics provide procedures for viewing and maintaining the database records and describe the inventory qualification lists that are provided with BrightWorks.

[Viewing Qualification Lists](#)

[Modifying Qualification Records](#)

[Qualification List Descriptions](#)

Viewing Qualification Lists

Use the following procedure to view the contents of a qualification list.

1. **Choose the Inventory command from the Administration menu. From the sub-menu which displays, choose the Qualification Lists command.**

The Qualification Lists option is also a sub-menu that lists the BrightWorks inventory databases.

2. **Choose the database you want to access.**

A database is chosen by clicking on the database name. The records of the selected database display. To review all database records, use the arrow buttons to scroll through the list.

At this point, the database information can be modified. Refer to the topic entitled [Modifying Qualification Records](#).

3. **Choose the Close button to exit the database.**

See Also:

[Modifying Qualification Records](#)

[Qualification List Descriptions](#)

Modifying Qualification Records

The inventory database information provided with BrightWorks can be customized to apply specifically to your network environment. For example, you might want to maintain a list of the hardware manufacturers who supply your network equipment. In addition to the Manufacturers determined by the BrightWorks inventory collection programs (i.e., EQUIP, SA_EQUIP or MACEQUIP), you might want to list and track another manufacturer.

Use the following procedure to modify a database record.

1. **Choose the Inventory command from the Administration menu. From the sub-menu which displays, choose the Qualification Lists command.**

The Qualification Lists option is also a sub-menu that lists the BrightWorks inventory databases.

2. **Choose the database you want to modify.**

A database is chosen by clicking on the database name. The records of the selected database display.

3. **Modify the database record information.**

The modifications that can be made to a record depend on the selected database. As mentioned above, many database records provided with BrightWorks cannot be edited or deleted.

Possible modifications include the following:

- * **Adding a record** - To add a new record, choose the Add button. An Add dialog box displays prompting you to specify required information about the new record. Enter the information, and choose OK. The new record is added to the qualification list.
- * **Editing a record** - To edit an existing record, select the record from the list and choose the Edit button. An Edit dialog box displays with the selected record's information. Make changes to the information, and choose OK. The record's information is updated.
- * **Deleting a record** - To delete an existing record, select the record from the list and choose the Delete button. A prompt displays asking "Do you Want to Proceed?" Choose the Yes button to delete the record.
- * **Printing all records** - To print all records in the database, choose the Print button. The entire database is sent to the printer as defined by your current print parameters.
- * **Unmarking all records** - The MAC and PC Software Lists have an Unmark option. To deselect all highlighted records, choose the Unmark button. (A record is highlighted or selected by clicking on the record entry. Multiple records can be selected by clicking on them one at a time.)
- * **Sorting all records** - The MAC and PC Software Lists have a Sort option. To change the sorting order for all records in a list, choose the Sort button. A dialog box displays listing the available sort options. Select a sort order, and choose OK. The records are sorted and displayed according to the selected method.
- * **Hiding/Unhiding a record** - The MAC and PC Software Lists have a Hide option. To keep certain records hidden from the software inventory list while performing an audit, select the record(s) and choose the Hide/Toggle button. A checkmark displays next to all hidden records. The hide action works as a toggle (e.g., if a hidden record is selected when Hide/Toggle is chosen, the record will be unhidden). Multiple records can be selected by clicking on them one at a time. Hidden records can be viewed in the database list; however, they will not appear in an inventory list.

NOTES:

a - Many Macintosh applications consist of only one file; therefore, be careful when hiding files in the

MAC Software List.

b - To hide a file from inventory, the file must be hidden before EQUIP is run on a machine which has the file. Once a file is detected by EQUIP, the file is recorded in inventory upon running an audit. If it is subsequently hidden, the file will be considered as "missing" after the next audit is performed.

4. Choose the Close button to close the database dialog box.

See Also:

[Viewing Qualification Lists](#)

[Qualification List Descriptions](#)

Inventory Qualification List Descriptions

This topic describes the contents of the BrightWorks inventory qualification lists. Note that not all of the lists are populated with information upon the installation of BrightWorks. For example, there are no records in the Components and Models lists until you perform an audit.

Categories Database

A Category is the broadest classification of LAN equipment, identifying hardware and software for PCs, MACs, Fileservers and Fax Servers. Examples of records on this database are File Server Hardware, File Server Software and PC Hardware. One 30 character field defines a category record.

Classes Database

Classes provide further definitions for the Category and Type records. They include the individual components and addons which make up a category or a type (e.g., Base Memory, Keyboard, Monitor, Ports).

Components Database

The Components database gathers information from several other database records to develop a complete record for a specific hardware component or software item. The key fields which must be specified when adding or editing a component record are:

- * Category
- * Class
- * Manufacturer
- * Product Name

Secondary/optional component record fields are:

- * Model
- * Vendor
- * Maintenance Vendor
- * Contract Type
- * Warranty Information (start date and length of warranty)

All fields of a component record can be selected from dropdown lists, with the exception of Product Name which must be manually entered.

Contract Types Database

The Contract Types database defines the generic types of contracts that are in effect in your company. For example, you can specify a contract type as the period of time a contract is effective (e.g., Two Years) or as a specific type of contract (e.g., Maintenance Only). One ten character field defines a contract type record.

Departments Database

Departments can be defined as:

- * An individual organization or group in your company, for example, Sales, Finance, or Customer Service.

- * A line of business or profit center, for example, retail operations or some product-specific definition.
- * Any logical grouping of people in your company, for example, MISSouthwest Operations and MISWestern Operations.

Departments are used to identify the hardware and software in specific areas during an inventory/audit. One 30 character field defines a Department record.

Locations Database

The Locations database contains a name for each site and/or physical location at which BrightWorks inventories are performed. Examples are Lab, Shipping or Accounting. One 25 character field defines a locations record.

MAC Software List Database

The MAC Software database provided with BrightWorks contains a list of popular Macintosh software applications. The software detected by MACEQUIP during an inventory is compared against this list. Each application record includes a Product Name (49 characters), File Creator (4 characters), Version (9 characters). A sub-set of this database that can be viewed when editing a record includes fields for File Size (7 characters), Manufacturer Name (selected from the Manufacturers database) and Category (an optional field selected from the Software Categories database). You have the option of sorting this database for display in File Creator order or Product Name order.

Many applications require more than one executable file to run the software. To accommodate this and reduce the number of files listed in the software databases, BrightWorks lets you mark a particular file to be considered as the key or identifying component of a software package. The package identifier is based on the package name, file size and version number.

Manufacturers Database

The Manufacturers database contains the names of companies that produce equipment and products or provide services (e.g., McAfee Associates Inc., Microsoft). A subset of this database allows addresses, phone, and FAX numbers of each manufacturer to be captured. One 30 character field defines a Manufacturers record.

Models Database

The Models database contains records for each model used in your network (e.g., Compaq 386, PS2). One 25 character field defines a Models record.

Names Database

The Names database contains a record for each network user. One 48 character field defines a user name record.

PC Software List Database

The PC Software database provided with BrightWorks contains a list of popular PC software applications. The software detected by EQUIP during an inventory is compared against this list. Each application record includes a File Name (12 characters), Product Name (49 characters), Version (9 characters). A sub-set of this database that can be viewed when editing a record includes fields for File Size (9 characters) and Manufacturer Name (selected from the Manufacturers database). You have the option of sorting this database for display in File Name order or Product Name order.

Many applications require more than one executable file to run the software. To accommodate this and reduce the number of files listed in the software databases, BrightWorks lets you mark a particular file to be considered as the key or identifying component of a software package. A package identifier is based

on the package name, file size and version number, and it is marked as the "identifier" by choosing the Hide/Toggle button to hide all the other supporting files (i.e., the identifier is the only file that *is not* hidden).

Software Categories Database

The Software Categories database classifies various types of software according to their general purpose. Several Software Categories database records are provided with BrightWorks (e.g., games, spreadsheets, word processors). Additional records can be added, and existing records can be modified.

Types Database

The Types database identifies specific machine types (e.g., File Server, Workstation, MAC, and Spare Parts). One 30 character field defines a types record.

See Also:

[Viewing Qualification Lists](#)

[Modifying Qualification Records](#)

Baseline Inventory Export/Import

About the Baseline Inventory

The baseline inventory is the equipment inventory database that is updated each time an audit is performed. This database is an accumulation of the most currently audited inventory information.

BrightWorks inventory collection programs (EQUIP, SA_EQUIP, MACEQUIP) place the collected hardware and software data in "transaction files." The audit process compiles the transaction files and compares the most recently collected equipment inventory information against the current baseline. As a result, a new updated baseline inventory database is created.

The baseline is made up of equipment inventory records which contain data specific to each type of equipment, as identified in the Types qualification list. The equipment inventory records maintain data such as manufacturer, network address, memory, software applications, and storage. During an audit, information is gathered from the BrightWorks inventory qualification lists and the equipment inventory. The data that is audited depends on the audit parameters that are defined during the audit set-up (e.g., scope, software options). A baseline may even include inventory data from standalone PCs and MACs, provided that the stand-alone inventory data is loaded into the transaction inventory before the audit is performed.

The following topics discuss the procedures that can be performed with the baseline inventory data.

[Exporting the baseline inventory data outside of BrightWorks](#)

[Importing baseline-type files into BrightWorks](#)

[Removing inventory data for a specific site from the baseline](#)

Exporting the Baseline Inventory

Use the following procedure to create an export file of your baseline. In addition to selecting the export file format, you have the option of selecting a specific site whose data you want to export.

1. **Choose the Inventory command from the Administration menu. From the sub-menu which displays, choose the Export Baseline command.**

The Export Baseline dialog box displays.

2. **Choose the site data to be exported.**

To view a complete list of sites, click on the down arrow button to the right of the Site to Export field. Highlight an individual site name, or click on the <All Values> entry to include all site data in the export file.

NOTE:

Either the data from one specific site can be exported, or the data from all sites can be exported.

3. **Define the Equipment Options to be included in the export file.**

Select the Long Identifier option if you want the export file to include the following identifying fields: location, department, asset tag, site, user name, unique ID, component category/class/description/tag, manufacturer, model, vendor, serial #, internal ID, cost center, warranty length/dates, purchase date/price, cost to date, and maintenance vendor/contract type.

Select the Short Identifier option if you want the export file to include all of the above except the location, department, and asset tag fields.

4. **Define the Component Options to be included in the export file.**

This is an optional step. If you want the export file to include specific export data that is extracted from the Equipment Inventory record, choose any number of the available component options.

- * Purchasing Information refers to the purchasing data entered into the Equipment Component dialog box for each workstation.
- * Maintenance Information refers to the maintenance contract data entered into the Equipment Component dialog box for each workstation.
- * Software Details refers to information about the software residing on the workstation or file server (or other type of equipment you may have in your Types qualification list).

NOTE:

The purchasing information, maintenance information, and software details referred to above can be reviewed by choosing the Inventory command from the View menu. Select a specific node from the Inventory dialog box to display the node's equipment details. Choose the Software Applications button to review the software programs associated with the workstation. Choose the Misc Equipment button to review the miscellaneous equipment associated with the workstation.

5. **Choose the Export button.**

The Save Export File dialog box displays, prompting you to enter the export file's name, format, and path information.

6. **Specify the export file information.**

In the File Name field, enter the name you want assigned to the export file. The default dBase extension .DBF is provided. The available formats include:

- * dBase (.DBF)

- * Lotus 1-2-3, Versions 1.0, 2.0 and 3.0 (.WK?)
- * Excel, Versions 2.0, 3.0 and 4.0 (.XL?)
- * VisiCalc (.DIF)

Click on the down arrow button to the right of the Save File as Type field to display a list of formats. Select the format in which you want the export file configured.

Specify the drive and directory path into which the export file should be saved.

7. Choose the OK button to begin the export process.

The export status screen displays. A status bar shows the percentage complete of the export. When the export is finished, the status screen is closed.

See Also:

[Importing baseline-type files into BrightWorks](#)

[Removing inventory data for a specific site from the baseline](#)

Importing a Baseline Inventory File

Use the following procedure to import baseline inventory data into BrightWorks. The imported data is added directly into your current baseline inventory file.

1. **Choose the Inventory command from the Administration menu. From the sub-menu which displays, choose the Import Baseline command.**

The Open File to Import dialog box displays.

2. **Specify the imported file name and file format.**

If you know the name of the file to be imported, enter the full path name into the File Name field. You can search for the full path name by clicking on entries in the Directories and Drives lists.

Click on the down arrow button to the right of the List File as Type field to display a list of formats. Select the format in which the import file is configured.

3. **Choose the OK button to continue.**

The Import File dialog box displays.

4. **Define the field-to-field relationships between the import file and the baseline inventory file.**

The Import File dialog box allows you to define the relationship between the data fields in the import file and those in the BrightWorks baseline inventory file. The dialog box consists of the following areas:

- * The Fusion Data Fields list contains every data field in a BrightWorks baseline record.
- * The Import File Data Fields list contains every data field in a record in your import file.
- * The Included Fields list identifies the relationship between the BrightWorks Baseline fields and those from the Import File.

The total number of records in the import file and the number of data fields per import file record are shown at the bottom of the dialog box.

To define the field-to-field relationship, highlight a BrightWorks data field from the Fusion Data Fields list, and then choose the Include button. Then move to the Import File Data Fields list, and highlight the data field which corresponds to the selected BrightWorks field. Repeat for every applicable field to build the BrightWorks baseline record.

The selected fields appear in the Included Fields area. BrightWorks data fields are shown in capital letters. Import data fields are shown in lower case letters.

For example, if your import file has a field called LOC which is equivalent to BrightWorks SITE field, the Included Fields entry appears as:

SITE = > LOC

NOTES:

a - Relationships must be defined for the Site, Name, Node ID1, Node ID2 and Type fields.

b - Imported Site names must be unique.

c - The imported file data fields must be accurately matched to the BrightWorks data fields to avoid misplacing the data in the baseline file.

5. **Choose the Import button to begin the import process.**

The import status screen displays. A status bar shows the progress of the import. When the import is complete, the status screen is closed.

See Also:

[Exporting the baseline inventory data outside of BrightWorks](#)

[Removing inventory data for a specific site from the baseline](#)

Removing a Site from the Baseline Inventory

Removing a site's data from the baseline inventory might be necessary if you no longer require an inventory of the site's equipment.

Use the following procedure to remove a site from the baseline. Use this procedure with caution as all equipment inventory records for the selected sites are removed from the baseline. You should retain a backup version of the baseline prior to purging the equipment inventory for certain sites. (The procedures for creating a baseline backup are discussed in [Defining Audit Parameters](#).)

NOTES:

a - The baseline for your Local Site (as identified during the BrightWorks installation) can be purged.

b - The Purge command removes the selected site data from the baseline files only. All site data, except that which was manually entered, remains in the transaction files. Therefore, if you perform an audit after purging the baseline, the data will be restored. (Refer to the next note for instructions on permanently removing the data.)

c - To exclude the site from future audits, you must remove the site from the audit scope and delete the site nodes from inventory (i.e., from the Inventory dialog box).

- 1. Choose the Inventory command from the Administration menu. From the sub-menu which displays, choose the Purge Baseline command.**

The Purge Site from Baseline dialog box displays. All sites defined for your BrightWorks environment are listed.

- 2. Select the site(s) you want to remove from the baseline inventory.**

To select a site for removal, click on the site name. The Number of Sites to Be Purged counter increases as sites are selected.

- 3. Choose the OK button.**

A dialog box displays with a warning message and the prompt "Do You Want to Proceed?" Choose the Yes button to purge the equipment inventory records for the selected sites.

See Also:

[Exporting the baseline inventory data outside of BrightWorks](#)

[Importing baseline-type files into BrightWorks](#)

Collecting Inventory

Equipment inventories are performed using the BrightWorks inventory collection programs. These programs include:

- * **EQUIP.EXE** - used for collecting the inventory on networked PC workstations
- * **SA_EQUIP.EXE** - used for collecting the inventory on stand-alone PC workstations
- * **MACEQUIP** - used for collecting the inventory on networked and stand-alone MAC workstations

NOTE:

File server inventory can be automatically collected during the audit process by selecting the 'Fileserver Software/Hardware' options in the Audit Parameters dialog box.

Automating Inventory Collection

The equipment inventory process can be automated for your networked PC workstations by placing the EQUIP command in the system login script

For networked users, each time the workstation is logged in to the network, the EQUIP inventory collection process is invoked. Depending on the EQUIP command line switches defined in the login script, the workstation user can be notified of the inventory progress as it occurs. The login process can continue after the inventory is complete. A BrightWorks inventory transaction file is updated with the date and hardware or software changes that have occurred since the last time an inventory was performed for the workstation.

NOTE:

The inventory collection process cannot be automated for standalone PCs and MACs. For these configurations, the equipment inventory is stored on diskettes. The collected data must then be added to your network inventory data.

Refer to the following topics for instructions on collecting inventory:

EQUIP Command Line Reference - Describes the command line options available with EQUIP.

Collecting Networked PC Inventory - Describes the procedures for using EQUIP in performing an inventory on networked PCs. Discusses executing EQUIP automatically upon workstation login.

Collecting Stand-alone PC Inventory - Describes the procedures for using SA_EQUIP in performing an inventory on stand-alone PCs. Describes the procedures for adding the stand-alone PC inventory to the BrightWorks baseline inventory.

Collecting Networked MAC Inventory - Describes how to install and configure MACEQUIP, in preparation for taking inventory on Macintosh equipment. Describes the procedures for using MACEQUIP in performing an inventory on networked MACs.

Collecting Stand-alone MAC Inventory - Describes the procedures for using MACEQUIP in performing an inventory on stand-alone MACs. Describes the procedures for adding the stand-alone MAC inventory to the BrightWorks baseline inventory.

EQUIP Command Line Reference

The EQUIP program is used to collect the hardware and software inventory of networked PC workstations. The program is executed at the machine for which you want to perform the inventory.

The EQUIP program's syntax is as follows:

```
EQUIP [drive:\path] /[options]
```

in which *drive:\path* is an optional parameter which indicates the path to the inventory data files and */options* indicates the scan parameters which are always preceded by the slash (/) character. The brackets are not typed. One or more options can be used on the EQUIP command line.

NOTE:

The optional [drive:\path] parameter is used to execute EQUIP from within a "transaction directory." A transaction directory contains the EQUIP program executable and the BrightWorks inventory transaction files. It is used to collect and maintain inventory data for a site other than the local BrightWorks site.

The following command illustrates EQUIP execution:

```
F:\USER\JOE>EQUIP J:\SALES /SHOW /S
```

where *F:\USER\JOE>* is the directory on the file server from which Joe is issuing the EQUIP command; *J:\SALES* is the directory in which the SALES site inventory data is maintained (several BrightWorks inventory files are also located here); */SHOW* is the EQUIP switch which specifies that the collected inventory be displayed on the screen; and */S* is the EQUIP switch which specifies that software inventory be collected.

NOTES:

- a - Because BrightWorks uses the Novell Btrieve Record Manager for database management, BTRIEVE or BREQUEST must be loaded before EQUIP is executed.*
- b - When running the Brequestor, BSPXCOM and BROUTER must also be loaded on the file server. For details on loading these programs, refer to your Novell documentation.*

More than one collection option can be specified on the EQUIP command line. For example, the following command executes EQUIP from the current directory and scans for software, hardware and system file inventory.

```
EQUIP /S /H /Y
```

When the EQUIP command is issued without parameters, the program will default to a hardware and system file scan using the current directory. Further, the collected inventory will not display (i.e., /NOSHOW mode).

Each of the EQUIP command line options is discussed below.

[drive:\path]

The [drive:\path] parameter tells EQUIP which directory to execute from to update the BrightWorks inventory data files maintained in the specified directory. This parameter is optional, and it can be used in any of the following three ways:

- * If no drive letter or path is specified on the EQUIP command line, then the current directory is used.
- * If only a drive letter is used (e.g., F:), then the current directory for that drive is used.
- * If the full path is specified, then the full path is used.

The brackets are not typed.

/NOSHOW

The /NOSHOW option instructs EQUIP not to display the detected inventory on screen as it is collected. This is the default condition when the EQUIP command is issued *without any parameters*.

NOTE:

When the /NOSHOW option is specified, scan parameters must also be provided. For example, the command 'EQUIP /NOSHOW' is invalid.

/SHOW

The /SHOW option instructs EQUIP to display the detected inventory on screen as it is collected.

NOTE:

When the /SHOW option is specified, scan parameters must also be provided. For example, the command 'EQUIP /SHOW' is invalid.

/E

The /E option is used to delete/erase all information about a particular network address/unique identifier from the BrightWorks inventory transaction files. For example:

```
EQUIP /E=2C581ACF:32E1
```

where [node ID] is the identifier of the node to be deleted. Use uppercase for alpha characters. Leading zeros are not required.

NOTES:

a - If the node to be deleted is the same node from which EQUIP is executed, then the [node ID] specification is not required. In this case, simply enter: EQUIP /E.

b - The node ID can be determined by running EQUIP with the /V option to view the hardware inventory. Refer to the /V option discussion below.

If you use the EQUIP /E option and then run an audit, all information about the deleted node will be removed from the equipment files and reported as "missing." If you use the EQUIP /E option and then *delete the node from the Inventory dialog box before running an audit*, the node will be completely removed from the inventory (i.e., not even considered as missing after the next audit).

/S

The /S option instructs EQUIP to scan for all software applications stored on the workstation's local hard drive(s). EQUIP searches for file names that are listed in the PC Software qualification list (i.e., the WPCSLIST.DAT file).

Upon finding a file name that *is listed* in BrightWorks PC Software List, EQUIP verifies the file size. If the detected file size is identical to the file size listed in the PC Software List, then the item is included in the updated Transaction file.

If EQUIP cannot detect a file name that previously had been detected on the node and currently is listed in the PC Software database, it will *not* include the file name in the updated transaction file. The item will be listed as a missing application in the Inventory Log results.

NOTE:

If the 'Workstation' option is checked in BrightWorks Software Options dialog box, then the Unidentified PC/Fileserver Software list also will be updated. In this case, upon finding a file name (or corresponding file size) that is not listed in the PC Software list, the file is added to the Unidentified PC/Fileserver Software list.

/H

The /H option instructs EQUIP to scan for all hardware components and network information. Refer to Appendix C in the BrightWorks manual for a complete listing of hardware inventory data.

NOTE:

EQUIP detects certain items on the basis of the CMOS set-up; therefore, the CMOS set-up on all workstations must be accurate.

/Y

The /Y option instructs EQUIP to scan for the system files that are specified in BrightWorks Software Options dialog box. System files generally include the following:

- * AUTOEXEC.BAT
- * CONFIG.SYS
- * WIN.INI
- * SYSTEM.INI

To detect the AUTOEXEC.BAT and CONFIG.SYS files, EQUIP scans the root directory on the drive from which the workstation was booted. The boot drive is determined in two ways:

- * In DOS 4.0 and above, EQUIP gets the AUTOEXEC.BAT and CONFIG.SYS files from the boot drive.
- * In DOS versions prior to 4.0, EQUIP searches the first hard disk that it finds. If there is no hard disk, EQUIP searches for the system files on the first floppy drive it finds.

NOTE:

To be detected by EQUIP, all system files must be in the workstations' search path.

/V

The /V option instructs EQUIP to scan for hardware and network information for viewing purposes only. The detected inventory information *is not* written to a database and does not require Btrieve to be loaded. This option can only be used with a scan for hardware.

The following example instructs EQUIP to perform a scan for hardware and display the results on screen:

```
EQUIP /H /V
```

NOTE:

The /V option only scans for hardware and network information. When using /V, it is necessary to specify the /H scan parameter.

/D, /W and /M

The /D, /W and /M options are called "frequency switches." These switches refer to running EQUIP on a daily, weekly and monthly basis, respectively.

Each switch instructs EQUIP to execute only after an entire cycle. For example, using the /D option, EQUIP will execute only once every 24 hours. In all other cases (i.e., before 24 hours has passed), EQUIP will display the message "No Action Taken" instead of executing. The /W option causes EQUIP to execute once every 7 days. The /M option causes EQUIP to execute once every month.

Placing the following command in a login script will cause EQUIP to execute monthly and scan for software and hardware:

```
.....
#EQUIP /M /S /H
.....
```

To instruct EQUIP to execute on a particular day, you must use the Novell DAY_OF_WEEK command, as in the following example:

```
.....
IF DAY_OF_WEEK="MONDAY" THEN
  MAP Q:=FS1/SYS:APPS\FUSION
  DRIVE Q:
  #BREQUEST /D:17000
  #EQUIP /H /Y /D
  #ENDBTRV
  MAP DEL Q:
END
.....
```

NOTES:

- a - The EQUIP frequency switches are machine specific the next time EQUIP executes on a machine depends on the last time EQUIP was executed on that machine.*
- b - If the user does not log in to the server during the specified day, week or month, the next time the user logs in, EQUIP determines how much time elapsed since the last login. EQUIP will execute if the elapsed time is greater than the option cycle.*
- c - In order to execute at correct cycles, a login script should have only one EQUIP command with frequency switches. The second EQUIP command will never be executed because of the previous EQUIP command (e.g., the cycle for the second command will never occur).*

/U

The /U option allows the BrightWorks user to pass an alphanumeric string to the inventory transaction files. This option can only be used the first time EQUIP is executed at a workstation. It allows a name other than the login name to be associated with the node.

For example, a Supervisor might enter the following command to log onto Joe's machine and execute EQUIP for the first time:

```
EQUIP /H /Y /S /U=JOE
```

The above command instructs EQUIP to scan for hardware, system files, software and pass the name "Joe" to the inventory transaction files.

Collecting Networked PC Inventory

The EQUIP program is used to collect the inventory of networked PCs. Upon execution, EQUIP scans the hardware for a unique ID in the root directory. This unique ID is used throughout the BrightWorks Inventory component as the workstation's identifying key.

NOTES:

- a - The first time EQUIP is executed at a workstation, EQUIP generates the workstation ID and stores it in a hidden file called LAIID.CFG maintained in the workstation's root directory.*
- b - The LAIID.CFG file is used to identify a workstation on which EQUIP is executed. If the file has been deleted, EQUIP will create a new LAIID.CFG file and consider the workstation to be a new node. As a result, two entries for the same PC will be listed in the Inventory dialog box. Therefore, to avoid confusion, if the LAIID.CFG file has been deleted, also delete the node's entry in the Inventory dialog box before running EQUIP and auditing the workstation inventory.*

The EQUIP command can be placed in the system login script so that inventory collection is performed automatically each time a user logs into the network.

BrightWorks also provides two EQUIP batch files which are placed in the BrightWorks program directory upon installation. The batch files names are listed below:

- * **EQUIP1.BAT** - loads server-based Btrieve (BREQUEST.EXE), executes EQUIP and then unloads Btrieve.
- * **EQUIP2.BAT** - loads local Btrieve (BTRIEVE.EXE), executes EQUIP and then unloads Btrieve.

Both batch files need to be modified to reflect the correct path and drive combination, as well as to specify the desired scan parameters.

Running EQUIP

Use the following procedure to execute EQUIP from a networked PC.

NOTES:

- a - To hide a file from inventory, the file must be hidden before EQUIP is run on a machine which has the file. Once a file is detected by EQUIP, the file is recorded in inventory upon running an audit. If it is subsequently hidden, the file will be considered as "missing" after the next audit is performed. Refer to Modifying Qualification Records for instructions on hiding records in the PC/MAC Software qualification lists.*
- b - Executing EQUIP on several machines that use the same boot disk might cause the machines' inventory to be incorrectly incorporated into the baseline inventory.*
- c - Executing EQUIP on a machine that uses more than one boot disk might cause the machine's inventory to appear more than once in the baseline inventory.*

- 1. From the workstation on which you want to perform an inventory, change into the directory containing the EQUIP executable and BrightWorks database files.**

Use the DOS CD command to change directories.

- 2. Load the Btrieve software.**

Btrieve must be properly configured on the file server from which it is run. Btrieve must also be loaded before EQUIP is executed, and it should be unloaded after EQUIP has run. Due to the resulting increase in performance, it is highly recommended that you use Btrieve's serverbased program (BREQUEST.EXE).

To load Brequest, use the BrightWorks BRQ.BAT file which consists of the following command:

```
BREQUEST /D:17000
```

To load local Btrieve, use the BrightWorks BTR.BAT file which consists of the following command:

```
BTRIEVE /P:3072 /F:22 /T:BTR.TRN /E
```

3. Execute EQUIP.

For example, to scan for the hardware, software and system file inventory, type:

```
EQUIP /H /S /Y <ENTER>
```

The inventory is collected and placed in the appropriate BrightWorks inventory transaction files.

4. Unload the Btrieve software.

To unload the Btrieve software, type:

```
ENDBTRV <ENTER>
```

NOTES:

a - To enable EQUIP to run under NetWare 2.1x, users need Read, Open, Write, Search, Create, and Delete rights to the directory where BrightWorks files are stored. NetWare 3.x or 2.2 users need Read, Write, Filescan, Create, and Erase rights.

b - Mark the .EXE files as "shareable" and make sure the COMSPEC is set properly.

See Also:

[Running an Audit](#)

Collecting Stand-alone PC Inventory

The SA_EQUIP program is used to collect the inventory of stand-alone PCs. The collected data is placed onto "collector diskettes" which are prepared expressly for this purpose.

As with EQUIP, SA_EQUIP scans the hardware for a unique ID in the root directory. This unique ID is used throughout the inventory as the stand-alone machine's identifying key.

NOTE:

The first time SA_EQUIP is executed at a stand-alone machine, SA_EQUIP generates the machine's ID and stores it in a hidden file called LAIID.CFG maintained in the root directory.

The steps for including stand-alone PC data in your BrightWorks baseline inventory include the following:

1. Creating a collector diskette
2. Running SA_EQUIP at the stand-alone PC
3. Loading the stand-alone data into BrightWorks
4. Running an audit to include the stand-alone data in the baseline inventory

Creating a Collector Diskette

Stand-alone PC inventory data collected by SA_EQUIP is placed onto specially prepared diskettes containing the BrightWorks equipment inventory system files. All inventory for the stand-alone PCs must be placed onto a collector diskette. If the collected inventory data requires more than one diskette, additional collector diskettes must be used.

Use the following procedure to create a collector diskette.

1. Insert a blank formatted diskette into the diskette drive.

Approximately 300K of disk space is required for the collector diskette files.

2. Choose the Inventory command from the Administration menu. From the sub-menu which displays, choose the Create Collector Diskette command.

The Create Collector Diskette dialog box displays.

3. In the Drives list, choose the drive into which you inserted the diskette.

The selected drive letter displays in the Path field. Any directories on the selected drive display in the Directories list.

4. If applicable, select the directory into which you want the equipment inventory files placed.

The directory and subdirectory (as applicable) display in the Path field.

5. Choose the OK button to create the collector diskette.

The SA_EQUIP executable and BrightWorks equipment inventory files are copied to the specified drive and directory path. As the files are copied, the copy progress displays.

NOTES:

a - All collector diskettes must be created using the above procedure. Collector diskettes created in any other manner (e.g., DOS DISKCOPY) will not work correctly.

b - All collector diskettes are associated with the main/local BrightWorks directory site name; therefore, all stand-alone data will be associated with the main BrightWorks inventory site.

See Also:

Running SA_EQUIP

Loading the Stand-alone Data

Running an Audit

Running SA_EQUIP

Use the following procedure to collect the equipment inventory of a stand-alone PC. You must use a prepared collector diskette to which the inventory data will be copied.

NOTES:

- a - Executing SA_EQUIP on several machines that use the same boot disk might cause the machines' inventory to be incorrectly incorporated into the baseline inventory.*
- b - Executing SA_EQUIP on a machine that uses more than one boot disk might cause the machine's inventory to appear more than once in the baseline inventory.*

- 1. Insert the collector diskette into the diskette drive of the stand-alone PC.**
- 2. Execute SA_EQUIP from the floppy diskette.**

Issue the SA_EQUIP command by typing:

```
SA_EQUIP <ENTER>
```

If applicable, use the DOS CD command to change into the directory containing the SA_EQUIP executable and BrightWorks inventory files.

The BrightWorks inventory files are activated and the Workstation Information screen displays.

NOTE:

Upon executing SA_EQUIP, if there is insufficient space available on the diskette, SA_EQUIP displays the message "Not enough disk space. Use a new collector disk."

- 3. If desired, modify the workstation information.**

The values of the following fields can be modified:

- * Location - the workstation location (e.g., First Floor)
- * Department - the department to which the workstation belongs (e.g., Sales)
- * User - the user name/ID assigned to the workstation
- * Asset Tag - the tag or identifying number assigned to the machine to be used as another ID for the workstation

Use the arrow keys to move the cursor into a field, and overwrite the existing entry with your new data.

- 4. Press the <ENTER> key to save the workstation information and continue.**

The machine's hardware data is automatically collected, and then the SA_EQUIP menu bar displays across the top of your screen.

Use the <left/right arrow> keys to highlight a desired menu. Press <ENTER> to choose the highlighted menu and display its drop down list of commands. Use the <up/down arrow> keys to highlight a desired command. Press <ENTER> to choose the highlighted command.

The available SA_EQUIP menus and their commands are as follows:

* File Menu

- **System Files** -command used to collect the system files that are specified in BrightWorks Software Options dialog box.
- **Exit** -command used to save the collected data to files on the collector diskette and exit the

SA_EQUIP program.

* **Hardware Menu**

- **Edit Hardware** - command used to modify hardware details about the PC (e.g., Model Number, Serial Number, and Component/Asset Tag).

* **Software Menu**

- **Scan Software** - command used to inventory the PC software. When chosen, a status bar displays showing the progress of the software inventory.

5. To edit the PC hardware inventory, choose the Edit Hardware command from the Hardware menu.

The Hardware Components screen displays showing the PC hardware inventory that is compiled automatically upon executing SA_EQUIP.

Values for the Model Number, Serial Number and Component (Asset) Tag fields can be specified for any of the listed hardware items. To do so, use the <up/down arrow> keys to highlight the desired item and press <ENTER>. When the Edit Hardware screen displays, enter or overwrite any of the three detail fields.

Press <ENTER> two times to save any changes and close the Edit Hardware screen. Press <ESC> to exit without saving your changes.

Press <ESC> to close the Hardware Components screen.

6. To inventory the PC software, choose the Scan Software command from the Software menu.

The software inventory is immediately performed. The inventoried software data cannot be edited.

7. To inventory the workstation's system files, choose the System Files command from the File menu.

All system files listed in the BrightWorks Software Options dialog box (accessible from the Inventory command on the Administration menu) are collected. The system file inventory is immediately performed. The inventoried system file data cannot be edited.

8. To save the inventory data and exit SA_EQUIP, choose the Exit command from the File menu and choose any key to save the configuration to the collector diskette.

The inventory data is copied onto the collector diskette. When all inventory data is copied to the diskette, SA_EQUIP is automatically exited. Remove the collector diskette from the PC. Repeat the procedure for additional stand-alone PCs.

The collected data must now be loaded into BrightWorks in order for the inventory to be included in the next audit.

NOTES:

a - If the collector diskette becomes full when saving the data, the message "Disk out of space. Do collection again" displays. In this case, exit the program and then re-execute SA_EQUIP using a new collector diskette. You must collect the data again.

b - If one collector diskette is used to inventory multiple stand-alone PCs, the information for each additional workstation is appended to the appropriate file.

See Also:

Creating a Collector Diskette

Loading the Stand-alone Data

Running an Audit

Loading Stand-alone Data into BrightWorks

After the stand-alone PC inventory has been gathered using SA_EQUIP, the data must be loaded into BrightWorks in order for the next audit to include the new inventory.

Using the BrightWorks console, use the following procedure to load the standalone PC inventory data into BrightWorks.

1. **Insert the collector diskette into the diskette drive.**
2. **Choose the Inventory command from the Administration menu. From the sub-menu, choose the Load Equipment from Collector Diskette command.**

The Load Equipment From dialog box displays.

3. **Select the drive and directory containing the stand-alone inventory data.**
4. **Choose the OK button to load the stand-alone PC equipment inventory into BrightWorks.**

A status screen displays the progress as the stand-alone inventory data is copied from the diskette to the equipment inventory files maintained in the BrightWorks program directory.

NOTE:

To incorporate the stand-alone data into the BrightWorks baseline inventory, the audit parameters must be defined to include PC hardware and software before performing the audit.

See Also:

[Creating a Collector Diskette](#)

[Running SA_EQUIP](#)

[Running an Audit](#)

Collecting MAC Inventory

MACEQUIP is used to collect the inventory of Macintosh computers attached to Novell file servers that are running the MAC VAP/NLM. MACEQUIP is also used to collect the inventory of stand-alone MACs.

BrightWorks provides two programs for collecting MAC inventory. Both programs are used to collect inventory information on stand-alone and networked Macintosh computers; however, their execution is slightly different:

- * **MACEQUIP 3.1** - Performs inventory collection and provides a menu from which you can choose to view the collected inventory and/or close the program.
- * **MACEQUIP AUTOMATIC 3.1** - Automatically closes the program when inventory collection is complete.

NOTE:

The term "MACEQUIP" is used to refer to both MAC programs. When following the procedures, use either the MACEQUIP 3.1 program or the MACEQUIP AUTOMATIC 3.1 program.

Upon execution, MACEQUIP automatically scans the machine's hardware components and the software applications on its hard drive. The hardware and software data is stored in the MACEQUIP.CFG and MACSOFT.LST files, respectively. The collected data is added to the BrightWorks baseline inventory by running an audit.

For each workstation on which MACEQUIP is executed, a unique ID is created using the Ethernet address. If there is no AppleTalk, the user name is converted to a numeric ID. The network number will be 0000 for all Macintosh's.

Refer to the following topics for instructions on using MACEQUIP.

[Installing MACEQUIP](#)

[Automating the Execution of MACEQUIP](#)

[Collecting Networked Macintosh Inventory](#)

[Collecting Stand-alone Macintosh Inventory](#)

Installing MACEQUIP

The following must be in place before continuing with the MACEQUIP installation:

- * BrightWorks must be properly installed according to the instructions outlined in Chapter 2 of the BrightWorks manual.
- * Your network file server must support the AppleTalk Filing Protocol (AFP).
- * Your network must be running one of the following: NetWare 286 2.15c or 2.2 with the Macintosh VAP; NetWare 386 3.11, 3.12 or 4.01 with the Macintosh NLM; AppleTalk 2.15c or 2.2 Macintosh VAPs.

Use the following procedure to install MACEQUIP. Because MACEQUIP is a Macintosh file, you MUST follow these steps to properly install the program on your network. MACEQUIP should be copied to the same directory where EQUIP and the other BrightWorks inventory files reside.

1. **At a Macintosh workstation, insert the floppy disk containing the MACEQUIP file into the disk drive.**
2. **Choose the Chooser command from the Apple menu.**
3. **Select Apple Share.**
4. **Select the file server on which MACEQUIP should be installed.**
5. **Mount the selected file server.**
6. **Close the Chooser.**

An icon is created that reflects the name and volume of the file server. For example:

MYSERVER.SYS

7. **Double click on the File Server icon.**
8. **Find the BrightWorks folder.**
9. **Open the floppy disk icon.**
10. **Drag the file icon for MACEQUIP from the floppy disk to the BrightWorks directory on the file server.**

This completes the MACEQUIP installation process. The MACEQUIP program can now be executed from your network's Macintosh workstations.

See Also:

[Automating the Execution of MACEQUIP](#)
[Collecting Networked Macintosh Inventory](#)
[Collecting Stand-alone Macintosh Inventory](#)

Automating MACEQUIP Execution

With System 7.0 or greater, the MACEQUIP program can be a Start Up item and run automatically when the Macintosh starts up. Use the following procedure to set up the automatic equipment inventory.

1. From the Macintosh, choose the Chooser command from the Apple menu.
2. Select Apple Share.
3. Select the file server on which MACEQUIP resides.
4. Mount the selected file server.
5. Close the Chooser.
6. Double click on the File Server icon.
7. Open the folder where MACEQUIP is located and select MACEQUIP.
8. Select the Make Alias command from the File menu to create an alias for MACEQUIP.
9. Drag the alias under MACEQUIP into the Start Up Items folder in the System folder.

Every time the machine is rebooted, the volume will be mounted and MACEQUIP will run automatically.

NOTE:

If you do not have SYSTEM 7.0 or greater, every time you want to use MACEQUIP, you must go through steps 17 above and then run MACEQUIP by double clicking on the MACEQUIP icon.

See Also:

[Installing MACEQUIP](#)

[Collecting Networked Macintosh Inventory](#)

[Collecting Stand-alone Macintosh Inventory](#)

Collecting Networked Macintosh Inventory

Use the following procedure to manually execute MACEQUIP from a networked Macintosh workstation.

1. **From the workstation on which you want to perform an inventory, change into the directory containing the MACEQUIP executable and BrightWorks database files.**
2. **Launch MACEQUIP.**

The inventory scan is performed, and the MACEQUIP.CFG and MACSOFT.LST files are updated to reflect the collected hardware and software inventory, respectively.

See Also:

[Installing MACEQUIP](#)

[Automating the Execution of MACEQUIP](#)

[Collecting Stand-alone Macintosh Inventory](#)

Collecting Stand-alone Macintosh Inventory

MACEQUIP is also used to collect the hardware and software data from MACs that are not connected to the network. The collected data is placed onto a diskette and later added to the BrightWorks inventory transaction files to be included in the next audit.

The steps for including stand-alone MAC data in your BrightWorks baseline inventory include the following:

1. Running MACEQUIP at the stand-alone MAC
2. Updating the BrightWorks MAC transaction files to reflect the stand-alone data
3. Running an audit to include the stand-alone data in the baseline inventory

NOTES:

a - The term "MACEQUIP" is used to refer to both MAC programs. When following the procedures, use either the MACEQUIP 3.1 program or the MACEQUIP AUTOMATIC 3.1 program.

b - If you collect and audit the inventory of a stand-alone MAC and then later put the same MAC on the network and do a subsequent inventory and audit, two entries for the same MAC will be listed in the Inventory dialog box. Therefore, to avoid confusion, before running MACEQUIP and auditing the networked MAC inventory, first delete the node's entry in the Inventory dialog box.

Use the following procedure to collect the equipment inventory of a stand-alone MAC.

1. **Copy the MACEQUIP file onto a Macintosh formatted floppy diskette.**
2. **Insert the floppy diskette into the stand-alone MAC.**
3. **Execute MACEQUIP from the floppy diskette.**

The inventory is collected.

4. **Choose the Quit command from the File menu.**

The inventory is complete. Repeat the procedure for additional stand-alone MACs, and use the same floppy diskette.

The BrightWorks MAC inventory transaction files must now be updated in order for the next audit to include the new inventory.

Updating MAC Transaction Files

As a result of taking inventory on a stand-alone MAC, two DOSbased files are created on the Macintoshformatted diskette:

- * MACEQUIP.CFG - contains the hardware inventory data
- * MACSOFT.LST - contains the software inventory data

To include the stand-alone MAC inventory in the next audit, copy the MACEQUIP.CFG and MACSOFT.LST files into the BrightWorks program directory on the network.

If the MACEQUIP.CFG and MACSOFT.LST files residing in the BrightWorks program directory already contain data from other networked or stand-alone MACs, you must append the new information to the

existing files in the BrightWorks program directory. To append the new data, first get the new data files (MACEQUIP.CFG and MACSOFT.LST) into a DOS environment (i.e., a Novell Volume [or directory], a DOS floppy disk, or a DOS hard disk). Then append the data by using the DOS "TYPE" command on the new data files and redirecting the output of the TYPE command to the existing MACEQUIP.CFG and MACSOFT.LST files that reside in the BrightWorks program directory.

The TYPE command syntax is as follows:

```
TYPE [drive:][path..]source file >> [drive:][path..]target file
```

For example, if the new data files are residing on Drive B: and the BrightWorks program directory is currently mapped to Drive K:, the commands to append both MACEQUIP.CFG and MACSOFT.LST would be executed as follows:

```
B:\>TYPE macequip.cfg >> k:macequip.cfg  
B:\>TYPE macsoft.lst >> k:macsoft.lst
```

In this example, Drive B: is the current drive. Refer to your DOS documentation for more information on the TYPE command.

NOTE:

To incorporate the stand-alone data into the BrightWorks baseline inventory, the MAC Hardware and Software options must be checked in the Audit Parameters dialog box before performing the audit. Also, the Macintosh Unknown Files to Identify option must be checked in the Software Options dialog box.

See Also:

Installing MACEQUIP

Automating the Execution of MACEQUIP

Collecting Networked Macintosh Inventory

Conducting an Audit

Conducting an audit compiles the collected equipment inventory data from your workstations and file servers and adds it to BrightWorks baseline inventory. As a result of performing an audit, the baseline inventory file is updated and represents an accumulation of your most currently audited inventory information. The baseline inventory file provides data for various BrightWorks inventory reports, on-screen viewing, automatic notification of inventory changes, and the BrightWorks software distribution capability.

Audits can be run at any time. However, the audit results depend on the settings defined via the Administration Inventory commands on the BrightWorks menu bar. Audit settings include: audit scope, audit parameters, software options and notification options. These settings need to be established before the audit is conducted. (For details on these procedures, refer to [Audit Set-up and Administration.](#))

Access to Audit Functions

Audit configuration is performed by choosing the Inventory command from the Administration menu. From the sub-menu which displays, choose the desired command.

An audit is initiated by choosing the Inventory command from the Administration menu. From the sub-menu which displays, choose the Audit command.

See Also:

[Auditing Checklist](#)

[Running an Audit](#)

[Reviewing Audit Results](#)

Auditing Checklist

Conducting an audit transfers the inventory information from the transactions database to the BrightWorks baseline database. For example, when a workstation inventory is performed, the collected inventory data gets stored in a transactions database. When an audit is conducted, the data in the transactions database gets compared to the data in the baseline database. In general, all new inventory data gets added to the baseline; all changed data gets updated in the baseline and any missing data gets removed from the baseline.

Several BrightWorks options enable you to customize the audit process to accommodate your specific auditing needs. This topic briefly lists the items you should consider before conducting an audit.

NOTE:

Many of the details on these items are provided in the topic entitled [Audit Set-up and Administration](#).

Time Considerations

Auditing a large network can be a time-consuming process. Eliminate wasted time by defining the scope of the audit to include only the specific sites, file servers and components you are interested in auditing. For example, if you just received new transaction files from your Chicago office, it may not be necessary to include your New York office data in the audit. Simply remove the New York site from your scope specification and limit the audit scope to include only Chicago. In addition to specific site audits, you can conduct exclusive audits for software, hardware, file servers and/or workstations.

What to Audit

An audit can include networked PC data, file server data, and even stand-alone PC and Macintosh inventory data. "Inventory data" can be further defined to include hardware and/or software. BrightWorks provides easy-to-use and intuitive programs to collect the inventory on networked and stand-alone workstations.

Scheduling

BrightWorks lets you schedule audits. You can specify the specific hour, day, week, or month that audits be conducted. You may want an audit to be performed each night at midnight, or you may want the audit to be performed on the fifteenth day of each month. BrightWorks takes care of logging in to the appropriate servers at the correct times to avoid any potential security violations. In addition, the results of an audit are always stored in the audit log for later review.

Multiple Sites

BrightWorks allows you to manage inventory for multiple sites. If you are consolidating information from multiple sites, have you created directories for each site and copied the inventory transaction database files into those directories from the respective sites?

Alerting Options

Do you want to be notified about the outcome of the audit? BrightWorks alerting parameters let you define the conditions under which you want to be notified, as well as the method of notification. For example, you might want to be notified of all hardware changes that have been made to the file servers since the last audit. Further, you might want these results to be sent to you via cc:Mail.

See Also:

[Running an Audit](#)

[Reviewing Audit Results](#)

Running an Audit

The process of running an audit uses the recently collected equipment inventory transaction information to update each existing individual equipment inventory record. In turn, the baseline inventory is updated. As a result of comparing the new current transactions to the baseline, audit statistics are generated. Reported changes in inventory include components, nodes or software that have been added, changed, or removed since the last audit. Audit results can be reviewed on screen or via hard copy report.

Audits can be initiated at any time, regardless of whether automatic audits (e.g., daily, weekly) have been scheduled in the audit parameters.

Use the following procedure to initiate a BrightWorks audit and update the baseline inventory with current equipment inventory data.

NOTES:

a - Before you initiate the audit, make sure that all audit parameters are defined correctly and any stand-alone inventory data has been loaded into BrightWorks.

b - Use the Btrieve NLM to improve the performance of data collection, auditing and reporting by as much as 500%. Refer to [Using Brequest](#) for instructions on configuring BREQUEST and the Btrieve NLM.

- 1. Choose the Inventory command from the Administration menu. From the sub-menu which displays, choose the Audit command.**

A dialog box displays with a warning message and the prompt "Do You Want To Proceed?"

- 2. Choose the OK button to proceed with the audit.**

An Audit in Progress dialog box displays a status bar showing the progress of the audit. When the audit is complete, the baseline is updated to reflect the new inventory data.

See Also:

[Auditing Checklist](#)

[Reviewing Audit Results](#)

Reviewing Audit Results

When an audit has been completed, the audit results are available via the Audit Log dialog box. The Audit Log dialog box provides both summary and detail information regarding the changes made to the baseline inventory as a result of the audit. The software applications that were detected upon inventory collection but not recorded in the inventory database are listed and reported in the Unidentified Software dialog box.

Access to Audit Results

Audit results can be reviewed and/or managed by choosing the Inventory command from the Administration menu. From the sub-menu which displays, choose the View Audit Log or Unidentified PC/Fileserver or Macintosh Software commands.

Refer to the following topics for instructions on reviewing audit results.

[Viewing the Audit Log](#)

[Managing Unidentified Software](#)

[Transferring Unidentified Software](#)

Viewing the Audit Log

The Audit Log provides both summary and detail information about the changes that were made to the baseline inventory as a result of an audit. The information available from the Audit Log includes:

*** Summarized Changes:**

- the date and time of the audit
- the name of the person who initiated the audit
- the number of changes made to inventory categories (e.g., 5 nodes added; 1 system file updated)

*** Detailed Changes:**

- an itemized list of components, nodes and system files that have changed since the last audit (e.g., at the site named SALES, a new node JOHN was found)

Audit summary and detail information is maintained by BrightWorks for as long as necessary. For example, your organization might require that the audit results be accessible on-line for a year. In this case, each audit performed within the last year is listed in the Audit Log dialog box, and the results of each audit can be viewed.

The only modification that can be made to the Audit Log is the deletion of an audit entry. Continuing with the above example, all audits that occurred more than a year ago would need to be deleted from the Audit Log.

Use the following procedure to view and manage on-line audit results.

- 1. Choose the Inventory command from the Administration menu. From the sub-menu which displays, choose the View Audit Log command.**

The Audit Log dialog box displays. The dialog box lists all the audits that have been performed. The Date, Time, Performed By, and Status information is listed for each audit.

- 2. To view the summarized results of an audit, select an audit in the Audit Log dialog box and choose the Results button.**

The Audit Results dialog box displays. This dialog box summarizes the changes made to inventory categories as a result of the audit.

NOTE:

The Audit Results dialog box can also be displayed by double clicking on an audit in the Audit Log dialog box.

To view detailed information about a category that has an entry greater than zero (0), double click on the category name. For example, if two system files are added as a result of the audit, you can double click on the System Files Added category to view the names of the added system files.

Choose the Close button to close the Audit Details dialog box. Choose the Close button to close the Audit Results dialog box.

- 3. To view the audit details, select an audit from the Audit Log dialog box and choose the Details button.**

An Audit Details dialog box displays which contains a itemized list of audited components, nodes, and system files.

Use the scroll buttons and scroll bars to view all the information. Choose the Close button to close the Audit Details dialog box.

NOTE:

Procedures for viewing the Audit Details dialog box are also provided in Step #2; however, the dialog box accessed in Step #2 only shows the details of a selected component. By choosing the Details button in the Audit Log dialog box, all component changes for an audit are displayed in the audit log details dialog box.

- 4. To remove an audit entry from the Audit Log dialog box list, highlight an audit and choose the Delete button.**

You are prompted to confirm the deletion. Choose the Yes button to continue and remove the selected audit from the list.

- 5. To print the results of an audit entry in the Audit Log dialog box list, highlight an audit and choose the Print button.**

The details of the selected audit are sent to the printer currently defined in your print set-up.

- 6. Choose the Close button to exit the Audit Log dialog box.**

Unidentified Software

"Unidentified software" is the group of software files that were detected during an inventory but were not listed in the PC or MAC Software qualification lists. As a result, BrightWorks is not able to categorize and identify the software during the audit. For example, assume that an inventory performed on PC workstation X results in detecting the file named ABC.EXE. If file ABC.EXE is not listed in the PC Software database when an audit is performed, then the file is labeled as "unidentified" and added to the Unidentified Software list.

NOTE:

Unidentified software is only collected if the "Unknown Files to Identify on Local Site" options for Fileserver, Workstation and/or Macintosh are checked in the Software Options dialog box. If none of these options are checked, then only the applications listed in the PC or MAC Software Lists are included in the inventory and audit.

In addition to viewing the list of unidentified software, you can "identify" the files by transferring them to their appropriate database. In the above example, file ABC.EXE can be transferred to the PC Software qualification list. By doing so, the next time an audit is performed or inventory is collected, the file will no longer be tagged as "unidentified."

Viewing and Managing Unidentified Software

Use the following procedure to view, sort, print and delete items from the list of unidentified software.

1. **Choose the Inventory command from the Administration menu. From the sub-menu which displays, choose either the Unidentified PC/Fileserver Software command or the Unidentified Macintosh Software command.**

The last command you select depends on the workstation type for which you want to view unidentified software.

The Unidentified Software dialog box displays. Use the line up/line down and page up/page down scroll buttons to scroll through the list of file names.

The following information is provided:

- * File Path
- * File Name (or File Creator for Macintosh software)
- * File Size

NOTE:

The dialog boxes displayed for both PC/Fileserver and Macintosh information are very similar. A File Name field is listed for PC applications; whereas, a File Creator field is listed for Macintosh applications.

2. **To sort the Unidentified Software list, choose the Sort button.**

The Sort Options dialog box displays allowing you to select the sort method. The items in the Unidentified Software list can be sorted by either File Path or File Name/File Creator.

Select the appropriate sort method, and choose the OK button. The list is sorted to reflect the selected method.

3. **To print the Unidentified Software list, choose the Print button.**

The entire list of Unidentified Software is sent to the printer as defined by your current print parameters.

- 4. To delete an item from the Unidentified Software list, highlight the item name and choose the Delete button.**

The highlighted file name is removed from the list. Multiple files can be selected and simultaneously deleted from the list. To select more than one file name, simply click on each desired file name.

- 5. To deselect any highlighted files, choose the Unmark button.**

The highlight is removed from all files that were previously selected.

- 6. To close the Unidentified Software list, choose the Close button.**

All changes made to the Unidentified Software list are saved.

See Also:

[Transferring Unidentified Software](#)

Transferring Unidentified Software

All files listed as unidentified software can be transferred to their appropriate database (i.e., PC or MAC Software list) for inclusion in subsequent audits. Once transferred, the file(s) are removed from the Unidentified Software list and added to the target database.

Many applications require more than one executable file to run the software. To accommodate this requirement and reduce the number of files listed in the software databases, BrightWorks lets you mark a particular file as the key or identifying component of a software product. Therefore, files can be transferred from the Unidentified Software list individually or in groups.

Use the following procedure to transfer unidentified software.

1. **Choose the Inventory command from the Administration menu. From the sub-menu which displays, choose either the Unidentified PC/Fileserver Software command or the Unidentified Macintosh Software command.**

The last command you select depends on the workstation type for which you want to view unidentified software. The Unidentified Software dialog box displays.

The items in the Unidentified Software list can be sorted by either file path/file creator or file name. To change the method by which the items are sorted, choose the Sort button. The Sort Options dialog box displays. Select the appropriate sort method, and choose the OK button. The list is sorted to reflect the selected method.

2. **Select an individual file name or a group of files to be transferred.**

Multiple files can be selected by clicking on more than one file name.

Choose the line up/line down and page up/page down scroll buttons to scroll through the list of file names.

3. **To transfer a single file, choose the Transfer Individually button.**

The Transfer Unidentified Software dialog box displays.

Depending on the type of unidentified software being viewed (i.e., either PC or MAC), the corresponding Software list displays on the bottom half of the dialog box for reference purposes.

- a - **Specify details of the file to be transferred.** The File Name and File Size fields are automatically filled in; however, the information can be edited. The following information must be supplied:

- * **Product Name** - enter the name which identifies the software product to which the file belongs.

NOTE:

If you want to add the single unidentified file to a group of files already defined in the PC or MAC Software List, then add a colon to the Product Name followed by the file name. For example, if you want to add the ABC.EXE unidentified file to the ABC product group, then enter 'ABC:ABC.EXE' in the Product Name field. By doing this, ABC.EXE will be listed with all other ABC product files in the software list.

- * **Version Number** - enter the version number of the software product to which the file belongs.
- * **Category** - from the drop down list associated with this field, select the category of the software product to which the file belongs.
- * **Manufacturer** - from the drop-down list associated with this field, select the manufacturer of the software product to which the file belongs. (If you enter a new manufacturer name, the

name is added to the Manufacturers qualification list.)

- * **Hide in Inventory** - place a checkmark in this field to hide the file from inventory display. (Hidden files are listed in the qualification list, but cannot be viewed in an inventory list.)

b - Choose the Transfer button.

The selected file is transferred to the target database.

4. To transfer a group of files, choose the Transfer As Group button.

The Transfer Unidentified Software as a Group of Applications dialog box displays.

a - Specify details of the group to be transferred. The following information must be supplied:

- * **Group Base Name** - enter the name which identifies the software product to which the group of files belong.
- * **Group Version** - enter the version number of the software product to which the group of files belong.
- * **Group Manufacturer** - from the drop-down list associated with this field, select the manufacturer of the software product to which the group of files belong. (If you enter a new manufacturer name, the name is added to the Manufacturers qualification list.)
- * **Category** - from the drop-down list associated with this field, select the software category in which the group of files belong (e.g., word processing, games). If you enter a new Category name, the name is added to the Software Categories qualification list.
- * **Inventory Options** - select one inventory option:
 - Hide in Inventory - hide the entire set of files from inventory
 - Inventory All Members - identify all files as separate components to be inventoried
 - Inventory Identifier Only - inventory only the package identifier
- * **Group Identifier** - the drop-down list associated with this field lists the file names selected for transfer. Select one file that should be used to identify the group of files.

NOTE:

A group of unidentified files cannot be added to another group of files already defined in the PC or MAC Software List. Files can only be added to groups individually. Refer to step #3 above for instructions on transferring each file in the group individually.

b - Choose the OK button to begin the group transfer.

The selected files are transferred to the target database.

5. When all files have been transferred, choose the Close button in the Unidentified Software list dialog box.

NOTE:

To include all transferred PC files in your inventory, you must subsequently run EQUIP on all workstations and then perform an audit. To include all transferred MAC files in your inventory, you must subsequently perform an audit.

See Also:

Unidentified Software

Maintaining the Baseline Inventory

The baseline inventory is the equipment inventory database that is updated each time an audit is performed. BrightWorks inventory collection programs (EQUIP, SA_EQUIP, MACEQUIP) place the collected hardware and software data in "transaction files." The audit process compiles the transaction files and compares the most recently collected equipment inventory information against the current baseline. As a result, a new updated baseline inventory database is created.

The baseline is made up of equipment inventory records which contain data specific to each type of equipment (e.g., file server, workstation). The equipment inventory records maintain data such as manufacturer, network address, memory, software applications, and storage. These records provide a basis for BrightWorks inventory reports and on-screen viewing. As a result, you might need to verify and update the equipment inventory information, as well as add supporting information to the component data.

Access to Equipment Database Information

Inventory equipment database information is accessible by choosing the Inventory tool bar button or by choosing the Inventory command from the Tools menu.

Refer to the following topics for instructions on maintaining the baseline inventory.

[Viewing Equipment Inventory Data](#)

[Managing Equipment Inventory Data](#)

[Maintaining Component Information](#)

Viewing Equipment Inventory Data

Equipment inventory records for each workstation, file server and stand-alone machine are created or updated automatically during an audit to reflect the most current hardware and software inventory information.

In addition to simply viewing the inventory data, record maintenance is sometimes necessary. For example, the floppy disk information collected by the EQUIP program is determined by the machine's CMOS set-up. If the CMOS set-up is incorrect, then the collected information regarding the machine's floppy disk(s) will not be accurate. To avoid having to run another inventory (after correcting the CMOS set-up), equipment inventory data can be corrected on-line.

Two primary windows are associated with equipment inventory records:

- * **Inventory Window** a summary screen listing each record (i.e., node) in the Equipment Inventory database.
- * **Site: Detail Equipment Inventory Record** displays equipmentspecific information for a selected node. Information includes computer data, mass storage, keyboard and port data, network adapter information and memory. Subdialog boxes provide software details, system files or miscellaneous equipment information for the selected record.

Viewing Equipment Inventory Data

Use the following procedure to display the Inventory dialog box which provides a summary of each record in the Equipment Inventory database.

1. Choose the Inventory tool bar button, or choose the Inventory command from the Tools menu.

The Inventory dialog box displays. This is a summary dialog box that displays a list of the workstations and file servers for each audited site.

If necessary, choose the line up/line down and page up/page down scroll buttons to scroll through the list of records.

2. To change the sort method of the records, choose the Sort button.

The records are initially listed in alphabetical Site order; however, you have the option of resorting the records.

Upon choosing the Sort button, the View Inventory Sorting Options dialog box displays. The Current Sort Order is displayed at the top of this dialog box. Up to two fields can be selected to define the order in which you want the records sorted. Choose the OK button to re-sort the inventory records according to the selected method.

3. To narrow the list of records displayed, choose the Filter button.

The Filter Options dialog box displays. Select one or more filter criteria fields, and choose the OK button. Only the inventory records that match the filter criteria will display in the Inventory dialog box.

4. To view the details of a record, highlight the desired record and choose the Edit button.

You can also double click on the record to view the record details. The detail dialog box displays.

The title bar of the dialog box indicates the last time the record's inventory data was updated.

The fields in the detail dialog box differ depending on the record type (e.g., file server, MAC, PC workstation, and spare part).

Choose the horizontal and vertical scroll buttons in any of fields to scroll through the information.

5. To close the Inventory dialog box, choose the OK button.

See Also:

Managing Equipment Inventory Records

Managing Equipment Inventory Records

From the Inventory dialog box, new equipment inventory records can be added, existing records can be removed, and/or a record's detail information can be modified. (The Inventory dialog box is displayed by choosing the Inventory tool bar button or by choosing the Inventory command from the Tools menu.)

Use the following procedure for managing records in the Equipment Inventory database.

1. To add a new equipment inventory record to the Inventory dialog box, choose the Add button.

The Add New Equipment dialog box displays allowing you to select field values from dropdown lists to define a new record to be included in future audits. (The Asset Tag field does not have an associated drop-down list and must be manually entered.)

Specify the field values for the new record, and choose the OK button. The new record is saved and added to the Inventory list. (Inventory data for the new record is automatically added by BrightWorks after you perform an inventory on the machine and then run an audit.)

2. To remove an equipment inventory record from the Inventory dialog box, highlight the desired record and choose the Delete button.

A dialog box displays with the prompt "Do you really want to delete this item?" Choose the Yes button to delete the record, or choose No to exit without removing the record.

3. To edit the details of an equipment inventory record, highlight the desired record and choose the Edit button.

You can also double click on the record in the Inventory dialog box to view the record details. The detail dialog box displays.

The following actions can be performed in the detail dialog box:

a - To cycle through the detailed equipment inventory records listed in the Inventory dialog box, choose the Previous and Next buttons. For example, choosing the Next button will display the detailed inventory for the next record/node listed in the Inventory dialog box.

b - To review the status of equipment inventories taken on the record shown in the detail dialog box, choose the Notes button. The Notes dialog box displays showing the last date on which an audit was performed which included this workstation, file server or MAC. Type any additional notes, and choose the OK button to return to the detail dialog box.

c - To print the displayed equipment inventory record, choose the Print button. A popup dialog box displays a "printing" message while the record is being printed.

d - To review the contents of the system files applicable to the record shown in the detail dialog box, choose the System Files button. (This button is not available for file servers and MACs.) The System Files Available list displays. To review the contents of a system file, highlight the desired file and choose the View button. Note that editing cannot be performed from this dialog box. Choose the Close button two times to return to the detail dialog box.

e - To add a component to the record shown in the detail dialog box, choose the Add button. The Components dialog box displays. To display specific component information, select a component Category and Class from the drop down lists in the lower section of the Components dialog box. The data which matches the selected Category and Class is displayed in the Components dialog box. To add a component to the record shown in the detail dialog box, highlight an item and choose the Select button.

To add new component data to the BrightWorks inventory database, choose the Add button. The Add Component dialog box displays. Enter the information for the new component. All entries except for Product Name can be selected from drop-down lists. Choose the OK button to add

the new component information to the Components dialog box. Choose Close to return to the detail dialog box.

- f - To modify miscellaneous equipment related to the record shown in the detail dialog box,** choose the Misc Equipment button. The Miscellaneous Equipment dialog box displays listing any items in this category. Highlight an item and choose the Edit button to modify the item, or choose Delete to remove the item. Choose the Close button to save any changes and to return to the detail dialog box.
 - g - To review the software residing on the workstation, file server or MAC displayed in the detail dialog box,** choose the Software Applications button. The Software Applications dialog box displays the application name, version, file name and path of each software application associated with this record. Choose the Close button to close the Software Applications dialog box.
 - h - To modify a field in the detail dialog box,** double click on the field to be modified. The Equipment Component dialog box displays. For detailed information on reviewing, adding, or modifying purchasing and/or maintenance information for a detail field, refer to the topic entitled Maintaining Component Information.
- 4. To exit the detail dialog box and save any changes made to the equipment inventory record details, choose the OK button.**

The Inventory (summary) dialog box displays.

See Also:

Viewing Equipment Inventory Records

Maintaining Component Information

When an audit is performed, BrightWorks compares the new inventory data against the data maintained in its databases and the existing equipment inventory records. Performing an audit updates the data in the equipment inventory records.

You can enhance the equipment inventory record by adding and maintaining data regarding the purchase and maintenance of individual components associated with your PC workstations, file servers, and MACs.

Use the following procedure to review, add, or modify purchasing and/or maintenance information for a detail field.

1. Choose the Inventory tool bar button, or choose the Inventory command from the Tools menu.

The Inventory dialog box displays. This summary dialog box displays the workstations and file servers for all audited sites.

2. Highlight the record for which you want to manage component data, and choose the Edit button.

You can also double click on the record in the Inventory dialog box to view the record details. The detail dialog box displays.

3. To add, edit or delete purchasing and/or maintenance information for a *hardware component* associated with this record, double click on the field which represents the component.

For example, double click on the "Clk Freq" entry in the Computer Information section of the detail dialog box. The Equipment Component dialog box displays.

This dialog box lists the purchasing and maintenance information for the selected hardware component.

The following actions can be performed from within the Equipment Component dialog box:

a - To add or edit purchasing or maintenance information for the hardware, type new values in the corresponding fields, or select values from the dropdown lists. Choose the OK button to save the new field values.

b - To delete the hardware component, choose the Delete button. A dialog box displays with the message "Do you really want to delete this item?" Choose the Yes button to remove the component from the equipment inventory record.

4. To review the *software application* associated with this record, choose the Software Applications button in the detail dialog box.

The Software Applications dialog box displays listing all software files associated with this record.

a - To add or edit purchasing and/or maintenance information for a software application associated with this record, double click on the application name. The Equipment Component dialog box displays listing the purchasing and maintenance information for the selected software file. Enter new values in the corresponding fields, or select values from the dropdown lists. Choose the OK button to save the new field values.

From within the Equipment Component dialog box, **to review the path, version or size of the displayed software file**, choose the Additional Application Info button. The Additional Software Information dialog box displays the Path, Version, and Size of the selected file. Choose the OK button to return to the Equipment Component dialog box.

5. Choose the OK button to close the Equipment Component dialog box and save the editing changes.

Inventory and Distribution Reports

BrightWorks is shipped with several pre-defined inventory and distribution reports called "style sheets." These style sheets represent frequently requested reports which can also be customized to accommodate a specific need. By performing a "query," the information included in a report can be filtered according to virtually any combination of values maintained in the BrightWorks inventory database. For example, a report based on the Applications by Name style sheet will list all software applications in the BrightWorks inventory database. The reported software data can also be filtered to include only the Microsoft Word software running on 386 machines with CPU speeds greater than 16 MHz.

Queries can be saved and attached to any number of style sheets. The same queries can be applied to the scopes used within BrightWorks software distribution feature. (For more information, refer to [Scopes Queries](#).)

In addition to customizing the pre-defined style sheets, reports that are created using the Crystal Reports software can be added into the BrightWorks system. These added reports can be customized, renamed and/or deleted.

NOTE:

The Crystal Reports software is installed using the BrightWorks install utility. Refer to the installation instructions in Chapter 2 of the BrightWorks manual. When installed, a Crystal Reports program icon is added to the McAfee Program Manager group. For detailed instructions on using the Crystal Reports software, refer to either the Crystal on-line help system or Part Five of the BrightWorks manual.

Access to BrightWorks Inventory Reports

By choosing the Inventory and Distribution command from the Report menu, the Choose Report dialog box displays listing the BrightWorks inventory and distribution reports.

NOTE:

Choosing the Reports tool bar button will display the Choose Report dialog box that was last selected from the Reports menu (e.g., either the inventory/distribution reports or the metering/monitoring/security reports).

Refer to the following topics for instructions on creating inventory and distribution reports.

[Using Pre-defined Report Style Sheets](#) - Presents the procedures for generating pre-defined reports and selecting an output option.

[Using Queries to Customize Reports](#) - Presents the procedures for creating queries to filter report data, applying queries to reports and saving the report under a new name.

[Adding Reports](#) - Presents the procedures for adding and managing new reports.

Using Pre-defined Inventory and Distribution Report Style Sheets

The pre-defined inventory and distribution style sheets supplied with BrightWorks represent frequently requested inventory reports. A pre-defined report can be generated "as is," or it can be customized by applying queries which further define the data to be included in the report.

This topic lists the procedures for generating a report using the pre-defined style sheets. (The procedures for creating, applying and managing queries are discussed in [Using Queries to Customize Reports](#).)

NOTE:

The pre-defined BrightWorks report style sheets cannot be deleted.

Use the following procedure to generate a pre-defined report without applying any queries.

1. Choose the Inventory and Distribution command from the Reports menu.

The Choose Report dialog box displays listing the reports pertaining to inventory and distribution.

2. From the list of Report Names, select the report you want to generate.

Your selection is highlighted. The default selection is the first report name in the list.

3. Select the report's Destination.

The following destinations are available:

- * **Printer** - Sends the report to the printer and uses the currently defined Printer Set-up parameters. (Refer to [Printer Set-up and Administration](#) for instructions on administering your print set-up.)
- * **Screen** - Sends the report to a dialog box on your screen. Use the scroll bars to scroll through the report contents. Double click on the control menu button to close the dialog box when you are finished.
- * **File** - Sends the report to a file. When this option is selected, the File Format options become available. The format options are:
 - **Text** output file is saved in ASCII format.
 - **Comma Delimited** output file is saved in a comma delimited format in which commas are used to separate the fields.

4. Choose the OK button to initiate the creation of the report.

A Printing dialog box displays indicating the status of the report generation.

NOTES:

- a - *If the report is being sent to a file, you are prompted to enter a file name. Enter the file name and destination, and choose the OK button. The Printing dialog box displays even if the report is being sent to a file.*
- b - *If the report is sent to the screen, the resulting report displays in a Crystal Reports dialog box. The buttons at the top of the dialog box from left to right can be chosen for page scroll to first page, previous page, next page, last page, stop scroll, page magnification and route report to printer. For detailed instructions on using the Crystal Reports software, refer to either the Crystal on-line help system or Part Five of the BrightWorks manual.*

Using Queries to Customize Reports

Queries can be defined and applied against a report to act as a filter for the data gathered from the BrightWorks inventory database. Queries can be saved and applied to any number of reports. The same queries can also be used when creating scopes for BrightWorks software distribution, as discussed in Part Four of the BrightWorks manual.

Refer to the following topics:

[Applying a Query to a Report](#)

[Customizing Report Style Sheets](#)

[Removing a Query from a Report](#)

[Creating a New Query](#)

[Editing a Query](#)

[Deleting a Query](#)

Applying a Query to a Report

Applying a query against report data can be used to generate a custom report. Use the following procedure to apply an existing query to a report.

1. Choose the Inventory and Distribution command from the Reports menu.

The Choose Report dialog box displays listing the reports pertaining to inventory and distribution.

2. From the list of Report Names, select the report to which you want to apply a query.

Your selection is highlighted, and the Current Query field displays the name of the query currently applied to the selected report.

NOTE:

The <None> entry in the Current Query field indicates that no query is currently applied to the report.

3. Choose the Select button to the right of the Current Query field.

The Select Query dialog box displays listing all queries. (Refer to [Creating a New Query](#) for instructions on creating new queries if only the <None> entry appears in this list.)

4. Select the query name from the Available Queries list, and choose the Apply button.

To select a query name, point to the query and click the left mouse button. Upon choosing the Apply button, the Select Query dialog box closes and the selected query name is placed into the Current Query field of the Choose Report dialog box. The BrightWorks database records are sorted, and only the records that match the query's specified filter criteria will be included when the report is generated.

NOTE:

Before applying a query, make sure that the correct report name is highlighted in the Choose Report dialog box.

See Also:

[Customizing Report Style Sheets](#)

[Removing a Query from a Report](#)

[Creating a New Query](#)

[Editing a Query](#)

[Deleting a Query](#)

Customizing Report Style Sheets

Customized BrightWorks inventory and distribution reports can be created by assigning unique names to the pre-defined reports and then applying a query to each customized report.

NOTE:

For instructions on using the Crystal Reports software to create new inventory and distribution reports, refer to Part Five of the BrightWorks manual. For instructions on incorporating the new Crystal reports into BrightWorks, refer to [Adding New Reports](#).

Use the following procedure to customize a pre-defined style sheet.

- 1. From the list of Report Names in the Choose Report dialog box, select the report on which you want to base the new report.**

Select a report that consists of fields and data which are similar to the report to be created. For example, you might select the Workstation Equipment report to create a new report which lists the 386 workstation nodes.

- 2. Choose the New button in the Choose Report dialog box.**

The New Report dialog box displays.

- 3. Enter a new unique name in the Report Name field.**

When the New Report dialog box first displays, the name of the selected report is in the Report Name field. Delete the existing name, and enter the new report name.

- 4. Select a file name to be associated with the new report.**

The File Name field has an associated drop-down list of file names residing in the BrightWorks program directory. Because the new report is to be based on the report selected in the Choose Report dialog box, click on the file name of the selected report.

For example, if the new report is to be based on the pre-defined Applications by Group report, then select the file name BYGROUP.RPT.

- 5. Choose the OK button to save the new report information.**

The New Report dialog box closes, and the new report name is added to the Choose Report dialog box.

See Also:

[Applying a Query to a Report](#)

[Removing a Query from a Report](#)

[Creating a New Query](#)

[Editing a Query](#)

[Deleting a Query](#)

Removing a Query from a Report

Use the following procedure to remove a query from a report.

1. **From the list of Report Names in the Choose Report dialog box, select the report for which you want to remove the query.**

Your selection is highlighted, and the Current Query field displays the name of the query currently applied to the selected report.

2. **Choose the Select button to the right of the Current Query field.**

The Select Query dialog box displays.

3. **Select the <None> query name, and choose the Apply button.**

The Select Query dialog box closes. The filter criteria is removed and all records that apply to the report will be included when the report is generated.

See Also:

[Applying a Query to a Report](#)

[Customizing Report Style Sheets](#)

[Creating a New Query](#)

[Editing a Query](#)

[Deleting a Query](#)

Creating a New Query

Use the following procedure to create a new query. The procedure assumes that you have already chosen the Select button in the Choose Report dialog box to display the Select Query dialog box.

NOTE:

All queries are also available from the BrightWorks software distribution feature when creating scopes, as discussed in Scope Queries.

1. Choose the Add button in the Select Query dialog box.

The Add Query dialog box displays. Press the <TAB> key to move from field to field within this dialog box.

2. Enter a Query Name and define a filter entry.

The purpose of each filter entry is to narrow down the list of records to be included in a report. If more than one filter entry is defined, the entries are "linked" using either the AND or OR relationship.

For example, assume the following two filter entries:

```
Central Processing Unit = Intel_80386  
CPU Clock Frequency > 66.00 Mhz
```

If the entries are linked with the AND relationship, only the nodes that satisfy *both* criteria (i.e., the 80386 machines that have a clock speed greater than 66 Mhz) are included in the report definition.

If the entries are linked with the OR relationship, the nodes that satisfy *either* criteria (i.e., all Intel 80386 machines and all machines that have a clock speed greater than 66 Mhz) are included in the report definition.

For each filter entry, specify the following:

- * **Query Name** - Enter a query name up to 80 characters in length.
- * **Component** - Choose a component from the BrightWorks inventory database to use as the filter basis. Select a component from the drop-down list associated with this field (e.g., Brand, Computer Model, CPU Clock Frequency).
- * **Condition** - Choose a conditional operator from the drop-down list associated with this field (e.g., equal to, less than, greater than, not equal to). 'Equal to' is the default condition.
- * **Description** - If desired, choose a description of the component. (This field can be NULL.) The items which automatically display in this list depend on the selected component. For example, "Intel_80386" might display if Central Processing Unit is entered in the Component field; "16.00 Mhz" might display if CPU Clock Frequency is entered in the Component field. See Note (a) below.
- * **Query Link** - Specify the relationship between the filter entries (e.g., Central Processing Unit = 80386 OR Central Processing Unit = 80486). The link options are AND and OR. See Note (b) below.

NOTES:

a - To create a query which tests for the presence of a component, leave the Description field blank. For example, to include all nodes with a hard disk, construct a query with the following entries:

```
Component =  
Hard Disk #1 < >
```

In this case, the Component description is left blank, and the query results in including all nodes

which have a hard disk (i.e., Hard Disk #1 does not equal blank).

b - All filter entries in a query must have the same Query Link type (e.g., all entries will be linked by AND or all entries will be linked by OR).

3. Choose the Insert button to accept the filter entry definition.

The entry is added to the Current Query list in the Edit Query dialog box.

4. If required, insert additional filter entries.

Repeat steps #2 and #3 above.

NOTE:

To add a filter entry between existing entries, first highlight the filter entry line in the Current Query list where you want the new entry to be placed. The new defined entry is placed in the highlighted position.

5. When all filter entries are defined, choose the Save button.

The query is saved and added to the Available Queries list in the Select Query dialog box. The new query can now be applied to a report.

See Also:

[Applying a Query to a Report](#)

[Customizing Report Style Sheets](#)

[Removing a Query from a Report](#)

[Editing a Query](#)

[Deleting a Query](#)

Editing a Query

Use the following procedure to edit the definition of an existing query. The procedure assumes that you have already chosen the Select button in the Choose Report dialog box to display the Select Query dialog box.

- 1. Select a query from the Select Query dialog box, and choose the Edit button.**

The Edit Query dialog box displays showing the query's filter entries.

- 2. Modify the information, and choose the Save button.**

For instructions on modifying the filter entries, refer to the topic entitled "[Creating a New Query](#)."

To delete a filter entry, highlight the entry in the Current Query List and choose the Delete button.

NOTE:

To add a filter entry between existing entries, first highlight the filter entry line in the Current Query List where you want the new entry to be placed. The new defined entry is placed in the highlighted position.

See Also:

[Applying a Query to a Report](#)

[Customizing Report Style Sheets](#)

[Removing a Query from a Report](#)

[Creating a New Query](#)

[Deleting a Query](#)

Deleting a Query

Use the following procedure to delete an existing query. The procedure assumes that you have already chosen the Select button in the Choose Report dialog box to display the Select Query dialog box.

- 1. Select the query to be deleted from the Select Query dialog box, and choose the Delete button.**

A prompt displays asking you to verify the delete action. Choose the Yes button to delete the query. The query is removed from the Available Queries list.

NOTE:

Queries that are currently applied to a software distribution scope and/or BrightWorks inventory/distribution report can be deleted.

See Also:

[Applying a Query to a Report](#)

[Customizing Report Style Sheets](#)

[Removing a Query from a Report](#)

[Creating a New Query](#)

[Editing a Query](#)

Adding New Inventory and Distribution Reports

Additional reports can be incorporated into the BrightWorks system through the use of the Crystal Reports software. Queries can also be applied to the new reports, and once added, the reports can be renamed and/or deleted.

NOTE:

The Crystal Reports software is installed with BrightWorks. A Crystal Reports program icon is added to the McAfee Program Manager group. For detailed instructions on using the Crystal Reports software, refer to either the Crystal on-line help system or Part Five of the BrightWorks manual.

New reports created using the Crystal Reports software can be added into BrightWorks by using the following procedure. All report files (.RPT) must be located in the BrightWorks program directory.

Use the following procedure to add a new report into BrightWorks.

1. Choose the Inventory and Distribution command from the Reports menu.

The Choose Report dialog box displays.

2. Choose the New button.

The New Report dialog box displays.

3. Enter the Report Name of the report to be added.

The name entered in this field is the name that will display in the Choose Report dialog box.

4. Select the File Name, and choose the OK button.

Select the .RPT file name to be added into BrightWorks. (The .RPT files that display are located in the BrightWorks program directory.) The selected file will be associated with the Report Name entered in the above step.

See Also:

[Renaming Added Reports](#)

[Deleting Reports](#)

Renaming Added Inventory and Distribution Reports

Use the following procedure to rename a Crystal report that has been added into BrightWorks.

1. **Choose the Inventory and Distribution command from the Reports menu.**

The Choose Report dialog box displays.

2. **From the list of Report Names, select the report to be renamed, and choose the Rename button.**

The Rename Report dialog box displays prompting you to enter a new report name.

3. **Enter the new report name, and choose the OK button.**

The new report name displays in the Choose Report dialog box, and the old name is removed. All attributes of the old report are preserved in the renamed report (i.e., the report contents and applied query do not change).

See Also:

[Adding New Reports](#)

[Deleting Reports](#)

Deleting Reports

Use the following procedure to delete a Crystal report that has been added into BrightWorks.

NOTE:

A pre-defined BrightWorks report style sheet cannot be deleted; however, a report that was created using the Crystal Reports software and then added into BrightWorks can be deleted.

1. Choose the Inventory and Distribution command from the Reports menu.

The Choose Report dialog box displays.

2. From the list of Report Names, select the report to be deleted, and choose the Delete button.

A prompt displays asking you to confirm the deletion.

3. Choose the Yes button to delete the report.

If deleted, the report name is removed from the Choose Report dialog box.

See Also:

[Adding New Reports](#)

[Renaming Added Reports](#)

BrightWorks Distribution Help Index

About Software Distribution

[Distribution Capabilities](#)

[Distribution Concepts](#)

[Distribution Modules](#)

How To

[Manage Filesets](#)

[Manage Scripts](#)

[Use the Distribution Script Language](#)

[Manage Scopes](#)

[Manage Packages](#)

[Monitor Software Distribution](#)

[Use the Update Program](#)

Software Distribution Capabilities

BrightWorks software distribution capabilities provide a method for distributing software packages and modifying workstation configuration files from a central location. The software distribution features facilitate consistency among the workstations across your local area network and improve the productivity of the LAN Administrator.

Distributing software and modifying workstation configuration files from a central location on your network allows the LAN Administrator to easily do the following:

- * update system executables and/or drivers (e.g., operating systems, network drivers)
- * update system files (e.g., AUTOEXEC.BAT, CONFIG.SYS, WIN.INI, network login script)
- * install and update software on user workstations across the local area network

NOTE:

BrightWorks Software Distribution capabilities can be used to distribute software and/or scripts to any workstation in the BrightWorks local site (i.e., the site which identifies the BrightWorks program directory). Sites are discussed in detail in the topic entitled [Defining LAN Sites](#).

See Also:

[Distribution Concepts](#)

[Distribution Modules](#)

Software Distribution Concepts

An understanding of the following concepts will help you in gaining full advantage of BrightWorks software distribution capabilities:

- * **Fileset** - A file that contains one or more compressed files. Each compressed file may also indicate a target directory structure in which the file should be decompressed. For example, assume a fileset named NEW_INI_FILES. The fileset might consist of two files: WIN.INI and SYSTEM.INI which have been defined to be decompressed into a target directory named PUB\WIN.310.
- * **Script** - A sequence of one or more commands which define an operation to be performed on a workstation receiving a distribution. For example, a script might include the commands to add a new group to the Windows Program Manager, to copy file(s) from one location to another, or to change parameters within certain files.
- * **Scope** - A group of one or more workstations that have been identified to receive a distribution. For example, to distribute a script to all 386 workstations, you must create a scope which includes the 386 workstations.
- * **Package** - The distributed object which contains scheduling information, as well as a fileset and/or script and a scope.

The following key features help you distribute software across your LAN:

- * Creating *filesets* which include files to be installed on remote workstations.
- * Creating *scripts* to be executed by remote workstations.
- * Defining *scopes* of workstations to receive distributed packages.
- * Creating and scheduling *packages* which consist of a scope and one fileset and/or one script.
- * Monitoring package progress via the Software Distribution Log History dialog box.

See Also:

[Distribution Capabilities](#)

[Distribution Modules](#)

Software Distribution Modules

The BrightWorks software distribution capabilities interact with two major functional modules. As an introduction to software and script distribution, this section briefly describes the following modules:

- * BrightWorks console and administrative functions
- * Remote workstation update program (SDUPDATE.EXE)

BrightWorks Console/ Administrative Program

BWORKS.EXE is the BrightWorks console and administrative program. This program provides access to all BrightWorks capabilities. This main module is a Windows-based program and is intended to be used by the LAN network manager to perform all software distribution functions.

The software distribution functions available from the BrightWorks console include:

- * Scope definition and management
- * Script creation and management
- * Package creation, scheduling and management
- * Pre-defined and custom distribution report generation

Update Program

The update program (SDUPDATE.EXE) must be executed from each remote workstation in order for the workstations to receive the distributed packages they have been sent. Upon BrightWorks installation, the update program is copied into the BrightWorks program directory.

The update program is DOS-based and must be executed from the machine which is to be updated. To ensure that SDUPDATE.EXE is executed on a regular basis, the command can be placed in the system login script. Refer to the section entitled Running the Update Program.

NOTE:

WSDUPD.EXE is the BrightWorks update program which handles the script functions related to installing Windows software. This program must not be directly run by the user it is automatically loaded when the ADDGROUP, ADDITEM or SCHEDULEWIN Windows System File script functions are used. For more information, refer to Windows System File Functions.

See Also:

Distribution Capabilities

Distribution Concepts

The Update Program

Introduction to Filesets

A fileset is a set of files, stored in compressed format, to be installed on a remote workstation. Distributing filesets from a central location simplifies a System Administrator's job. Instead of physically moving from workstation to workstation to install or upgrade application files, the Administrator only needs to centrally distribute one fileset consisting of the application files. Upon receipt at a remote workstation, the fileset contents are decompressed and copied onto the workstation's hard drive.

Access to Fileset Functions

Most fileset functions are accessed by choosing the Filesets command from the Tools menu. The Filesets dialog box displays listing all available filesets.

The process of defining a directory in which filesets are saved (the "default fileset directory") is performed by choosing the Distribution command from the Administration menu. From the sub-menu which displays, choose the Preferences command.

Fileset Features

In addition to containing a number of files to be distributed, filesets can be defined to create a target directory structure. For example, if you create a fileset which includes all files for Windows 3.1, you must also define the contents of the SYSTEM subdirectory. BrightWorks can do this for you automatically by including the full path name of every file included in the fileset.

Filesets and scripts are a powerful combination. Consider the following examples:

- * Packaging the latest WIN.INI file with a script which determines whether the existing WIN.INI file is outdated. The script will also copy the new WIN.INI if an old file is detected.
- * Packaging the Novell IPXODI files and sending them to the scope of nodes using IPX. After the fileset is decompressed in the target directory, the script will update the AUTOEXEC.BAT file to reflect the use of IPXODI.

Filesets can be stored, used and reused as a resource within BrightWorks. An administrator can create a new fileset, as well as edit, copy, rename and delete a fileset. Refer to the following topics:

The Fileset Directory

Creating Filesets

Editing Filesets

Renaming Filesets

Copying Filesets

Deleting Filesets

The Fileset Directory

The fileset directory defines the path in which filesets are stored. Upon saving a fileset, a copy of the files that are included in the fileset are compressed. They are stored in a file which is placed in the fileset directory defined at the time the fileset is saved. For example, if your fileset directory is defined as F:\FUSION\FILESETS, then the filesets that you create will be stored in the F:\FUSION\FILESETS directory.

Defining a Fileset Directory

Use the following procedure to define the directory in which filesets should be stored.

1. **Choose the Distribution command from the Administration menu. From the sub-menu which displays, choose the Preferences command.**

The Preferences dialog box displays.

2. **Choose the Browse button to define the pathname into which the compressed filesets are to be stored.**

The Path Browse dialog box displays enabling you to select from the lists of Drives and Directories. Click on the Drives and Directories fields to select the desired pathname.

3. **When the pathname is selected, choose the OK button.**

The Path Browse dialog box closes, and the selected pathname displays in the Path to Filesets field of the Preferences dialog box.

4. **Choose the Save button to define the fileset directory.**

All saved filesets will be stored in the defined directory.

NOTE:

The fileset directory instructs the update program as to where the filesets are located. As a result, changing the fileset directory after you have created filesets and included them in packages can invalidate those packages. If you change the default directory, you must also copy all fileset files (.SET) into the new fileset directory. Be sure that each user has the same drive letter mapped to the same server/volume specified in the Preferences dialog box. Also, you cannot store filesets on a non-network drive.*

See Also:

[Creating Filesets](#)

[Editing Filesets](#)

[Renaming Filesets](#)

[Copying Filesets](#)

[Deleting Filesets](#)

Creating Filesets

Use the following procedure to create a new fileset.

1. Choose the Filesets command from the Tools menu.

The Filesets dialog box displays listing the names of all defined filesets.

2. Choose the New button.

The New Fileset dialog box displays prompting you to enter a name for the new fileset.

3. Enter the new fileset name, and choose the OK button.

A fileset name can be up to 80 characters, and all typed characters are valid.

Upon choosing the OK button, the Edit Fileset dialog box displays prompting you to define the contents of the new fileset.

The fileset name being created or edited displays in the title bar of the Edit Fileset dialog box.

For each file included in the fileset, the following information displays: File Name, Original Size, Compression Ratio, Date, Time, and Path. The file list area is empty for new filesets.

The Filename field in this dialog box displays the name of the file which will hold the compressed fileset. This file is automatically created by BrightWorks when the fileset is created. It is stored in the fileset directory which is currently defined.

4. To add a file to the fileset, choose the Add button.

The Add File dialog box displays. This dialog box is a standard Windows dialog box used for opening, selecting and browsing files.

5. Make selections from the Directories and Drives lists to find the file(s) to include in the fileset.

For example, choose the Drives down arrow button, and click on Drive C: to display the directories on drive C. From the list of directories which displays, click on one to display its file list.

6. Select a file(s) from the File Name list.

Multiple files can be selected using the Windows extended select procedures (i.e., hold down the <CTRL> or <SHIFT> key while selecting files).

7. To include the selected file(s)' path in the Edit Fileset dialog box, enable the Include Path option.

Placing a checkmark in this field causes the full pathnames of each selected file to be listed in the Edit Fileset dialog box. (Step #9 below provides the option to instruct the fileset to create the directory structure at the receiving workstation.)

8. Choose the OK button.

Upon choosing the OK button, the selected files are listed in the Edit Fileset dialog box. Only the File Name and Path information display at this time. The other fields are not available until the fileset is saved.

9. Enable or disable the Create Directory Structure option.

Enabling this option causes the full pathnames of each file listed in the Edit Fileset dialog box to be created at the receiving workstation. For example, assume that this option is enabled and a file is listed in the Edit Fileset dialog box as \USER\MARY\INVOICE.DOC. In this case, the directories USER and MARY will be created at the receiving workstation if they do not already exist.

NOTE:

A fileset is always decompressed into the target directory that is specified when creating a package. In the above example, if the Create Directory Structure option is checked and the fileset is included in a package that has a default path of C:\SALES, then the INVOICE.DOC file will be decompressed into C:\SALES\USER\MARY.

10. To save the fileset contents, choose the Save button.

The changes made to a fileset are only committed to upon choosing the Save button. The Updating Fileset dialog box displays while the fileset contents are being saved and compressed. If you attempt to close the Edit Filesets dialog box without saving, you are prompted to save the fileset changes.

The fileset is created and added to the Filesets dialog box.

See Also:

[The Fileset Directory](#)

[Editing Filesets](#)

[Renaming Filesets](#)

[Copying Filesets](#)

[Deleting Filesets](#)

Editing Filesets

Editing a fileset may become necessary in order to add or delete a file according to a change in a fileset's intent.

NOTE:

It is recommended that you temporarily deactivate any packages which use the fileset you intend to edit.

Use the following procedure to edit the contents of a fileset. The procedure assumes that you have already chosen the Filesets command from the Tools menu to display the Filesets dialog box.

1. Select the fileset from the list of Filesets, and choose the Edit button.

A fileset can also be selected for editing by double clicking on the fileset name in the Filesets dialog box. The Edit Fileset dialog box displays listing all files included in the fileset.

For each file in the fileset, the following information displays:

- * **File Name** - the name of the file
- * **Original Size** - the file size before compression
- * **Compressed Size** - the file size after compression
- * **Ratio** - the compression ratio
- * **Date** - the file's creation date
- * **Time** - the file's creation time
- * **Path** - the file's path which displays only if the Include Path option is checked in the Add File dialog box

NOTE:

Some files may show a 0% compression ratio. This occurs when the file is already compressed or when the file is very small.

2. To add a file to the fileset, choose the Add button.

The Add File dialog box displays. Refer to the topic [Creating Filesets](#) for detailed procedures on using this dialog box.

3. To delete a file from the fileset, highlight the file name and choose the Delete button.

A prompt displays asking you to confirm the deletion. Choose the Yes button to continue with the delete action.

If deleted, the file name is removed from the Edit Filesets dialog box.

4. To save the edited fileset contents, choose the Save button.

The changes made to a fileset are only committed to upon choosing the Save button. The Updating Fileset dialog box displays while the fileset contents are being saved and compressed. If you attempt to close the Edit Filesets dialog box without saving, you are prompted to save the fileset changes.

See Also:

[The Fileset Directory](#)

Creating Filesets

Renaming Filesets

Copying Filesets

Deleting Filesets

Renaming Filesets

Changing the name of an existing fileset renames all instances of the former fileset name. For example, the new fileset name is reflected in the Filesets dialog box as well as in any packages which include the fileset.

Use the following procedure to rename a fileset. The procedure assumes that you have already chosen the Filesets command from the Tools menu to display the Filesets dialog box.

NOTES:

a - A fileset can be renamed even if it is part of an actively scheduled package.

b - The name of the .SET file which is maintaining the compressed fileset and is stored in the fileset directory does not change.

- 1. To rename a fileset, select the fileset name from the list of Filesets, and choose the Rename button.**

The Rename Fileset dialog box displays prompting you to enter a new fileset name.

- 2. Enter the new fileset name, and choose the OK button.**

The new fileset name displays in the Filesets dialog box, and the old name is removed. All attributes of the old fileset are preserved in the renamed fileset (i.e., the fileset contents do not change).

See Also:

[The Fileset Directory](#)

[Creating Filesets](#)

[Editing Filesets](#)

[Copying Filesets](#)

[Deleting Filesets](#)

Copying Filesets

Use the following procedure to copy a fileset. The procedure assumes that you have already chosen the Filesets command from the Tools menu to display the Filesets dialog box.

NOTE:

A fileset can be copied even if the original fileset is part of an actively scheduled package.

1. **To copy a fileset, select the fileset name from the list of Filesets, and choose the Copy button.**

The Copy Fileset dialog box displays prompting you to enter a name for the new fileset.

2. **Enter the new fileset name, and choose the OK button.**

The new fileset name is added to the Filesets dialog box. The new fileset contents are identical to the original fileset contents.

See Also:

[The Fileset Directory](#)

[Creating Filesets](#)

[Editing Filesets](#)

[Renaming Filesets](#)

[Deleting Filesets](#)

Deleting Filesets

Use the following procedure to delete a fileset. The procedure assumes that you have already chosen the Filesets command from the Distribution menu to display the Filesets dialog box.

NOTE:

A fileset that is part of a scheduled package cannot be deleted.

1. **To delete a fileset, select the fileset from the list of Filesets, and choose the Delete button.**

A prompt displays asking you to confirm the deletion.

2. **Choose the Yes button to delete the fileset.**

The fileset name is removed from the Filesets dialog box.

See Also:

[The Fileset Directory](#)

[Creating Filesets](#)

[Editing Filesets](#)

[Renaming Filesets](#)

[Copying Filesets](#)

Introduction to Scripts

A script is a series of commands to be executed on a remote workstation. Scripts must be written according to a defined syntax, and they must be compiled successfully to be included in a package.

NOTES:

a - The commands and instructions for using BrightWorks software distribution scripting language can be referenced in [Using the Distribution Script Language](#).

b - BrightWorks is shipped with several script files that can be customized for your own use.

Access to Script Functions

Script functions are accessed by choosing the Scripts command from the Tools menu. The Scripts window displays listing all available scripts.

Script management is performed by either choosing the buttons in the Scripts window or by choosing the corresponding commands from the File menu. For example, when the Scripts window is active, a new script can be created either by choosing the New button in the Scripts window or by choosing the New Script command from the File menu.

The Edit menu commands provide the standard Cut, Copy and Paste functionality for use during script editing.

Script Features

The ability to send scripts from a central location can be used to contribute to the consistency and standardization of LAN workstations. Scripts enable the LAN administrator to easily do the following:

- * update system executables and/or drivers (e.g., operating systems, network drivers)
- * update system files (e.g., AUTOEXEC.BAT, CONFIG.SYS, WIN.INI, network login script)
- * install software on a user's workstation

A user can create a new script, as well as edit, compile, copy, rename and delete a script. Refer to the following topics:

[Creating Scripts](#)

[Compiling Scripts](#)

[Editing Scripts](#)

[Renaming Scripts](#)

[Copying Scripts](#)

[Deleting Scripts](#)

Creating Scripts

A script is created by assigning both a script name and a file name to the new script. The script *name* is used for identification purposes within BrightWorks. For example, it is immediately obvious that the script named "Upgrade to Win 3.1" is responsible for upgrading the Windows software to version 3.1. The script *file name* identifies the ASCII text file containing the script commands. The script file name must be a valid DOS file name (e.g., 8 characters plus the 3 character extension).

After assigning the script name and file name, an "empty" script is created. The empty script must be edited in order to add commands.

Use the following procedure to create a new script.

- 1. Choose the Scripts command from the Tools menu.**

The Scripts window displays. This window lists the names of all defined scripts. For each script, the last compilation date, the status and the file name also displays.

- 2. Choose the New button.**

The Open New Script dialog box displays prompting you to enter the name, file name and destination directory for the new script.

- 3. Enter the new script information, and choose the OK button.**

The script *name* can be up to 80 characters, and all typed characters are valid. The script *file name* must follow the standard DOS conventions.

NOTES:

a - It is recommended that .SCR be assigned as the extension for all script file names. A script file is a text file and can be edited with an external editor.

b - The scripts must reside on a network drive to which all users who will receive the script have access

Upon choosing OK, the message "This file does not exist. Create the file?" displays. Choose the Yes button to create the script file and display the Script Editor window.

The script name being edited displays in the title bar of the Script Editor window. All commands that are included in this script are listed. (The list is empty for new scripts.)

- 4. Type the script commands.**

Script commands can be directly typed into the Script Editor window. Commands can also be selected from a list of commands by choosing the Functions button in the Script Editor window or the Paste Script Function command from the BrightWorks Edit menu (refer to the explanation below).

The script compiler requires one command per line. No error checking is performed until the script is compiled.

Optional comments can be placed in the script preceded by a semi-colon. These comments are ignored at compile time. For example:

```
;This is a comment.
```

NOTE:

The commands and instructions for using BrightWorks software distribution scripting language can be referenced in [Using the Distribution Script Language](#).

Standard editing functions are available from the Edit menu on the BrightWorks menu bar. The commands that are available from the Edit menu are as follows:

- * **Undo** - removes the last change made to the script.
- * **Cut** - copies a block of selected text to the clipboard and removes the text from the Script Editor window.
- * **Copy** - copies a block of selected text to the clipboard.
- * **Paste** - places the block of text from the clipboard into the Script Editor window at the current cursor location.
- * **Paste Script Function** - displays the Choose Script Function dialog box. This dialog box allows you to select a function (from a list of all script functions) to be placed in the script at the current cursor location. A function can be selected by either double clicking on the function name, or highlighting the name and choosing the OK button. Choosing the Help button displays help text for the highlighted function.
- * **Find** - searches the script for a user-specified text string.
- * **Next** - searches the script for the next occurrence of the user-specified text string.
- * **Replace** - searches the script for a user-specified text string and replaces the found text with another user-specified text string.
- * **Fonts** - enables you to select the font, style and size of the script type.

NOTE:

During script editing, the status bar in the BrightWorks application window indicates the current line and column position of the typing cursor.

5. To compile the script, choose the Compile button in the Script Editor window.

Refer to [Compiling Scripts](#) for detailed instructions.

6. To save the script contents, choose the Save button in the Script Editor window.

The saved script contents are stored in ASCII text format. The script must be compiled to be used in a package.

7. Choose the Close button to close the Script Editor window.

If you did not save the script changes as in Step #6 above, you are prompted to do so now. Choose the Yes button to save the script changes, or choose No to close the Script Editor without saving any changes.

The new script is added to the Scripts window. The status of all uncompiled scripts is 'ASCII.' A script must be compiled to be used in a package.

See also:

[Compiling Scripts](#)

[Editing Scripts](#)

[Renaming Scripts](#)

[Copying Scripts](#)

[Deleting Scripts](#)

Compiling Scripts

The Status field in the Scripts window indicates the status of each script. Script status can be either ASCII or COMPILED. A script's status must be COMPILED to be used in a package for distribution. The compilation process checks the syntax and validity of the script's commands.

Use the following procedure to compile a script. The procedure assumes that you have already chosen the Scripts command from the Tools menu to display the Scripts window.

1. To compile a script, select the script in the Scripts window and choose the Compile button.

While a compile is in progress, the Compile Status dialog box displays.

When the compile is complete, the Status field in the Compile Status dialog box indicates success or failure. If the compile fails, the Function field indicates the first function found which has invalid parameters. The Statistics area indicates the total number of lines in the script (Lines field) and the number of errors found (Errors field).

2. Choose the OK button to continue.

If the script compile is successful, then choose the Close button in the Script Editor window to return to the Scripts window which shows the script's status as COMPILED.

If the script compile fails, then the Compiler Messages dialog box displays listing the first script line which contains errors.

3. To correct a compiler error condition, double click on an error line in the Compiler Messages dialog box.

The Script Editor window displays with the script that you are attempting to compile. The selected error line is automatically highlighted.

4. Correct all error conditions, and attempt to re-compile the script.

Refer to [Using the Distribution Script Language](#) for details on the scripting rules and commands.

After successful compilation of the script, the script can be used in a package.

NOTE:

If you edit a script that has already been compiled, the script must be successfully re-compiled in order to be used in a package. Refer to the "Last Comp" field in the Scripts window to discover the date on which the file was last compiled.

See also:

[Creating Scripts](#)

[Editing Scripts](#)

[Renaming Scripts](#)

[Copying Scripts](#)

[Deleting Scripts](#)

Editing Scripts

Editing a script may become necessary under two circumstances:

- * Existing scripts might need to be edited in order to add or delete commands according to a change in a script's intent.
- * When a script compilation fails, the script must be edited to resolve the error(s).

NOTE:

It is recommended that you temporarily deactivate any packages which use the script you intend to edit.

Use the following procedure to edit the contents of a script. The procedure assumes that you have already chosen the Scripts command from the Tools menu to display the Scripts window.

1. Select the script from the Scripts window, and choose the Edit button.

A script can also be selected for edit by double clicking on the script name in the Scripts window. The Script Editor window displays.

The script name being edited displays in the title bar of the Script Editor window. All commands that are included in this script are listed.

2. Edit the script commands.

Script commands can be directly typed into the Script Editor window. Commands can also be selected from a list of commands by choosing the Functions button in the Script Editor window or the Paste Script Function command from the BrightWorks Edit menu.

The script compiler requires one command per line. No error checking is performed until the script is compiled.

NOTES:

a - The commands and instructions for using BrightWorks software distribution scripting language can be referenced in [Using the Distribution Script Language](#).

b - During script editing, the status bar in the BrightWorks application window indicates the current line and column position of the typing cursor.

3. To compile the script, choose the Compile button in the Script Editor window.

Refer to [Compiling Scripts](#) for detailed instructions.

4. To save the edited script contents, choose the Save button in the Script Editor window.

5. Choose the Close button to close the Script Editor window.

NOTE:

If you edit a script that has already been compiled, the script must be successfully re-compiled in order to be used in a package.

See also:

[Creating Scripts](#)

[Compiling Scripts](#)

[Renaming Scripts](#)

[Copying Scripts](#)

[Deleting Scripts](#)

Renaming Scripts

Changing the name of an existing script renames all instances of the former script name. For example, the new script name will be reflected in the Scripts window as well as in any packages which include the script.

Use the following procedure to rename a script. The procedure assumes that you have already chosen the Scripts command from the Tools menu to display the Scripts window.

NOTE:

A script can be renamed even if it is part of an actively scheduled package.

1. **To rename a script, select the script name from the Scripts, and choose the Rename button.**

The Rename Script dialog box displays prompting you to enter a new script name.

2. **Enter the new script name, and choose the OK button.**

The new script name displays in the Scripts window, and the old name is removed. All attributes of the old script are preserved in the renamed script (i.e., the script contents do not change).

NOTE:

The script rename procedure only changes the script name; the script file name does not change.

See also:

[Creating Scripts](#)

[Compiling Scripts](#)

[Editing Scripts](#)

[Copying Scripts](#)

[Deleting Scripts](#)

Copying Scripts

Use the following procedure to copy a script. The procedure assumes that you have already chosen the Scripts command from the Tools menu to display the Scripts window.

NOTE:

A script can be copied even if the original script is part of an actively scheduled package.

1. To copy a script, select the script from the Scripts, and choose the Copy button.

The Copy Script dialog box displays prompting you to specify a name, file name and destination directory for the new script. The script name can be up to 80 characters, and all typed characters are valid. The script file name must follow the standard DOS conventions and can reside in any directory.

NOTE:

It is recommended that .SCR be assigned as the extension for all script file names.

2. Enter the new script information, and choose the OK button.

The new script name is added to the Scripts window. The new script is populated with the same commands as the original script.

See also:

[Creating Scripts](#)

[Compiling Scripts](#)

[Editing Scripts](#)

[Renaming Scripts](#)

[Deleting Scripts](#)

Deleting Scripts

Use the following procedure to delete a script. The procedure assumes that you have already chosen the Scripts command from the Tools menu to display the Scripts window.

NOTE:

A script that is part of a scheduled package cannot be deleted.

1. **To delete a script, select the script from the Scripts, and choose the Delete button.**

A prompt displays asking you to confirm the deletion.

2. **Choose the Yes button to delete the script.**

If deleted, the script name is removed from the Scripts window.

NOTE:

The delete action only deletes the script name from the Scripts window. The corresponding .SCR file is not deleted. Therefore, if a script name is inadvertently deleted, you can create a new script and assign the same script file name to retrieve the deleted script contents.

See also:

[Creating Scripts](#)

[Compiling Scripts](#)

[Editing Scripts](#)

[Renaming Scripts](#)

[Copying Scripts](#)

Introduction to the Distribution Script Language

A script is a series of commands to be executed on a remote workstation. Scripts must be written according to a defined syntax, and they must be compiled successfully to be included in a package.

The commands and instructions for using the BrightWorks software distribution scripting language are presented in the following topics:

[Function Set Overview](#)

[Rules and System Variables](#)

[DOS Error Codes](#)

[DOS Functions](#)

[Easy System File Functions](#)

[Windows System File Functions](#)

[Miscellaneous Functions](#)

See Also:

[Introduction to Scripts](#)

Function Set Overview

Each script "command" is treated as a "function" (e.g., a C function) that has two basic properties:

- * each command has 0 to 4 parameters that it will be passed
- * most commands have a return code

As such, the script language supports user defined variables (of integer and string type), as well as "system" variables. When necessary, the functions also implement return values from the parameters that are passed.

Each function has one or more parameters that it can be passed. In the following discussions, the required parameters are surrounded by [], and the optional parameters are surrounded by {}. Each parameter is the name of a rule, whose allowable values are listed in [Rules and System Variables](#).

Notes on Function Syntax

The following items must be noted when writing scripts:

- * Only one command can be placed on a line.
- * The syntax for each command/function is as follows:

```
FUNC_NAME [parameter1], [parameter2], ... [parameterN]
```

- * Unless otherwise noted, each function returns a 0 if it is successful (i.e., the system variable [retval] is set to 0). The action to be taken as a result of a script's return code is defined when the script is included in a package. Refer to [Advanced Package Options](#) for more information.
- * Some functions take "optional" parameters. It is up to the administrator to decide whether or not these parameters are to be specified. If they are not specified, an empty or NULL value must be passed to the compiler to act as placeholders.

For example, the COPY function has the following parameters:

```
COPY [path] [filewild] [path] {filewild}
```

where the last parameter, {filewild}, is optional. The COPY command below provides an example for copying all .BAT files from the C: drive to the B: drive, using a placeholder to stand for the last {filewild} parameter:

```
COPY "C:\" "*" .BAT" "B:\" ""
```

In this example, the files are not renamed and retain their original .BAT extensions.

DOS Functions

The DOS function set is used for managing a machine's files and directories. For example, files can be searched for, copied, deleted, renamed and tagged with a specified attribute; directories can be created and deleted.

Return values are generated when appropriate (unless otherwise noted, the functions return 0 if successful). Any applicable system variables are also noted.

Most DOS functions return a DOS error code if unsuccessful. Refer to DOS Error Codes for a list of the DOS errors that may be returned.

NOTES:

a - When an "explicit <path>" is mentioned, it can take the form of D:\PATH (SERVER/VOLUME:\PATH is not currently supported).

b - Some functions take optional "options." It is up to the administrator to decide whether or not these options are to be specified.

c - In the following function specifications, parameters in quotes represent literal parameters; all other parameters represent rules. For more information, refer to the list of Rules and System Variables.

See Also

ATTRIB

COPY

DELETEDIR

DELETEFILE

FINDFILE

MDIR

RENAME

UPGRADEOS

ATTRIB [path] [filewild] [attribute]

Parameter	Description and Notes
[path]	Source path to files. This path must exist.
[filewild]	The file name whose attributes are to be changed. May contain wildcards (? and *).
[attribute]	RO - Read only RW - Read/Write A - Set Archive bit SY - System file H - Hidden file SH - Shareable (network <path> only) -A - Turn off archive attribute -SY - Turn off system attribute -H - Turn off hidden attribute -SH - Turn off shareable attribute (network <path> only)

Description - Changes the attributes of a file or multiple files.

Tip - To remove the Read Only attribute, use the RW attribute. (There is no -RO attribute.)

Return Values:

[RETVAL] = 0 if successful

[RETVAL] = -1 if the SH or -SH attributes are used and the drive letter specified in [PATH] is not a network drive

[RETVAL] = -2 if the SH or -SH attributes are used and no drive letter is specified in [PATH]

[RETVAL] = DOS error code in all other cases

Example - Set the AUTOEXEC.BAT file on a user's boot drive to Read Only:

```
ATTRIB [BOOT_ROOT] "AUTOEXEC.BAT" RO
```

COPY [path] [filewild] [path] {filewild}

Parameter	Description and Notes
[path]	Source path of file to be copied.
[filewild]	Source file name to be copied. May contain standard DOS wild cards (? and *).
[path]	Destination path.
{filewild}	<i>Optional</i> destination file name. (If not specified, *.* is assumed.) May contain standard DOS wild cards (? and *). May be used to rename file(s) during file copy. If not used, NULL must be specified.

Description - Copies a file or files to another directory and file name(s).

Return Values:

[RETVAL] = 0 if file(s) are copied correctly

[RETVAL] = DOS error code if the function is unsuccessful

Example - Copy the WIN.INI file from the Windows directory found at login to the local Windows directory. Two examples of this are:

```
COPY [WINDIR] "WIN.INI" "C:\WINDOWS" ""
```

or

```
COPY [WINDIR] "WIN.INI" "C:\WINDOWS" NULL
```


DELETEDIR [path] [filename] {deleteopt}

Parameter	Description and Notes
[path]	Source path to the directory to be deleted. This path must exist.
[filename]	Directory name to be deleted.
{deleteopt}	<i>Optional</i> delete option: ALL - causes DELETEDIR to delete the specified directory and everything under it, including any subdirectories, hidden, system and read only files. If not used, NULL must be specified.

Description - Deletes a directory.

Tip - Use the ALL delete option with caution since it can delete entire directory trees.

Return Values:

[RETVAL] = 0 if the directory is successfully deleted

[RETVAL] = DOS error code if the function is unsuccessful

Example - Delete the Windows directory found at login and all of its files and sub-directories:

```
DELETEDIR [WINDIR] ALL
```

DELETEFILE [path] [filewild]

Parameter	Description and Notes
[path]	Source path to the file(s) to be deleted. This path must exist.
[filewild]	File name(s) to be deleted. Wild cards may be specified (? and *) to delete multiple files.

Description - Deletes a file or multiple files.

Return Values:

[RETVAL] = 0 if the file(s) are deleted

[RETVAL] = DOS error code if the function is unsuccessful

Example - Delete all .DOC files from the F:\UZR\JOHN sub-directory:

```
DELETEFILE "F:\UZR\JOHN" "*.DOC"
```

FINDFILE [path] [filewild] [strvar]

Parameter	Description and Notes
[path]	Source path in which to search for files. This path must exist.
[filewild]	The search criteria. May contain wildcards (? and *).
[strvar]	A string variable which contains the file name of the first file found. (Before being used as a parameter, this variable must be defined using the DEFINE function.)

Description - Finds a file.

Return Values:

[RETVAL] = 0 and copies the name of the first file found into [STRVAL] if successful

[RETVAL] = -1 and sets [STRVAL] to NULL if no files are found

Example - Test for the presence of the NET.CFG file in the [NET.CFG] directory:

```
DEFINE "Result" STRING  
  
FINDFILE [NETCFG] "NET.CFG" RESULT
```

MDIR [path] [filename]

Parameter	Description and Notes
[path]	Path in which to create the new directory. This path must exist.
[filename]	Sub-directory to create. Wild cards may not be specified.

Description - Creates a directory.

Return Values:

[RETVAL] = 0 if the directory is successfully created

[RETVAL] = DOS error code if the function is unsuccessful

Example - Create the JOHN sub-directory in the UZR directory:

```
MDIR "F:\UZR" "JOHN"
```

RENAME [path] [filewild] [path] [filewild]

Parameter	Description and Notes
[path]	Source path to files to be renamed. This path must exist.
[filewild]	Source file name to be renamed. May contain wildcards (? and *).
[path]	Destination path (can be different than [path] to enable moving files, but the drives must be the same).
[filewild]	New file name. May contain wildcards (? and *). If so, the standards used by the DOS REN command are followed.

Description - Renames a source file(s).

Return Values:

[RETVAL] = 0 if successful

[RETVAL] = DOS error code in all other cases

Example - Rename all .BAT files in the C:\ drive to .BAK:

```
RENAME "C:\ " "*.BAT" "C:\ " "*.BAK"
```

UPGRADEOS [upgopt]

Parameter	Description and Notes
[upgopt]	5.00 - upgrade DOS version to 5.00; 6.00 - upgrade DOS version to 6.00

Description - Upgrades DOS version from 3.x-5x to either 5.00 or 6.00.

Tips:

1) In order for BrightWorks to have access to the upgraded DOS files, EQUIP must first be run on a machine that has the desired DOS files. For example, to upgrade a machine's DOS version to 6.00, EQUIP first must be executed on a machine that has DOS 6.00. By executing EQUIP from the same directory in which the BrightWorks software distribution update program (SDUPDATE.EXE) file is located, the DOS files become accessible by BrightWorks. Note also that the machine on which EQUIP is run must not contain any system that modifies the machine's boot record (e.g., OS/2, Windows NT).

NOTE:

Do not use this function on a workstation that has Windows NT installed in a dual boot configuration. It will cause the boot menu to be lost. The PC will boot DOS only.

2) The machine must be rebooted after the script is executed. Use the REBOOT function as the last script function.

Return Values:

[RETVAL] = 0 if successful

[RETVAL] = DOS error code in all other cases

Example - Upgrade a user's DOS version to 5.00:

```
UPGRADEOS 5.00
IF [RETVAL]=0
    ...
    ; copy DOS files, edit CONFIG.SYS, etc.
    REBOOT
ENDIF
```

Easy System File Functions

The Easy System File functions allow for easy manipulation of basic system files, such as CONFIG.SYS, AUTOEXEC.BAT, NET.CFG, or a login script. (Use the Windows System File functions to edit .INI files.)

Most Easy System File functions return a DOS error code if unsuccessful. Refer to [DOS Error Codes](#) for a list of the DOS errors that may be returned.

NOTE:

Prior to using any of the functions in this category, you must call SETSYSFILE. Also, none of the functions will create a backup of the file that they are modifying; however, a file will not be modified if a function fails. It is your responsibility to backup any file as necessary.

All of the Easy System File functions make use of a "key" value. This value is used to search the file to aid in determining where to make a modification. All key searches are *case insensitive*. If a key is found, its corresponding value is defined as the first non- whitespace (e.g. tab, cr/lf, =, etc.) group of characters after the found key value. For example, consider the following line:

```
PATH=C:\DOS;C:\WINDOWS
```

If "PATH" is specified as the key, then the corresponding value is "C:\DOS;C:\WINDOWS." However, consider the following line:

```
STACKS 9,256
```

If "STACKS" is specified as the key, then the corresponding value is "9,256." As a result, an equal sign is not necessary to identify a value that you might want to edit.

NOTE:

In the following function specifications, parameters in quotes represent literal parameters; all other parameters represent rules. For more information, refer to the list of [Rules and System Variables](#).

See Also

[ADDDEVICE](#)

[ADDLINE](#)

[ADDPATH](#)

[CFGGETVALUE](#)

[CFGSETVALUE](#)

[CFGGETSTRING](#)

[CFGSETSTRING](#)

[REPLACEKEY](#)

[REPLACELINE](#)

REPLACELINEADD
SETSYSFILE

ADDDEVICE [strvalue1] [strvalue2] [addopt]

Parameter	Description and Notes
[strvalue1]	The path and driver name (e.g. C:\WINDOWS\EMM386.EXE).
[strvalue2]	The key value to search for (e.g. HIMEM.SYS).
[addopt]	Where [strvalue1] is to be placed: either BEFORE or AFTER [strvalue2].

Description - Adds a new DEVICE= line to a system file (typically the DOS CONFIG.SYS).

Tip - If [strvalue2] is a null string or the key is not found, ADDDEVICE will add [strvalue1] in the position of the file indicated by [addopt].

Return Values:

[RETVAL] = 0 if successful

[RETVAL] = DOS error code in all other cases

Example - Place "DEVICE=C:\WINDOWS\EMM386.EXE" after the "DEVICE=HIMEM.SYS" line in the CONFIG.SYS file:

```
SETSYSFILE "C:\" "CONFIG.SYS"
```

```
ADDDEVICE "C:\WINDOWS\EMM386.EXE" "HIMEM.SYS" AFTER
```

ADDLINE [strvalue1] [strvalue2] [addopt]

Parameter	Description and Notes
[strvalue1]	The entire line of text you wish to add.
[strvalue2]	A reference key value to be positioned relative to [strvalue1]. This is a "keyword" that will be searched for in the file. Specify as much or as little as you like. When the first occurrence of the keyword in a line is found, that line is used as the reference.
[addopt]	Specify where [strvalue1] is to be placed: either BEFORE or AFTER [strvalue2].

Description - Adds a line of text to a system file.

Tip - If [strvalue2] is a null string, ADDLINE will place [strvalue1] in the position of the file indicated by [addopt].

Return Values:

[RETVAL] = 0 if successful

[RETVAL] = DOS error code in all other cases

Example - Add a new line to the end of a user's CONFIG.SYS file:

```
SETSYSFILE "C:\\" "CONFIG.SYS"  
ADDLINE "THIS IS NEW.." "" AFTER
```

NOTE:

As in the example above, non-specified parameters (e.g., [strvalue2]) can be indicated by empty quotes. Entering NULL with no quotes is also acceptable.

ADDPATH [strvalue1] [strvalue2] [strvalue3] [addopt]

Parameter	Description and Notes
[strvalue1]	The name of the path environment variable to edit (PATH for DOS, or DPATH for OS/2, or TEMP, etc).
[strvalue2]	The sub-directory to be added.
[strvalue3]	The sub-directory that [strvalue2] will be placed either before or after.
[addopt]	Specify where [strvalue2] is to be placed: either BEFORE or AFTER [strvalue3].

Description - Adds a sub-directory to a path environment variable.

Tips:

- 1) If [strvalue3] is a null string, ADDPATH will place [strvalue2] in the position of the path statement indicated by [addopt] (i.e., the new path will be placed at the beginning or end of the path statement).
- 2) If the key specified in [strvalue1] is not found, then a new one is added, with a "SET" prepended. This allows for adding path like environment variables such as "SET TEMP=", and so on.
- 3) This function can also be used to edit other lines such as a TEMP environment variable, or any other line that does something like "SET envvar=d:\path."

Return Values:

[RETVAL] = 0 if successful

[RETVAL] = DOS error code in all other cases

Example - Add the sub-directory WINDOWS to the path and place it before the DOS variable in the AUTOEXEC.BAT file:

```
SETSYSFILE "C:\ " "AUTOEXEC.BAT"  
ADDPATH "PATH" "C:\WINDOWS" "C:\DOS" BEFORE
```

CFGGETVALUE [strvalue] [intvar]

Parameter	Description and Notes
[strvalue]	The variable to be retrieved.
[intvar]	An integer variable to hold the retrieved value. (Before being used as a parameter, this variable must be defined using the DEFINE function.)

Description - Gets a numeric variable from a system file (e.g., FILES, BUFFERS, etc.).

Tip - If the value of the key specified is non-numeric (e.g., the DOS=HIGH), CFGGETVALUE sets parameter 2 to 0, but does not return an error code. Use CFGGETSTRING to get a string value.

Return Values:

[RETVAL] = 0 if successful

[RETVAL] = -2 if the key value could not be found

[RETVAL] = DOS error code in all other cases

Example - Place the value of the FILES= statement in the CONFIG.SYS file into a user defined variable called nRESULT (which must first be defined!):

```
DEFINE "nRESULT" INTEGER

SETSYSFILE "C:\" "CONFIG.SYS"

CFGGETVALUE "FILES" nRESULT
```

CFGSETVALUE [strvalue] [intvalue]

Parameter	Description and Notes
[strvalue]	The variable to be set.
[intvalue]	The integer value.

Description - Sets a numeric variable in a system file (e.g., FILES, BUFFERS, etc.).

Tip - Use ADDLINE to add a new statement if one does not exist.

Return Values:

[RETVAL] = 0 if successful

[RETVAL] = -2 if the key value could not be found

[RETVAL] = DOS error code in all other cases

Example - Set the value of the FILES= statement in the CONFIG.SYS file to 50, provided a FILES= statement already exists in the file:

```
SETSYSFILE "C:\" "CONFIG.SYS"
```

```
CFGSETVALUE "FILES" 50
```

CFGGETSTRING [strvalue] [strvar]

This function acts exactly the same as CFGGETVALUE, except it deals with string values rather than integer values. An administrator might use this to check non-numeric variables (e.g., STACKS=9,256 is a non numeric value).

Note that before using the [strvar] variable as a parameter, the variable must be defined using the DEFINE function.

CFGSETSTRING [strvalue1] [strvalue2]

This function acts exactly the same as CFGSETVALUE, except it deals with string values rather than integer values. An administrator might use this to check non-numeric variables (e.g., STACKS=9,256 is a non numeric value).

Note that before using the [strvar] variable as a parameter, the variable must be defined using the DEFINE function.

REPLACEKEY [strvalue1] [strvalue2] [strvalue3]

Parameter	Description and Notes
[strvalue1]	The line in the system file which contains the key value to be replaced.
[strvalue2]	The key value to be replaced.
[strvalue3]	The new value.

Description - Similar to REPLACELINE; however, it replaces a key value rather than the entire line.

Tip - If [strvalue3] is a null string, [strvalue2] will be removed.

Return Values:

[RETVAL] = 0 if successful

[RETVAL] = DOS error code in all other cases

Example - Change the "40" to a "50" in the FILES= line in the CONFIG.SYS file:

```
SETSYSFILE "C:\\" "CONFIG.SYS"  
REPLACEKEY "FILES=40" "40" "50"
```


REPLACELINE [strvalue1] [strvalue2]

Parameter	Description and Notes
[strvalue1]	The key value of the line you wish to replace, such as PATH, COMSPEC or DEVICE.
[strvalue2]	The new value of the entire line.

Description - Replaces an existing line in a system file with a new line.

Tips:

- 1) If [strvalue2] is a null string, then the line will be deleted.
- 2) If the key value exists more than one time in the file, only the first instance is modified.

Return Values:

[RETVAL] = 0 if successful

[RETVAL] = DOS error code in all other cases

Example - Replace the existing COMSPEC line in the CONFIG.SYS file with a new line:

```
SETSYSFILE "C:\" "CONFIG.SYS"
```

```
REPLACELINE "COMSPEC" "SET COMSPEC=C:\DRDOS\COMMAND.COM"
```

REPLACELINEADD [strvalue1] [strvalue2] [adopt]

Parameter	Description and Notes
[strvalue1]	The key value of the line you wish to replace, such as PATH, COMSPEC or DEVICE.
[strvalue2]	The new value of the entire line.
[adopt]	Where [strvalue1] is to be placed: either BEFORE or AFTER [strvalue2].

Description - Similar to REPLACELINE, this function replaces an existing line in a system file with a new line. However, if the key specified in [strvalue1] is not found, then the line specified in [strvalue2] is added to the file, at the beginning or end of the file depending on the position defined by [adopt].

Tip: If [strvalue1] is not found, then the line specified as [strvalue2] will be added to the file in the position defined by [adopt].

Return Values:

[RETVAL] = 0 if successful

[RETVAL] = DOS error code in all other cases

Example - Replace the existing NETX line with the new line C:\NET\VLM. If NETX is not found, then the line will be appended to the end of the file:

```
SETSYSFILE "C:\" "NET.BAT"
```

```
REPLACELINEADD "NETX" "C:\NET\VLM" AFTER
```

SETSYSFILE [path] [filename]

Parameter	Description and Notes
[path]	The path to the file to be modified.
[filename]	The name of the file to be modified.

Description - Sets a system file to be manipulated.

Tips:

- 1) This function must be called prior to calling any of the functions in the Easy System File function category. It needs to be called only once, unless you change the file you are working on in the script.
- 2) Using [BOOT_ROOT] as the [path] parameter will *always* modify the file on the boot disk, regardless of whether or not the user is given the option to override the installation path (in the package definition). Use [TARGET] as the [path] parameter if the user is given the option to override the installation path.

Return Values:

[RETVAL] = 0 if file is found

[RETVAL] = 2 if file is not found

Example - Designate a user's CONFIG.SYS file as the file to be edited. Two examples of this are:

```
SETSYSFILE "C:\" "CONFIG.SYS"
```

or

```
SETSYSFILE [BOOT_ROOT] "CONFIG.SYS"
```

Windows System File Functions

The Windows System File functions provide the ability to edit INI files and create and manipulate Program Manager groups.

NOTES:

- a - In the following function specifications, parameters in quotes represent literal parameters; all other parameters represent rules. For more information, refer to the list of Rules and System Variables.*
- b - Many of the Windows System File functions have a [pathfile] parameter which specifies the path name and file name to an INI file. If you do not specify a full path to the Windows directory, then the actions performed by these functions occur on the first instance of Windows found, as determined by the path statement of the receiving machine. If Windows is not found in the path, then the distribution update program will search for the INI file in [BOOT_ROOT]\WINDOWS. If Windows is still not found, the update program will then try [BOOT_ROOT]\WIN31.*
- c - The functions ADDGROUP, ADDITEM, and SCHEDULEWIN use the WSDUPD.EXE update program which is copied into the local Windows directory each time these functions are used. The next time the user runs Windows, WSDUPD.EXE runs and executes the appropriate function(s). It then deletes WSDUPD.EXE and WSDUPD.INI. If a user has SHARE.EXE loaded, a "sharing violation" message will display when trying to delete WSDUPD.EXE. This message can be ignored.*

See Also

ADDGROUP

ADDITEM

GETINIINT

GETINISTR

SCHEDULEWIN

WRITEINIINT

WRITEINISTR

ADDGROUP [strvalue]

Parameter	Description and Notes
[strvalue]	The string which specifies the name of the Program Manager group to be added.

Description - Creates a new Program Manager group.

Tip: When the ADDGROUP script function is executed, the BrightWorks software distribution update program WSDUPD.EXE is automatically copied into the workstation's Windows directory. The WSDUPD.EXE command is also added to the "Load=" line in the WIN.INI file. The next time Windows is run at the workstation, the function is executed and WSDUPD.EXE is removed from the WIN.INI "Load=" line.

Return Values:

[RETVAL] = 0 if successful

[RETVAL] = DOS error code if unsuccessful. The function might fail if WSDUPD.EXE could not be copied into the Windows directory or if the WSDUPD.EXE control file (WSDUPD.INI) could not be created.

Example - Create a Program Manager group named COMPANY:

```
ADDGROUP "COMPANY"
```

NOTE:

This function can be used with any third party shell program which emulates the Program Manager DDE interface.

ADDITEM [strvalue1] [strvalue2] [pathfile]

Parameter	Description and Notes
[strvalue1]	The group to which the item will be added.
[strvalue2]	The name of the new item.
[pathfile]	The .EXE file to be associated with the new item.

Description - Adds a new item to a Program Manager group.

Tip: When the ADDITEM script function is executed, the BrightWorks software distribution update program WSDUPD.EXE is automatically copied into the workstation's Windows directory. The WSDUPD.EXE command is also added to the "Load=" line in the WIN.INI file. The next time Windows is run at the workstation, the function is executed and WSDUPD.EXE is removed from the WIN.INI "Load=" line.

Return Values:

[RETVAL] = 0 if successful

[RETVAL] = DOS error code if unsuccessful. The function might fail if WSDUPD.EXE could not be copied into the Windows directory or if the WSDUPD.EXE control file (WSDUPD.INI) could not be created.

Example - Create a Program Manager group named APPS, and then create a program icon within the new APPS group named EXCEL:

```
ADDGROUP "APPS"
```

```
ADDITEM "APPS" "EXCEL" "U:\MS\EXCEL\EXCEL.EXE"
```

NOTE:

This function can be used with any third party shell program which emulates the Program Manager DDE interface.

GETINIINT [pathfile] [strvalue1] [strvalue2] [intvar]

Parameter	Description and Notes
[pathfile]	The path and file name of the INI file.
[strvalue1]	The section of the INI file in which the entry is located (e.g., [386Enh]).
[strvalue2]	The entry whose associated string is to be retrieved (e.g., keyboard.drv=, however, do not include the = sign!).
[intvar]	Variable in which to place the found integer. (Before being used as a parameter, this variable must be defined using the DEFINE function.)

Description - Gets a key value (integer) from an INI file, and places the result in a variable.

Return Values:

[RETVAL] = 0 if successful

[RETVAL] = -1 if the [strvalue2] section name does not exist

[RETVAL] = -2 if the [strvalue3] key does not exist

[RETVAL] = DOS error code in all other cases

GETINISTR [pathfile] [strvalue1] [strvalue2] [strvar]

Parameter	Description and Notes
[pathfile]	The path and file name of the INI file.
[strvalue1]	The section of the INI file in which the entry is located (e.g., [386Enh]).
[strvalue2]	The entry whose associated string is to be retrieved (e.g., keyboard.drv=, however, do not include the = sign!).
[strvar]	Variable in which to place the found string. (Before being used as a parameter, this variable must be defined using the DEFINE function.)

Description - Gets a key value (string) from an INI file, and places the result in a variable.

Return Values:

[RETVAL] = 0 if successful

[RETVAL] = -1 if the [strvalue2] section name does not exist

[RETVAL] = -2 if the [strvalue3] key does not exist

[RETVAL] = DOS error code in all other cases

Example - Determine whether Windows version 3.1 is installed at a workstation by looking at the CONTROL.INI file:

```
DEFINE "VER" STRING
```

```
GETINISTR "C:\WIN\CONTROL.INI" "[INSTALLED]" "3.1" VER
```


SCHEDULEWIN [path] [filename] [text]

Parameter	Description and Notes
[path]	The path to the file to be run.
[filename]	The file name to be run upon Windows execution.
[text]	Optional command line arguments for the file.

Description - Schedules a file to be run the next time the user runs Windows.

Tip - This function could be used to automate the installation of a Windows program if a macro playback utility is used.

Return Values:

[RETVAL] = 0 if successful

[RETVAL] = DOS error code if unsuccessful. The function might fail if WSDUPD.EXE could not be copied into the Windows directory or if the WSDUPD.EXE control file (WSDUPD.INI) could not be created.

Example - Schedule the Notepad program to run the next time Windows is run, and also open the README.TXT notepad file:

```
SCHEDULEWIN "C:\WINDOWS" "NOTEPAD.EXE" "README.TXT"
```

WRITEINIINT [pathfile] [strvalue1] [strvalue2] [intvalue]

Parameter	Description and Notes
[pathfile]	The path and file name of the INI file.
[strvalue1]	The section in which [strvalue2] is located (e.g., [386Enh]).
[strvalue2]	The entry whose associated string is to be modified (e.g., keyboard.drv=, however, don't include the = sign!).
[intvalue]	The integer to be written to the INI file.

Description - Gets a key value (integer) from an INI file, and writes the result to the INI file.

Tips:

1) If the section name specified in [strvalue1] is not found, then it will be added to the end of the INI file, with a new key=value added in that section.

2) If the [strvalue1] section is found but the key value specified in [strvalue2] is not found, the new key value is added directly after the section name [strvalue1].

Return Values:

[RETVAL] = 0 if successful

[RETVAL] = DOS error code in all other cases

WRITEINISTR [pathfile] [strvalue1] [strvalue2] [strvalue3]

Parameter	Description and Notes
[pathfile]	The path and file name of the INI file.
[strvalue1]	The section in which [strvalue2] is located (e.g., [386Enh]).
[strvalue2]	The entry whose associated string is to be modified (e.g., keyboard.drv=, however, don't include the = sign!).
[strvalue3]	The string to be written to the INI file.

Description - Gets a key value (string) from an INI file, and writes the result to the INI file.

Tips:

1) If the section name specified in [strvalue1] is not found, then it will be added to the end of the INI file, with a new key=value added in that section.

2) If the [strvalue1] section is found but the key value specified in [strvalue2] is not found, the new key value is added directly after the section name [strvalue1].

Return Values:

[RETVAL] = 0 if successful

[RETVAL] = DOS error code in all other cases

Example - Define a "medium priority" in the [SPOOLER] section of the WIN.INI file:

```
WRITEINISTR "C:\WIN\WIN.INI" "[SPOOLER]" "PRIORITY" "MEDIUM"
```

Miscellaneous Functions

The Miscellaneous Functions include basic functions for defining, assigning, copying, comparing and concatenating variables.

NOTE:

In the following function specifications, parameters in quotes represent literal parameters; all other parameters represent rules. For more information, refer to the list of [Rules and System Variables](#).

See Also

APPENDPATH

ASSIGN

DEFINE

EXIT

IF

ELSE

ENDIF

NUMTOSTR

PAUSE

REBOOT

SHELL

STRCAT

STRCOMPARE

STRCOPY

WRITELN

APPENDPATH [strvar] [strvalue]

Parameter	Description and Notes
[strvar]	The variable to contain the appended string (i.e., destination). (Before being used as a parameter, this variable must be defined using the DEFINE function.)
[strvalue]	The string value to be appended (i.e., source).

Description - Adds a file name to a path or builds a path. This function acts the same way as STRCAT, except that it will check if the last character of [strvar] is a "\". If it is not, APPENDPATH will append a "\" to [strvar], and then [strvalue] will be appended. This is very useful (and necessary!) in building paths.

Return Value:

[RETVAL] = 0 always

Example - Define the variable named PATH to be a string-type. Copy the location of the network configuration files into the PATH variable and then append it to the C:\DRIVERS directory.

```
DEFINE "PATH" STRING  
  
STRCOPY PATH [NETCFG]  
  
APPENDPATH "C:\DRIVERS" PATH
```

ASSIGN [intvar] [intvalue]

Parameter	Description and Notes
[intvar]	The integer type variable name which will be assigned a value. (Before being used as a parameter, this variable must be defined using the DEFINE function.)
[intvalue]	The numeric value to be assigned to the integer type variable.

Description - Performs a basic integer assignment operation (e.g., a = b).

Return Value:

[RETVAL] = 0 always

Example - Define the variable "NUM" as an integer type, and later assign 33 to the variable NUM:

```
DEFINE "NUM" INTEGER
ASSIGN NUM 33
```

DEFINE [text] [defineopt]

Parameter	Description and Notes
[text]	The variable being defined.
[defineopt]	The type of variable being defined (e.g., STRING or INTEGER).

Description - Used to create user defined variables of a string or integer type. This variable can then be used later in the script.

Tips:

- 1) All DEFINE statements must be declared before any script command is executed.
- 2) If a STRING type variable is declared, the login module will allocate 255 bytes (= 255 characters) of memory for the string. If an INTEGER type variable is declared, the login module will allocate 4 bytes (C type long which equals to an approximately -2 billion to +2 billion size integer).

Return Value:

[RETVAL] = 0 always

Example - Define the variable "ANSWER" as a string type.

```
DEFINE "ANSWER" STRING
```

EXIT [intvalue]

Parameter	Description and Notes
[intvalue]	An integer type variable.

Description - Ends the script.

Tips:

1) If [intvalue] is set to a non-zero value, then the login module will increment the error count by one for the upgrade package and note the log with the error number returned.

2) If the package has been defined to execute the script before decompressing the fileset, then the EXIT command will prevent the decompression of the fileset. (For more information, refer to [Advanced Package Options.](#))

Return Value: none

Example - End the script if an obtained value is greater than 50:

```
IF RESULT <= 50

    CFGSETVALUE "FILES" 55

ELSE

    EXIT 1

ENDIF
```


IF [intvalue1] [condoper] [intvalue2] ... {ELSE...} ENDIF

Parameter	Description and Notes
[intvalue1]	An integer type variable to be evaluated against [intvalue2].
[condoper]	Valid conditional operators are: =, !=, <, >, <=, >=
[intvalue2]	An integer type variable to evaluate [intvalue1] against.

Description - Allows conditional processing of functions. IF..ELSE evaluates the conditional expression defined by [intvalue1] [condoper] [intvalue2]. If the condition evaluates to be TRUE, then all functions following ELSE are executed until an ELSE or ENDIF is reached. If the condition evaluates to FALSE and ELSE is defined, then all functions following the ELSE are executed until an ENDIF is reached.

Tip - IFs can be nested up to 50 levels deep.

Return Value: none

Example - Obtain the FILES= value from the CONFIG.SYS file. If the value is less than or equal to 50, then change the value to 55; otherwise, exit the script:

```
DEFINE "RESULT" INTEGER

SETSYSFILE "C:\" "CONFIG.SYS"

CFGGETVALUE "FILES" RESULT

IF RESULT <= 50

    CFGSETVALUE "FILES" 55

ELSE

    EXIT 1

ENDIF
```

IF [intvalue1] [condoper] [intvalue2] ... {ELSE...} ENDIF

Parameter	Description and Notes
[intvalue1]	An integer type variable to be evaluated against [intvalue2].
[condoper]	Valid conditional operators are: =, !=, <, >, <=, >=
[intvalue2]	An integer type variable to evaluate [intvalue1] against.

Description - Allows conditional processing of functions. IF..ELSE evaluates the conditional expression defined by [intvalue1] [condoper] [intvalue2]. If the condition evaluates to be TRUE, then all functions following ELSE are executed until an ELSE or ENDIF is reached. If the condition evaluates to FALSE and ELSE is defined, then all functions following the ELSE are executed until an ENDIF is reached.

Tip - IFs can be nested up to 50 levels deep.

Return Value: none

Example - Obtain the FILES= value from the CONFIG.SYS file. If the value is less than or equal to 50, then change the value to 55; otherwise, exit the script:

```
DEFINE "RESULT" INTEGER

SETSYSFILE "C:\" "CONFIG.SYS"

CFGGETVALUE "FILES" RESULT

IF RESULT <= 50

    CFGSETVALUE "FILES" 55

ELSE

    EXIT 1

ENDIF
```

IF [intvalue1] [condoper] [intvalue2] ... {ELSE...} ENDIF

Parameter	Description and Notes
[intvalue1]	An integer type variable to be evaluated against [intvalue2].
[condoper]	Valid conditional operators are: =, !=, <, >, <=, >=
[intvalue2]	An integer type variable to evaluate [intvalue1] against.

Description - Allows conditional processing of functions. IF..ELSE evaluates the conditional expression defined by [intvalue1] [condoper] [intvalue2]. If the condition evaluates to be TRUE, then all functions following ELSE are executed until an ELSE or ENDIF is reached. If the condition evaluates to FALSE and ELSE is defined, then all functions following the ELSE are executed until an ENDIF is reached.

Tip - IFs can be nested up to 50 levels deep.

Return Value: none

Example - Obtain the FILES= value from the CONFIG.SYS file. If the value is less than or equal to 50, then change the value to 55; otherwise, exit the script:

```
DEFINE "RESULT" INTEGER

SETSYSFILE "C:\" "CONFIG.SYS"

CFGGETVALUE "FILES" RESULT

IF RESULT <= 50

    CFGSETVALUE "FILES" 55

ELSE

    EXIT 1

ENDIF
```

NUMTOSTR [strvar] [intvalue]

Parameter	Description and Notes
[strvar]	The variable to contain the converted value. (Before being used as a parameter, this variable must be defined using the DEFINE function.)
[intvalue]	The numeric value to be converted.

Description - Converts a numeric value to a string variable.

Return Value:

[RETVAL] = 0 always

Example - Convert the number 100 to a string and store the value in the defined variable named ONEHUNDRED:

```
DEFINE "ONEHUNDRED" STRING  
  
NUMTOSTR ONEHUNDRED 100
```

PAUSE [text]

Parameter	Description and Notes
[text]	The text to be displayed on the user's screen during the pause. (This can be NULL)

Description - Pauses execution of the script until the user presses a key.

Tip - If [text]1 is NULL, then the default message "Strike any key to continue" is displayed on the screen.

Return Value = 0 always

Example - Display the message "Pausing... press any key to continue" during script execution.

```
PAUSE "PAUSING ... PRESS ANY KEY TO CONTINUE."
```

REBOOT

This function immediately reboots the user's PC. It does not accept any parameters and does not return any values. Before the reboot, the script file is closed, the log database is closed, and any necessary cleanup is performed.

NOTES:

a - The PC will not reboot if a fileset is to be executed after the script.

b - The REBOOT function might not work if the workstation is not 100% PC compatible.

SHELL [pathfile] {text} {shellopt}

Parameter	Description and Notes
[pathfile]	The path and file name to execute.
{text}	The file's optional command line arguments. (This can be NULL.)
{shellopt}	<i>Optional</i> argument which can only be either [KEEPPATH] or NULL.

Description - Allows a user to execute an external DOS batch file, executable program, or DOS command.

Tip - To execute the program or batch file in [pathfile] and change to the specified path, use the KEEPPATH option as the {shellopt} parameter. If you don't specify the KEEPPATH option, SHELL will try to use the path from which the SDUPDATE program was run. KEEPPATH allows you to temporarily switch to the path from where you want to run the program.

Return Values:

[RETVAL] = 0 if successful

[RETVAL] = -1 if failed

Example - Execute LIST.COM and load the contents of the README.TXT file. Temporarily make the current directory C:\PUB\LIST.COM.

```
SHELL "C:\PUB\LIST.COM" "README.TXT" "" KEEPPATH
```

STRCAT [strvar] [strvalue]

Parameter	Description and Notes
[strvar]	The variable to contain the concatenated string (i.e., destination). (Before being used as a parameter, this variable must be defined using the DEFINE function.)
[strvalue]	The string value to be appended (i.e., source).

Description - Appends the contents of [strvalue] to the end of the string [strvar].

Tip - If the resulting text in [strvar] is longer than the space allowed (255 bytes), then it will be truncated and properly null terminated.

Return Value:

[RETVAL] = 0 always

Example - Add the string "ADDTHIS" to a string variable named STRINGS1&2:

```
DEFINE "STRINGS1&2" STRING  
  
STRCAT STRINGS1&2 "ADDTHIS"
```


STRCOMPARE [strvar] [strvalue]

Parameter	Description and Notes
[strvar]	The variable to be compared. (Before being used as a parameter, this variable must be defined using the DEFINE function.)
[strvalue]	The value to compare the variable against.

Description - Does a byte for byte comparison of two strings.

Return Values:

[RETVAL] = 0 if the strings are identical

[RETVAL] = < 0 if [strvar] is less than [strvalue]

[RETVAL] = > 0 if [strvar] is greater than [strvalue]

Example - Check the current NetWare login name against a specified login name ("Supervisor").

```
DEFINE "NAME" STRING  
  
STRCOPY NAME [LOGINNAME]  
  
STRCOMPARE NAME "SUPERVISOR"
```

STRCOPY [strvar] [strvalue]

Parameter	Description and Notes
[strvar]	The variable to receive the copied string value (i.e., destination). (Before being used as a parameter, this variable must be defined using the DEFINE function.)
[strvalue]	The string value to be copied (i.e., source).

Description - Copies a value into a string, overwriting the previous contents of the string.

Return Value:

[RETV] = 0 always

Example - Copy the string "ABC" into the string variable named "HOLDABC":

```
DEFINE "HOLDABC" STRING  
  
STRCOPY HOLDABC "ABC"
```

WRITELN [strvalue]

Parameter	Description and Notes
[strvalue]	The string to display on screen.

Description - Writes the [strvalue] line to stdout (e.g., the screen). This might be useful for displaying error messages, etc.

Return Value:

[RETVAL] = 0 always

Example - Define the variable named RESULT. Place the value of the FILES= statement in the CONFIG.SYS file into RESULT, and then write the value of RESULT.

```
DEFINE "VALUE" STRING
DEFINE "RESULT" INTEGER
SETSYSFILE "C:\" "CONFIG.SYS"
CFGGETVALUE "FILES" RESULT
NUMTOSTR VALUE RESULT
WRITELN VALUE
```

Rules and System Variables

Most of the functions in the BrightWorks script language have parameters that are specified or passed to them. The valid entries for each parameter type are called *rules*. For example, the UPGRADEOS function has one parameter named [upgopt]. As indicated in the table below, the value of the [upgopt] parameter can be either 5.00 or 6.00. Therefore, the allowable values for the [upgopt] parameter are 5.00 and 6.00.

NOTE:

When a user defined variable of string type is expected, [STRVAR] is the rule. When a user defined variable of integer type is expected, [INTVAR] is the rule.

The table below lists the rules (allowable values) for each parameter.

Rule Name	Allowed Values
ADDOPT	BEFORE, AFTER
ATTRIBUTE	RO, RW, A, SY, H, SH, -A, -SY, -H, -SH
CONDOPER	< > =, !=, >=, <=
DEFINEOPT	STRING, INTEGER
DELETOPT	ALL
FILENAME	[STRVAR], "filename.ext" (wild cards not allowed for file name)
FILEWILD	[STRVAR], "filename.ext" "*" "*" (wild cards are allowed but not required for a file name)
INTVALUE	[INTVAR], [RETVAR], # (where # is a valid integer)
INTVAR	[INTVAR]
PATH	[STRVAR], "path", [TARGET], [BOOT_ROOT], [WINDIR], [WINSYSDIR], [NETCFG], [HDRIVE], [NDRIVE], [SERVERNAME], [LOGINNAME], [FUSIONNAME], [LOGSCRNAME]
PATHFILE	[STRVAR], "{path}\filename.ext"
SHELLOPT	KEEPPATH
STRVAR	[STRVAR]
STRVALUE	[STRVAR], "text", [TARGET], [BOOT_ROOT], [WINDIR], [WINSYSDIR], [NETCFG], [HDRIVE], [NDRIVE], [SERVERNAME], [LOGINNAME], [FUSIONNAME], [LOGSCRNAME]
TEXT	"text"
UPGOPT	5.00, 6.00

System Variables

The rules listed in the above table are defined as follows:

String Type Rules:

[BOOT_ROOT] - the root of the boot drive of the workstation on which the script is executed

[HDRIVE] - drive letter of the first available hard drive (may be boot or network drive)

[FUSIONNAME] - primary user name from BrightWorks databases (generally same as LOGINNAME)

[LOCATION] - location field from BrightWorks inventory databases

[LOGINNAME] - login name of user

[LOGSCRNAME] - full path and file name of login script for user running update.

[NDRIVE] - drive letter of the first available network drive

[NETCFG] - path to NET.CFG used at NetWare shell load (must be in path)

[SERVERNAME] - name of server on which the update program is running

[TARGET] - installation path as defined by the administrator (or changed by user, if able to)

[WINDIR] - the user's Windows directory (directory in which the login module finds WIN.INI - this directory must be in the path)

[WINSYSDIR] - the user's Windows\System directory (directory in which the login module finds USER.EXE - this directory may be in the SYSTEM directory below WINDIR, or in the path)

Integer Type Rules:

[DISKSPACE] - available disk space in drive specified in [TARGETDIR]. The number is in bytes.

[RETVAl] - return code of last command completed

DOS Error Codes

The following table lists the DOS error codes that may be returned from the script functions.

#	Message	Reason for Error	Action	Functions that Return the Error
2	File not found	A file specified in the script does not exist.	Check the filename and path.	GETINISTR() - The file from which you requested a string does not exist. GETINIINT() - The file from which you requested an integer does not exist.
3	Path not found	A directory path specified in the script does not exist.	Check the path and directory name.	DELETEDIR() - The directory that you requested to delete does not exist, or it does not exist in the location you specified.
4	Too many open files (no handles left)	Insufficient file handles specified in CONFIG.SYS	Increase the number of file handles in CONFIG.SYS.	COPY() All Easy System File and Windows System File functions.
5	Access denied	Unable to access the specified drive or file. Insufficient user rights, read only files, disk full.	Verify the user rights, file attributes and available disk space.	DELETEFILE(), ADDPATH(), ADDLINE(), REPLACELINE(), REPLACEKEY(), ADDDEVICE(), CFGSETVALUE(), CFGSETSTRING(), REPLACELINEADD(), WRITEINISTR(), WRITEINIINT() - Is the file flagged read only? Is the disk full? Does the end user have insufficient rights in the specified directory? DELETEDIR() - Subdirectories and/or files exist, and the "ALL" option was not used in the script. ADDGROUP(), ADDITEM(), SCHEDULEWIN() - Is WIN.INI flagged read only?
8	Insufficient memory	Not enough memory to complete the specified action.	Unload unnecessary TSRs, check workstation memory management.	All DOS, Easy System File and Windows System File functions.
15	Invalid drive	The drive specified does not exist.	Check the drives specified in the script.	All DOS, Easy System File and Windows System File functions.
16	Attempt to remove current directory	The directory you attempted to delete is active on a drive.		DELETEDIR() - Is the directory specified active on the drive? If it is a network drive, are other users active on the drive?
17	Not same device	An action was specified on two separate drives.	Ensure that you are not attempting to "cross drives" on an action that does not permit this (e.g., RENAME)	RENAME() - Are the source and target locations different?
18	No more files	The specified file could not be found.	Check the path and filename. Check end user rights in the directory specified.	DELFILE(), ATTRIB(), RENAME(), SETSYSFILE(), COPY() - Does the specified file exist in the location specified? Does the end user have sufficient rights to see the file?
19	Disk is write protected	The write protect tab is enabled on the disk specified in the operation.	Remove the write protect tab.	All DOS, Easy System File and Windows System File functions.
21	Drive not ready	There is no disk in the drive specified in the operation.	Insert the diskette.	All DOS, Easy System File and Windows System File functions.
22	Invalid disk command	Media access error.	Check the diskette or drive.	Bad or damaged diskette.
23	CRC error	Media access error.	Check the diskette	Bad or damaged diskette.

24	Invalid length	Media access error.	or drive. Check the diskette or drive.	Bad or damaged diskette.
25	Seek error	Media access error.	Check the diskette or drive.	Bad or damaged diskette.
27	Sector not found	Media access error.	Check the diskette or drive.	Bad or damaged diskette.
29	Write fault	Media access error.	Check the diskette or drive.	Bad or damaged diskette.
30	Read fault	Media access error.	Check the diskette or drive.	Bad or damaged diskette.
31	General failure	Media access error.	Check the diskette or drive.	Diskette may not be formatted.

Introduction to Scopes

A scope is a group of workstations defined to receive a distributed package. Defining a scope is as easy as assigning a name to the new scope. After the scope is created, any number of workstations can be included in the scope definition.

Access to Scope Functions

Scope functions are accessed by choosing the Scopes command from the Tools menu. The Scopes dialog box displays listing all available scopes.

Scope Features

By taking advantage of the database of inventory information maintained by BrightWorks, users can create scopes by selecting from nodes that match specific filtering criteria. The filtering criteria is saved as a "query" and then applied against the database to narrow down the list of applicable workstations. Scope membership is subsequently assigned using the list of nodes that match the user-specified filtering criteria. Items such as CPU speed, workstation memory and installed software can be used to accommodate a scope's intent. For example, a scope named CPU386 might consist of the network's 386 workstations; a scope named 386>16MHz might consist of the network's 386 workstations that also have a CPU clock frequency greater than 16 MHz.

Scopes and queries can be stored, used and reused as resources within BrightWorks. A user can create a new scope, as well as edit, copy, rename and delete a scope. Refer to the following topics:

[Creating Scopes](#)

[Editing Scopes](#)

[Renaming Scopes](#)

[Copying Scopes](#)

[Deleting Scopes](#)

[Scope Queries](#)

Creating Scopes

Use the following procedure to create a new scope.

NOTE:

At least one audit must have been run in order to create a scope. For detailed instructions, refer to [Running an Audit](#).

1. Choose the Scopes command from the Tools menu.

The Scopes dialog box displays listing the names of all defined scopes.

2. Choose the New button.

The New Scope dialog box displays prompting you to enter a name for the new scope.

3. Enter the new scope name, and choose the OK button.

A scope name can be up to 80 characters, and all typed characters are valid. For example, enter the new scope name "386/16MHZ" which will include the network's 386/16MHz nodes.

Upon choosing the OK button, the Edit Scope dialog box displays prompting you to define the new scope.

The Edit Scope dialog box consists of two lists:

- * **Nodes in SITE** - the list on the left side of the dialog box consists of all node names that apply to the local site. The site name is indicated by the SITE text in the list title. The nodes in this list are not included in the selected scope (i.e., the list to the right).
- * **Nodes included in Scope** - the list on the right side of the dialog box consists of the nodes that are in the selected scope.

NOTE:

The Query Options section of the Edit Scope dialog box is used to perform a query to filter the node names in the Nodes in SITE list. For detailed instructions on performing queries, refer to [Scope Queries](#).

4. Define the nodes to be included in the scope.

Use the push buttons in the center of the Edit Scope dialog box to define the scope members. To define scope membership, select a node name from either list and choose the appropriate button:

- * **Include >>** - choose this button to include the selected node in the scope. The node name moves from the left list to the right list.
- * **Include All** - choose this button to include all listed nodes in the scope. All node names move from the left list to the right list.
- * **<< Remove** - choose this button to remove the selected node from the scope. The node name moves from the right list to the left list.
- * **Remove All** - choose this button to remove all nodes from the scope. All node names move from the right list to the left list.

NOTE:

Because user names can be edited via the View Inventory Details dialog box, the node names in the Nodes in SITE list do not necessarily correspond to NetWare user names. As a result, there may be duplicate names in this list. Viewing the inventory details of nodes with the same name enables you to differentiate between the nodes. Choose the View button or double click on a name in either list to

invoke the View Inventory Details dialog box for the selected node.

- 5. When the scope members are defined, choose the Save button.**

See Also:

[Editing Scopes](#)

[Renaming Scopes](#)

[Copying Scopes](#)

[Deleting Scopes](#)

[Scope Queries](#)

Scope Queries

When no filtering criteria is applied to a scope, all nodes in the local BrightWorks site are listed in the Nodes in SITE list of the Edit Scope dialog box. This condition is indicated by the <None> entry in the Last Query field. Searching through a large list of nodes might make the process of defining scope membership cumbersome. Applying a query to a scope refines the number of nodes in the Nodes in SITE list so that scope membership can be made from a "qualified" list of nodes.

NOTE:

Only one query can be applied to a scope at any time. Each query can consist of more than one filtering criteria. An applied query always filters from the entire list of nodes in the local BrightWorks site.

Queries can be saved and applied to any number of scopes. The same queries can be applied to BrightWorks inventory and distribution reports.

Refer to the following topics:

[Creating a New Query](#)

[Editing a Query](#)

[Deleting a Query](#)

[Applying a Query to a Scope](#)

[Removing a Query from a Scope](#)

Creating a New Query

Use the following procedure to create a new query. The procedure assumes that you have already chosen the Select button in the Edit Scope dialog box to display the Select Query dialog box.

NOTE:

All queries are also available when generating BrightWorks inventory and distribution reports.

1. Choose the Add button in the Select Query dialog box.

The Add Query dialog box displays. Press the <TAB> key to move from field to field within this dialog box.

2. Enter a Query Name and define a filter entry.

The purpose of each filter entry is to narrow down the list of nodes that apply to the specified criteria. If more than one filter entry is defined, the entries are "linked" using either the AND or OR relationship.

For example, assume the following two filter entries:

```
Central Processing Unit = Intel_80386
CPU Clock Frequency > 66.00 Mhz
```

If the entries are linked with the OR relationship, only the nodes that satisfy *either* criteria (i.e., all Intel 80386 machines and all machines that have a clock speed greater than 66 Mhz) are included in the database sort.

If the entries are linked with the AND relationship, the nodes that satisfy *both* criteria (i.e., all 286 machines and all machines that have a 720k floppy drive) are included in the database sort.

For each filter entry, specify the following:

- * **Query Name** - Enter a query name up to 80 characters in length.
- * **Component** - Choose a component from the BrightWorks inventory database to use as the filter basis. Select a component from the drop-down list associated with this field (e.g., Brand, Computer Model, CPU Clock Frequency).
- * **Condition** - Choose a conditional operator from the drop-down list associated with this field (e.g., equal to, less than, greater than, not equal to). 'Equal to' is the default condition.
- * **Description** - If desired, choose a description of the component. The items which automatically display in this list depend on the selected component. For example, "Intel_80386" might display if Central Processing Unit is entered in the Component field; "16.00 Mhz" might display if CPU Clock Frequency is entered in the Component field. See Note (a) below.
- * **Query Link** - Specify the relationship between the filter entries (e.g., Central Processing Unit = 80386 OR Central Processing Unit = 80486). The link options are AND and OR. See Note (b) below.

NOTES:

a - To create a query which tests for the presence of a component, leave the Description field blank. For example, to include all nodes with a hard disk, construct a query with the following entries:

```
Component =
Hard Disk #1 < >
```

In this case, the Component description is left blank, and the query results in including all nodes which have a hard disk (i.e., Hard Disk #1 does not equal blank).

b - All filter entries in a query must have the same Query Link type (e.g., all entries will be linked by AND or all entries will be linked by OR).

3. Choose the Insert button to accept the filter entry definition.

The entry is added to the Current Query List in the Edit Query dialog box.

4. If required, insert additional filter entries.

Repeat steps #2 and #3 above.

NOTE:

To add a filter entry between existing entries, first highlight the filter entry line in the Current Query list where you want the new entry to be placed. The new defined entry is placed in the highlighted position.

5. When all filter entries are defined, choose the Save button.

The query is saved and added to the Available Queries list in the Select Query dialog box. The new query can now be applied to a scope.

See Also:

[Editing a Query](#)

[Deleting a Query](#)

[Applying a Query to a Scope](#)

[Removing a Query from a Scope](#)

Editing a Query

Use the following procedure to edit the definition of an existing query. The procedure assumes that you have already chosen the Select button in the Edit Scope dialog box to display the Select Query dialog box.

1. Select a query from the Select Query dialog box, and choose the Edit button.

The Edit Query dialog box displays showing the defined filter entries for the query.

2. Modify the filter information, and choose the Save button.

To delete a filter entry, highlight the entry in the Current Query List and choose the Delete button.

To add a filter entry, specify the Component, Condition and Description, and choose the Insert button. (For detailed instructions on adding filter entries, refer to the procedures in [Creating a New Query](#).)

NOTE:

New filter entries are appended to the end of the Current Query List unless a filter entry is selected. If an existing filter entry is selected, then the new filter entry gets inserted above it when you choose the Insert button.

See Also:

[Creating a New Query](#)

[Deleting a Query](#)

[Applying a Query to a Scope](#)

[Removing a Query from a Scope](#)

Deleting a Query

Use the following procedure to delete an existing query. The procedure assumes that you have already chosen the Select button in the Edit Scope dialog box to display the Select Query dialog box.

1. **Select the query to be deleted from the Select Query dialog box, and choose the Delete button.**

A prompt displays asking you to confirm the deletion.

2. **Choose the Yes button to delete the query.**

If deleted, the query name is removed from the Available Queries list.

NOTE:

Queries that are currently applied to a scope and/or BrightWorks inventory and distribution report can be deleted.

See Also:

[Creating a New Query](#)

[Editing a Query](#)

[Applying a Query to a Scope](#)

[Removing a Query from a Scope](#)

Applying a Query to a Scope

Use the following procedure to apply an existing query to a scope. The procedure assumes that you have already chosen the Edit button in the Scopes dialog box to display the Edit Scope dialog box.

- 1. Choose the Select button in the Query Options section of the Edit Scope dialog box.**

The Select Query dialog box displays listing all defined queries.

- 2. Select the query name from the Available Queries list, and choose the Apply button.**

To select a query name, point to the query and click the left mouse button.

NOTE:

Applying a query to a scope causes a Printing Status dialog box to display while the database records are being sorted. When this occurs, nodes are being selected (i.e., the information is not being sent to the printer).

The Select Query dialog box closes, and the selected query name is placed into the Last Query field of the Edit Scope dialog box. The BrightWorks database records are sorted, and only the records that match the query's specified filter criteria will display in the Nodes in SITE list. Now you can assign scope members using a "qualified" list of nodes.

NOTE:

If you already selected nodes to be included in the scope (i.e., the nodes listed in the Nodes Included in Scope list), the nodes continue to be "included" even if they do not match the filter criteria.

See Also:

[Creating a New Query](#)

[Editing a Query](#)

[Deleting a Query](#)

[Removing a Query from a Scope](#)

Removing a Query from a Scope

Use the following procedure to remove a scope query.

1. **Choose the Select button in the Query Options section of the Edit Scope dialog box.**

The Select Query dialog box displays.

2. **Select the <None> query name, and choose the Apply button.**

The Select Query dialog box closes. All nodes in the local BrightWorks site are listed in the Nodes in SITE list of the Edit Scope dialog box.

See Also:

[Creating a New Query](#)

[Editing a Query](#)

[Deleting a Query](#)

[Applying a Query to a Scope](#)

Editing Scopes

Editing a scope may become necessary under two circumstances:

- * Existing scopes might need to be edited in order to add or delete members according to a change in a scope's intent.
- * Scopes that are attached to packages might need to be edited when re-sending packages.

Scope *members* are the nodes that are defined as a group to receive a distributed package.

NOTE:

An existing scope can be edited even if the scope is part of a scheduled package. This is useful if you need to re-send a package to a node(s). If new nodes are added to a scope that is included in an active package, then the package will automatically get distributed to the new nodes.

Use the following procedure to edit a scope. The procedure assumes that you have already chosen the Scopes command from the Tools menu to display the Scopes dialog box.

1. Select the scope from the list of Scopes, and choose the Edit button.

A scope can also be selected for editing by double clicking on the scope name in the Scopes dialog box. The Edit Scope dialog box displays.

The Edit Scope dialog box consists of two lists:

- * **Nodes in SITE** - the list on the left side of the dialog box consists of all node names that apply to the local site. The site name is indicated by the SITE text in the list title. The nodes in this list are not included in the selected scope (i.e., the list to the right).
- * **Nodes included in Scope** - the list on the right side of the dialog box consists of the nodes that are in the selected scope.

NOTE:

The Query Options section of the Edit Scope dialog box is used to perform a query to filter the node names in the Nodes in SITE list. For detailed instructions on performing queries, refer to [Scope Queries](#).

2. Define the nodes to be included in the scope.

Use the push buttons in the center of the Edit Scope dialog box to define the scope members. To define scope membership, select a node name from either list and choose the appropriate button:

- * **Include >>** - choose this button to include the selected node in the scope. The node name moves from the left list to the right list.
- * **Include All** - choose this button to include all listed nodes in the scope. All node names move from the left list to the right list.
- * **<< Remove** - choose this button to remove the selected node from the scope. The node name moves from the right list to the left list.
- * **Remove All** - choose this button to remove all nodes from the scope. All node names move from the right list to the left list.

NOTE:

Because user names can be edited via the View Inventory Details dialog box, the node names in the

Nodes in SITE list do not necessarily correspond to NetWare user names. As a result, there may be duplicate names in this list. Viewing the inventory details of nodes with the same name enables you to differentiate between the nodes. Choose the View button or double click on a name in either list to invoke the View Inventory Details dialog box for the selected node.

3. When the scope members are defined, choose the Save button.

See Also:

[Creating Scopes](#)

[Renaming Scopes](#)

[Copying Scopes](#)

[Deleting Scopes](#)

[Scope Queries](#)

Renaming Scopes

Use the following procedure to rename a scope. The procedure assumes that you have already chosen the Scopes command from the Tools menu to display the Scopes dialog box.

NOTE:

A scope can be renamed even if it is part of an actively scheduled package.

1. **To rename a scope, select the scope from the list of Scopes, and choose the Rename button.**

The Rename Scope dialog box displays prompting you to enter a new scope name.

2. **Enter the new scope name, and choose the OK button.**

The new scope name displays in the Scopes dialog box, and the old name is removed. All attributes of the old scope are preserved in the renamed scope (i.e., the scope members do not change).

See Also:

[Creating Scopes](#)

[Editing Scopes](#)

[Copying Scopes](#)

[Deleting Scopes](#)

[Scope Queries](#)

Copying Scopes

Use the following procedure to copy a scope. The procedure assumes that you have already chosen the Scopes command from the Tools menu to display the Scopes dialog box.

NOTE:

A scope can be copied even if the original scope is part of an actively scheduled package.

1. **To copy a scope, select the scope from the list of Scopes, and choose the Copy button.**

The Copy Scope dialog box displays prompting you to enter a name for the new scope.

2. **Enter the new scope name, and choose the OK button.**

The new scope name is added to the Scopes dialog box. The new scope members are identical to the original scope members.

See Also:

[Creating Scopes](#)

[Editing Scopes](#)

[Renaming Scopes](#)

[Deleting Scopes](#)

[Scope Queries](#)

Deleting Scopes

Use the following procedure to delete a scope. The procedure assumes that you have already chosen the Scopes command from the Tools menu to display the Scopes dialog box.

NOTE:

A scope that is part of a scheduled package cannot be deleted.

1. **To delete a scope, select the scope from the list of Scopes, and choose the Delete button.**

A prompt displays asking you to confirm the deletion.

2. **Choose the Yes button to delete the scope.**

If deleted, the scope name is removed from the Scopes dialog box.

See Also:

[Creating Scopes](#)

[Editing Scopes](#)

[Renaming Scopes](#)

[Copying Scopes](#)

[Scope Queries](#)

Introduction to Packages

Creating and activating a package is the method by which software is distributed across your local area network. When a package is created, it is assigned a scope and a "Start Date." Upon reaching the start date and running the SDUPDATE.EXE program at a workstation in the scope, an active package automatically gets sent to the workstation.

A package also consists of a fileset and/or a script to be distributed to the group of remote workstations. For example, a package named UPGRADE_DOS might include a scope of users running DOS 3.3. The package might also include a script which installs a fileset consisting of the DOS 6.0 files.

The software distribution update program (SDUPDATE.EXE) is integrated with the packaging functionality. The update program is responsible for determining the conditions for accepting or rejecting a package. The program is also responsible for reporting on package status as input to the Packages window and the Software Distribution Log.

Access to Package Functions

Most package functions are accessed by either:

- * choosing the Packages command from the Tools menu, or
- * choosing the Distribute tool bar button

Both of the above actions cause the Packages window to display.

Package management is performed by either choosing the buttons in the Packages window or by choosing the corresponding commands from the File menu. For example, when the Packages window is active, a new package can be created either by choosing the New button in the Packages window or by choosing the New Package command from the File menu.

The information in the Packages window is updated according to an interval called the "package timer." Package timer functions are accessed by choosing the Distribution command from the Administration menu.

Package Features

In addition to the contents and scope, every package is assigned a schedule and a method for delivery. A package's schedule is the date on which the package is to be distributed. The method of delivery specifies instructions for the receiving workstation. There are several options available to the workstation user when a package is received. For example, package ABC might be configured to prompt the user five times to accept the package before proceeding with the installation of its fileset.

A user can create a new package, as well as edit, rename and delete a package. Refer to the following topics:

[Creating Packages](#)

[Editing Packages](#)

[Renaming Packages](#)

[Deleting Packages](#)

Prioritizing Packages

Activating and Deactivating Packages

The Package Timer

Creating Packages

Use the following procedure to create a new package.

1. **Choose the Packages command from the Tools menu, or choose the Distribute tool bar button.**

The Packages window displays.

This window lists the names of all defined packages. For each package, the following is indicated:

- * **Start Date** - the date the package will be distributed
- * **Status** - the package's status (Active or Inactive)
- * **Total** - the total number of workstation in the package's scope
- * **Done** - the number of successful updates so far
- * **Errors** - the total number of failed attempts at performing an update

NOTE:

Packages that have the same Start Date are distributed in the order in which they appear in the Packages window. To change a package's priority, refer to [Prioritizing Packages](#).

2. **Choose the New button in the Packages window.**

The New Package dialog box displays prompting you to enter a name for the new package.

3. **Enter the new package name, and choose the OK button.**

The package name can be up to 80 characters, and all typed characters are valid.

Upon choosing OK, a New Package dialog box displays. The name of the new package is indicated in the title bar of the dialog box.

4. **Select a fileset and/or a script to be included in the package.**

Choose the down arrow button to the right of the Update Fileset and/or Update Script fields to display the list of existing filesets and scripts. Select the desired items from the drop-down lists.

NOTE:

A package must include either one fileset or one script, or both.

5. **Select the scope to receive the package.**

Choose the down arrow button to the right of the Update Scope field to display the list of existing scopes. Select a scope from the drop-down list.

NOTE:

A scope must be assigned to the package.

6. **Assign the package's Start Date.**

Enter the date on which the package is to be distributed. The current date appears as a default in this field. Use the up/down arrow buttons to the right of the Start Date field to scroll to the desired date, or press the <F4> key to display a calendar from which a date can be selected.

7. **Assign the Active or Inactive status to the package.**

Check the "Update active when saved" option to automatically place the package in an active state upon saving the package. (An active package will get distributed automatically on its assigned start

date.)

If this field is not checked, an Inactive status will be assigned to the package. An inactive package will not get distributed automatically on its assigned start date.

8. Specify the package's update option.

The selected Update Option instructs the software distribution update program how it should interact with the receiving workstation user at login time.

The four Update Options are:

- * **Force upgrade at next login** - This option forces the package's receipt on the user at the next login. If an error occurs, the distribution is halted so the administrator can address the problem and reschedule the package.
- * **User optional until [DATE] and then [ABANDON, FORCE] update** - This option allows the user to refuse the package until the indicated DATE. After the DATE, the package is either abandoned or forced to be received by the user.
- * **User optional, prompt user [# TIMES] and then [ABANDON, FORCE] update** - This option allows the user to refuse the package a certain number of times (# TIMES). After the threshold is reached, the package is either abandoned or forced to be received by the user.
- * **Run this package always** - This option forces the package's receipt on the user at each and every login. This update option is most useful in situations where system variables are being modified directly by users.

9. Specify the target path in which the fileset should be decompressed.

A default path must be assigned to the package. The default path is the target path in which the distributed software (i.e., fileset) is to be installed or copied. The default path can be:

- * entered as a hard-coded drive mapping and directory combination (e.g., C:\BIN\DRIVERS).
- * entered as a hard-coded server, volume and directory combination (e.g., SERVER/VOLUME:\DIR or VOLUME:\DIR). If a server is specified, then the user receiving the package must be attached to the server.
- * reliant upon a system configuration found at the receiving workstation. Reliance is implemented by using one of the following target default path options available from the drop down list associated with this field:
 - **[BOOT_ROOT]** - the root of the receiving machine's boot disk
 - **[HDRIVE]** - the receiving machine's first hard drive
 - **[NDRIVE]** - the receiving machine's first network drive
 - **[NETCFG]** - path to the receiving machine's network configuration (which must be in the path)
 - **[WINDIR]** - the receiving machine's Windows directory (the directory in which the login module finds WIN.INI - this directory must be in the path)
 - **[WINSYSDIR]** - the receiving machine's Windows\System directory (directory in which the login module finds USER.EXE - this directory may be in the SYSTEM directory below WINDIR, or in the path)

These variables can be used in combination with a hard-coded value (e.g., [WINSYSDIR]\TEMP). In this case, the backslash character (\) is required and the variable name must be first. If a specified directory does not exist, it will be created.

If desired, check the "Allow user to override installation path" option to allow the user at the receiving workstation to override the selected path.

10. To define advanced package options, choose the Advanced button.

The Advanced Options dialog box displays.

The advanced package options consist of the following categories:

- * **Windows Error Options**- options associated with how the software distribution update program should react in the event that the Windows directory is not discovered at a receiving workstation.
- * **Fileset and Script Options** - options which define the execution precedence for the package's fileset and script (e.g., which one the software distribution update program should run first at the receiving workstation).
- * **Script Error Options** - options associated with how the software distribution update program should react in the event of script errors. (The script function error codes are detailed in [Introduction to the Distribution Script Language](#).)

NOTE:

Several script functions may return a non-zero value that is not considered to be an error. For example, the FINDFILE function returns a -1 if the file is not found; the STRCOMPARE function returns non-zero value if the two strings are not equal. You might want to disable the Script Error Options when using functions that return non-zero values even when successful.

11. When all package attributes are defined, choose the Save button.

See Also:

[Editing Packages](#)

[Renaming Packages](#)

[Deleting Packages](#)

[Prioritizing Packages](#)

[Activating and Deactivating Packages](#)

[The Package Timer](#)

Advanced Package Options

The Advanced Options dialog box displays by choosing the Advanced button in the Package window. The advanced package options consist of the following categories:

- * **Windows Error Options** - options associated with how the software distribution update program should react in the event that the Windows directory is not discovered at a receiving workstation.
- * **Fileset and Script Options** - options which define the execution precedence for the package's fileset and script (e.g., which one the software distribution update program should run first at the receiving workstation).
- * **Script Error Options** - options associated with how the software distribution update program should react in the event of script errors. (The script function error codes are detailed in [Introduction to the Distribution Script Language](#).)

NOTE:

Several script functions may return a non-zero value that is not considered to be an error. For example, the [FINDFILE](#) function returns a -1 if the file is not found; the [STRCOMPARE](#) function returns non-zero value if the two strings are not equal. You might want to disable the Script Error Options when using functions that return non-zero values even when successful.

See Also:

[Creating Packages](#)

Editing Packages

Editing a package may become necessary in order to modify package attributes.

Use the following procedure to edit the attributes of a package. The procedure assumes that you have already chosen the Packages command from the Tools menu to display the Packages window.

1. **To edit package attributes, select the package from the Packages window, and choose the Edit button.**

A package can also be selected for edit by double clicking on the package name in the Packages window. The Edit Package dialog box displays showing the configuration of the selected package. The fields and options in this dialog box are identical to the New Package dialog box.

The name of the package being edited displays in the title bar of the Edit Package dialog box.

2. **Modify the package attributes.**

Changes can be made to any field except the Update Scope, Update Fileset and Update Script fields.

NOTES:

a - Although the package's scope, fileset and script cannot be changed, their definition can be changed. For example, if Scope ABC is assigned to the package, you cannot assign Scope XYZ to the package but you can edit the members of scope ABC.

b - If a package fails, it can be re-distributed to a user by first removing the user from the scope and saving the edited scope, and then adding the user back into the scope and again saving the edited scope. Be careful when doing this because editing a scope changes all instances of the scope (i.e., even those included in packages).

If the distribution of a package has already begun, the changes made to the package take effect on the next node in the scope to receive the package.

3. **Choose the Save button.**

The Packages window re-displays.

NOTE:

To display the Software Distribution Log History details associated with a package, highlight the package in the Packages window and choose the Details button. The Log Details dialog box displays showing the details for the selected package. This dialog box is discussed in [Monitoring Software Distribution](#).

See Also:

[Creating Packages](#)

[Renaming Packages](#)

[Deleting Packages](#)

[Prioritizing Packages](#)

[Activating and Deactivating Packages](#)

[The Package Timer](#)

Prioritizing Packages

The priority with which each package is run is determined by its position in the Packages window. Packages that have the same Start Date are distributed in the order in which they are listed.

To modify a package's run time, highlight the package in the Packages window, and choose either the up or down arrow buttons. The highlighted package will be moved in the selected direction.

A package status having an "(A)" (e.g., Complete(A)) indicates that although the package has been distributed to all workstations in the scope, the package is still active. Therefore, if the package's scope is edited to include additional nodes, the package will automatically get distributed to those new nodes.

See Also:

[Creating Packages](#)

[Editing Packages](#)

[Renaming Packages](#)

[Deleting Packages](#)

[Activating and Deactivating Packages](#)

[The Package Timer](#)

Renaming Packages

Changing the name of an existing package renames the package in the Packages window.

Use the following procedure to rename a package. The procedure assumes that you have already chosen the Packages command from the Tools menu to display the Packages window.

NOTE:

Actively scheduled packages can be renamed.

1. **To rename a package, select the package from the Packages list, and choose the Rename button.**

The Rename Package dialog box displays prompting you to enter a new package name.

2. **Enter the new package name, and choose the OK button.**

The new package name displays in the Packages window, and the old name is removed. All attributes of the old package are preserved in the renamed package (i.e., the package's scope, script and/or filesset definitions do not change).

See Also:

[Creating Packages](#)

[Editing Packages](#)

[Deleting Packages](#)

[Prioritizing Packages](#)

[Activating and Deactivating Packages](#)

[The Package Timer](#)

Activating and Deactivating Packages

New packages are assigned the Active status if the Update Active when Saved option is selected. An active package automatically gets distributed upon reaching its assigned Start Date. An inactive package will not get distributed until it is re-activated.

The status indication of a selected package in the Packages window toggles between Active/Inactive by choosing the Activate/Deactivate toggle button in the Packages window.

The status of a completed package (i.e., a package that has been sent to all nodes in its scope) remains active. By highlighting a completed active package (e.g., "Complete (A)") and choosing the Deactivate toggle button, the status changes to "Complete (I)" which indicates an inactive status.

See Also:

[Creating Packages](#)

[Editing Packages](#)

[Renaming Packages](#)

[Deleting Packages](#)

[Prioritizing Packages](#)

[The Package Timer](#)

Deleting Packages

Use the following procedure to delete a package. The procedure assumes that you have already chosen the Packages command from the Tools menu to display the Packages window.

NOTE:

Actively scheduled packages cannot be deleted. To delete a package with an Active status, first make the package status Inactive by highlighting the package and choosing the Deactivate button. Then perform the Delete action.

1. **To delete a package, select the package from the Packages window, and choose the Delete button.**

A prompt displays asking you to confirm the deletion.

2. **Choose the Yes button to delete the package.**

If deleted, the package name is removed from the Packages window.

NOTE:

Deleting a package does not delete the associated log entry in the Software Distribution Log History dialog box.

See Also:

[Creating Packages](#)

[Editing Packages](#)

[Renaming Packages](#)

[Prioritizing Packages](#)

[Activating and Deactivating Packages](#)

[The Package Timer](#)

The Package Timer

The Packages window displays the status of each scheduled package. Status information includes the number of workstations in the package's scope (Total), the current number of successful updates (Done) and the total number of failed attempts at performing an update (Errors). The window contents are updated according to a defined package timer interval.

Setting the Package Timer Interval

Use the following procedure to set the package timer and define the interval at which the Packages window data is updated.

1. **Choose the Distribution command from the Administration menu. From the sub-menu which displays, choose the Set Package Timer command.**

The Set Package Timer dialog box displays.

2. **Enter the time interval at which the Packages window should be updated, and choose the OK button.**

Enter the time in seconds. (The default is 30 seconds.) The information in the Packages window will be updated at the defined interval.

NOTE:

The status of the Packages window can also be updated on demand by choosing the Distribution command from the Administration menu. From the sub-menu which displays, choose the Query Now command.

See Also:

[Creating Packages](#)

[Editing Packages](#)

[Renaming Packages](#)

[Deleting Packages](#)

[Prioritizing Packages](#)

[Activating and Deactivating Packages](#)

Monitoring Software Distribution

The chronological events associated with scheduled packages can be viewed from the Software Distribution Log History dialog box. The log history provides the information necessary for monitoring the success or failure of a distributed package. For example, if package WINUPDATE is distributed to a scope having three members, the installation of the software and update details for all three receiving workstations can be verified via the log history details.

Access to the Software Distribution Log

The Software Distribution Log History dialog box is displayed by choosing the Distribution command from the Administration menu. From the sub-menu which displays, choose the View Distribution Log command.

The Software Distribution Log History dialog box is also displayed by choosing the Details button in the Packages window.

Refer to the following topic:

[Viewing Log History Details](#)

Viewing Log History Details

Use the following procedure to monitor and maintain the log history of distributed packages.

1. **Choose the Distribution command from the Administration menu. From the sub-menu which displays, choose the View Distribution Log command.**

The Software Distribution Log History dialog box displays.

The history log provides a summary of all scheduled packages. The following information is provided for each package:

- * **Start Date** - the date the package will be distributed
- * **Total** - the total number of workstation in the package's scope
- * **Done** - the number of successful updates so far
- * **Errors** - the total number of failed attempts at performing an update

NOTE:

The completed updates (Done column) may be greater than the number of users scheduled to receive a package (Total column). This can occur as a result of rescheduling packages or when using the "Run this package always" update option during package scheduling.

2. **To view the individual events of a package, select the package from the Software Distribution Log History dialog box, and choose the Details button.**

A package can also be selected by double clicking on the package name in the Software Distribution Log History dialog box. The Log Details dialog box for the selected package displays.

In addition to the distribution Date/Time and target Site Name, the Details of the package's chronological events are shown in three lines:

- * Identification of the target workstation - node address, user name
- * Additional target workstation information - location, asset tag
- * Results - some possible results are:
 - Package installed successfully.
 - Error [#]: Script "[Script name]" has not been completed successfully.

NOTE:

Refer to DOS Error Codes for a list of error numbers related to unsuccessful script execution. Non-zero error numbers only display if the corresponding option was selected when the package was created. For more information on the available package options, refer to Advanced Package Options.

3. **To delete a log entry, select the package from the Software Distribution Log History dialog box, and choose the Delete button.**

The package is removed from the list of scheduled packages.

NOTE:

A log entry cannot be deleted if its associated package still exists.

4. **Choose the OK button to close the Software Distribution Log History dialog box.**

The Update Program

The update program SDUPDATE.EXE must be run from each workstation in order for it to receive the distributed packages it has been sent. Upon BrightWorks installation, the update program is copied into the BrightWorks program directory.

The SDUPDATE program's syntax is as follows:

```
SDUPDATE [drive:[\path]]
```

in which *drive* and *path* are optional parameters. The brackets are not typed.

Consider the following examples:

Example	Result
SDUPDATE	SDUPDATE will look in the current directory.
SDUPDATE F:	SDUPDATE will look in the current directory on drive F:.
SDUPDATE F:\path	SDUPDATE will look in the directory F:\path.

NOTES:

- a - The Btrieve Record Manager must be loaded before running SDUPDATE.EXE. Refer to Consider Improving BrightWorks Database Performance for a discussion on the Btrieve options.*
- b - When running the Brequestor, BSPXCOM and BROUTER must also be loaded on the file server. For details on loading these programs, refer to your Novell documentation.*
- c - When running SDUPDATE.EXE in a DOS box, you must load another session of BREQUEST by entering the following command: BREQUEST /D:17000 /L After running SDUPDATE, end the additional session by issuing the ENDBTRV command.*
- d - Refer to "Using Brequest" for general instructions for configuring the Btrieve NLM.*

See Also:

Running the Update Program

Automating the Update Program

Running the Update Program

Use the following procedure to manually run the update program at a workstation.

1. **At the workstation which is to receive the distributed package, load the Btrieve Record Manager.**

Either Btrieve or Brequest can be used. Refer to Consider Improving BrightWorks Database Performance for a discussion on the Btrieve options.

2. **Make the BrightWorks directory your current directory.**

Use the DOS CD command to change into the BrightWorks program directory, or map a physical drive to the Fusion directory.

3. **Execute the SDUPDATE.EXE program.**

For example, enter the following at the DOS command line:

```
SDUPDATE
```

Upon executing SDUPDATE, several messages will display at the workstation. If the user has not been given the option to refuse the update or change the installation path, then the update program will continue automatically (e.g., the package's script or fileset will be installed at the workstation).

- a. **If you are given the option of refusing the package**, then a prompt displays asking whether or not you want to accept the package. To install the package at this time, type <Y>. To install the package another time (e.g., the next time the update program is run), type <N>.

NOTE:

If the date or maximum number of times has expired and the package is configured to 'force upgrade,' then the package will be installed regardless of the user's response to this prompt.

- b. **If you are given the option of overriding the installation path**, then a prompt displays asking whether or not you want to override the default installation path. To override the default installation path, type <Y>. To accept the default installation path, type <N>.

If you type <Y> to override the default installation path, you are prompted to specify a new installation path. Type the new path and press the <ENTER> key. The update program continues, and the package is installed.

See Also:

Automating the Update Program

Automating the Update Program

To ensure that SDUPDATE is executed on a regular basis, the command can be placed in the system login script.

The following example illustrates SDUPDATE being executed from within the system login script. (Refer to [Using Brequest](#) instructions on configuring the Btrieve NLM or VAP.)

```
....  
MAP G:=SERVER/SYS:FUSION  
DRIVE G:  
#BREQUEST /D:17000  
#SDUPDATE  
#ENDBTRV  
....
```

where *G:=SERVER/SYS:FUSION* is the drive ID and complete path where the BrightWorks files and update program are stored.

See Also:

[Running the Update Program](#)

BrightWorks Error Messages

All error messages associated with BrightWorks are listed alphabetically in each of the following categories:

- * [Inventory Error Messages](#)
- * [Metering Error Messages](#)
- * [Script Editing Error Messages](#)
- * [Software Distribution Error Messages](#)
- * [Software Distribution Update Program Error Messages](#)

Inventory Error Messages

The following error messages may be displayed while using BrightWorks inventory features.

Add Inventory This Field Cannot be Left Empty

The Site field must contain information.

Audit Not Completed On Site

The audit was not successfully completed.

Btrieve Error

All Btrieve errors are reported by a number. Refer to [Btrieve Error Messages](#) to discover the reason for the error.

Cannot Copy File Source File

The source file names for saving the baseline may not be set properly; there may not be enough disk space; you may not have sufficient rights to the destination directory; or the files might be flagged as Read Only.

Cannot Create File Target File

The target file(s) for saving the baseline could not be created. Check rights and disk space/directory entries.

Cannot Delete a PreDefined Report

The reports that are provided with BrightWorks cannot be deleted.

Cannot Initialize Btrieve

Btrieve failed to initialize; check available memory.

Cannot Initialize Libraries

DATALIB and/or WNCFS DLL's are missing. Check available memory.

Cannot ReDefine this Report

The reports that are predefined and provided with BrightWorks cannot be redefined. Try saving the report under a different name.

Could Not Access Shell Network Information

Your network is not responding. You must reboot the machine and retry the operation..

Could Not Create ID File

The file that generates IDs for stand-alone equipment could not be created. Check that the diskette is not write protected.

Could Not Login to Server

The name/password you entered is invalid.

Delete Unidentified Software This Item is in the Transfer List Delete it First.

A software title that is included in the transfer list was selected to be deleted. It must first be removed from the transfer list.

Entry Is Reserved for Fusion Use, Cannot Edit Entry

Some Categories, Classes and Types are reserved for BrightWorks use and cannot be edited or deleted. The following fields cannot be left blank:

- * Category
- * Class
- * Manufacturer
- * Product Name

For PC and MAC Software Lists, the following fields cannot be left blank:

- * Product Name
- * File Name/File Creator

- * File Size
- * Version Number

Error # Deleting Report

The system encountered the specified error attempting to delete a report.

Error #: Server "server name"

A network error occurred on the specified server. Consult NetWare documentation for a description of the error number.

Error Allocating Global Memory

BrightWorks could not allocate extra memory to store system files.

Error Reading Audit Parameters

The file WAUDCFG.DAT, which contains the Audit parameters, could not be read.

Error Removing Alert Timer

The system was unable to remove the timer for alerting.

Error Removing Audit Timer

The system was unable to remove the timer for an automatic audit.

Field Types Do Not Match

The BrightWorks field format (character or numeric) and the field to be imported format differ.

File 'WEQTRNA.DXT' Could Not be Found

The path specified to load stand-alone equipment from does not contain this file, which is needed for the stand-alone hardware.

Invalid Hour Entry

Entry must be in the range 12 AM - 12 PM.

Invalid Minutes Entry

Entry must be in the range 0 - 59.

Local Site Already Has This Name

The name you entered is already assigned to the local site. Select a different Site name.

Local Site Cannot be Deleted

The Site name you wish to delete is the name of the local site and cannot be deleted.

Local Transaction Files Could Not be Found

The directory from which BrightWorks is running does not have the local transaction files to which stand-alone information should be loaded. Make sure you run EQUIP or do an audit before trying to load stand-alone data.

Login Name Cannot Be Left Empty

The Login Name field cannot be empty when specifying access to a server.

Macintosh Update Aborted

The Macintosh update was aborted at user's request.

Memory Allocation Error

The system was unable to allocate memory for the specified operation. Close one or more applications and try the operation again.

NetWare Error

Could not attach to server due to one of the following:

- * a hardware error
- * the 8 server slots are in use
- * the maximum number of users for the server has been reached

No Data For This Report

There was no data satisfying the filtering conditions set in the report.

No Fields Selected

At least one field must be selected to print for the Custom Report or the Detailed Equipment Report.

No Fileservers Specified

No servers were included in the Audit.

No Sites Specified

No sites were specified in the Scope of Audit.

Not Enough Disk Space Available

The collector diskette does not have the space needed for stand-alone files.

Option Available Only from Menu Bar (Delete Component)

When creating new equipment components, bringing up the list of components, the user cannot delete an existing component.

Path Name Cannot be Left Empty Edit Site

The Path Name must contain information when editing sites.

Path Name is Already in Use

Sites must be assigned unique path names.

Product has Not been Installed

BrightWorks needs to be properly installed following the instructions in Chapter 2 of the BrightWorks manual. If the program has been properly installed, run CIN.EXE to re-enable BrightWorks.

Required Fields Are Missing

When importing data into the BrightWorks inventory baseline, one or more of the fields that uniquely identifies a workstation are missing: site nodeid_1 nodeid_2 name.

Selected Fields Will Not Fit on Page

Using the current page set-up, the fields that have been selected for printing will not fit across the page. Either select a smaller font, set a smaller maximum field width, select the "One Field Per Line" option, or select fewer fields to print.

Server Update Aborted

Server update aborted at user's request.

Server Update Not Completed

File Server information was not updated in the transaction file due to one of the following reasons:

- * The file WSERVER.DAT (containing the list of servers) could not be read
- * No servers were included in the Scope of Audit
- * No unique ID could be determined for that server

Site Name Cannot be Left empty edit site

When editing a Site, the Site field must contain information.

Source File Not Specified

When backing up the baseline, the source baseline file must be specified.

Supervisor Rights Required

Supervisor rights or equivalent are needed to run an audit on a server.

System Files To Audit This is Not a Valid File Name

A valid file name is up to 14 characters long; the period is required.

The Transaction File Is Empty

The transaction file is empty due to one of the following reasons:

- * EQUIP has not been run
- * MAC files were not loaded into the transaction files
- * Stand-alone equipment has not been loaded
- * The file server information update failed

There is no Audit Selected

When running the Audit Results Report, a specific audit must be selected.

Transfer Unid Software as Group Base Name / Version No. Already In Use

The combination of base name / version no is in use by another software title.

Unable to Configure Printer

The system was not able to initialize the printer. Make sure that a printer is set up for Windows and the correct driver is present in the Windows system directory. If necessary, delete the file WLA1.INI from the Windows directory and try to print again.

Unable to Create Timer Window

The system was unable to create the timer window for an automatic audit or alerting. This is probably due to low memory or system resources. Close one or more applications and try the operation again.

Unable to Create Window

The system was unable to create a window. This is probably due to insufficient memory or system resources. Close one or more applications and try the operation again.

Unable To Determine Unique ID

The file that stores the unique file server ID, which is in the SYS:SYSTEM dir as a hidden file called LAIID.CFG, could not be created or opened. If the hidden file is zero bytes, delete it and try again.

This message might also appear if the number of files in use exceeds the maximum number specified in the CONFIG.SYS file (usually when the user is running other Windows or DOS applications).

Unable to Initialize NetWare Structures

Your network is not responding. You must reboot the machine and retry the operation.

Unable to Initialize Timer

The system was unable to initialize the internal timer used to track automatic audits and alerts. This is probably caused by other applications using all system timers. Close an application that uses a timer and try the operation again.

Unable to Open Export File

The user must have write/create rights, and there must be enough disk space to create the export file. If you are trying to export to a root directory, try exporting to a sub-directory instead.

Unable to Open File

The system was not able to open the specified file. Make sure that a valid directory was specified and that the user has sufficient rights to this directory.

Unable to Open Import File

The file selected to be imported could not be opened. The file could be corrupted.

Unable to Open Report Files

The system was not able to open the files which contain the data being reported.

Unable to Retrieve Report Information

The system was not able to open or read one or more of the files required to run the specified report. Before creating reports, you must run an audit and have data in your baseline.

Unable to Write to Temporary File

There is not enough disk space for the temporary file. Free up some space, and then run the report again.

Metering Error Messages

The following error messages may be displayed while using BrightWorks inventory features.

A print destination must be chosen.

Please select a printer in the printer setup dialog box to run a report.

A report type must be selected.

Please specify a report type in the reports dialog box to run a report.

A server must be selected.

Please specify a server in the reports dialog box to run a report.

A source file must be indicated.

Please specify an input source in the reports dialog box to run a report.

A valid date range must be provided.

Please specify a date range for filter criteria in the reports dialog box to run a report.

Are you sure you want to export the data?

Choose the Yes button to continue with the export of the data file.

At least one filter criteria choice must be given!

Please specify at least one of the filter criteria, either date, user, or application.

File size exceeds 65,535 lines. Unable to export.

The data file is too big to be exported any of the standard formats. Please try Btrieve as the export type.

Invalid user name supplied.

Please specify a valid user name in the attach dialog box.

No additional servers can be found.

BrightWorks metering capability could not detect additional servers.

No default printer! Select a printer from PRINT SETUP menu.

Please select a printer in the printer setup dialog box to run a report.

No input file specified. Cannot continue.

Please specify the name of the input file.

No output file specified. Cannot continue.

Please specify the name of the output file on the export dialog box for data output.

No output type specified. Cannot continue.

Please select one of the output formats specified in the drop down list.

No server specified. Cannot continue.

Please select a server to generate reports.

No source type specified. Cannot continue.

Please specify one of the four radio buttons on the export dialog box for input source.

Please select or enter a server name before pressing the OK pushbutton.

Please specify a server in the attach or detach dialog box.

Report setup choices incomplete.

All selections necessary to run reports have not been made in the reports dialog box.

The 'sitedata' file has been corrupted. Please contact McAfee Technical Support for assistance.

Cannot read the SITEDATA file. Please contact McAfee Technical Support for assistance at 908-530-9650.

Unable to establish connection with DataLIB DLL. Cannot continue.

Please check to see that the file DLWBC31.DLL is in the same directory as SREPORT.EXE.

Unable to initialize Btrieve DLL file.

Please check and make sure that the files WBTRCALL.DLL and WBHANDLE.DLL are in the same directory as SREPORT.EXE.

Unable to initialize DataLIB DLL; export functions will be unreliable.

Please check to see that the file DLWBC31.DLL is in same directory as SREPORT.EXE.

Unable to open input file. Cannot continue.

Please check to see that the input files SITEDATA or VIRUSDTA are in the directory you have selected.

Unable to open output file. Cannot continue.

Please check to see that the file DLWBC31.DLL is in the same directory as SREPORT.EXE.

When 'File' is the print destination, a file type must be selected.

Please specify an output file name to print a report to a file.

Would you like to export new baseline source files for your report?

Choose the Yes button to continue with the export of the data files. You may choose the No button if you have recently exported the data files. This will save some time in the report generation process.

Unable to create Btrieve output file. File may be in use.

The Btrieve data output file with extension .DAT may already be in use by another user. Ensure that no one else is using the file and try again. If the problem still persists, please use the Btrieve utility provided by NetWare. Type BUTIL -RESET filename.DAT.

Unable to create peaks.dat file.

The Btrieve data output file PEAKS.DAT may already be in use by another user. Ensure that no one else is using the file and try again. If the problem still persists, please use the Btrieve utility provided by NetWare. Type BUTIL -RESET PEAKS.DAT.

Unable to initialize Btrieve requester.

Btrieve requester Brequester may not be running. Please exit windows and run BREQUEST.EXE.

Unable to insert Btrieve record.

Btrieve was unable to insert one of the records in the export process. The original file may be corrupted, please call McAfee Technical Support.

Unable to open peaks.dat file.

The Btrieve data output file PEAKS.DAT may already be in use by another user. Ensure that no one else is using the file and try again. If the problem still persists, please use the Btrieve utility provided by NetWare. Type BUTIL -RESET PEAKS.DAT.

Unable to open sitedata.dat file.

The Btrieve data output file SITEDATA.DAT may already be in use by another user. Ensure that no one else is using the file and try again. If the problem still persists, please use the Btrieve utility provided by NetWare. Type BUTIL -RESET SITEDATA.DAT.

Script Editing Error Messages

The following error messages may be displayed while using BrightWorks script editing features.

Another script is already using this file.

You must enter a unique file name for each script.

Do you want to save the changes?

You have tried to close the script editor window without saving your changes. Choose the Yes button to save your editing changes and close the script editor, or choose the No button to cancel your changes.

Duplicate names are not allowed.

A script with this name already exists. Enter a unique name.

File does not exist.

The script source file cannot be found. Verify the drive and directory on which you are searching.

File exceeds capacity of this editor.

A script source file cannot exceed 64k in size.

Source and target file names are the same.

When copying or renaming scripts, you must enter a target name that is different than the script source file name.

Text Not Found.

The string being searched for via the Find String dialog box cannot be found.

There is no entry selected.

You must first select a script before choosing the delete or rename options.

This script cannot be deleted because it is a part of a scheduled package.

You cannot delete a script that is part of an actively scheduled package.

Unable to create new file.

The new script source file cannot be created. Verify that you have sufficient rights to create the new file.

You cannot edit more than eight documents at a time.

You can have a maximum of eight script editor windows open.

Software Distribution Error Messages

The following error messages may be displayed while using BrightWorks distribution features.

node(s) have been deleted from the Inventory since the last time this scope was edited. These nodes will be deleted from this scope and the corresponding packages upon saving.

Some nodes have been deleted from your baseline since the last time you modified this scope.

These nodes will be deleted from your scope and all packages this scope is attached to when you save the scope.

A site must be defined before this feature may be utilized.

You must define a Site before using any of the distribution features (Packages, Scopes, etc..).

An error occurred compressing fileset.

An internal compression error occurred. The file(s) in the fileset may be corrupt.

An error occurred copying a file.

An error occurred while copying a fileset. Check your rights and the available disk space.

An invalid path to filesets was detected. It has been reset to the current directory.

This is an informational message. It generally occurs if you change the drive mapping from when BrightWorks was originally installed. Use the Administration/Distribution/Preferences menu commands to set the fileset path to the desired location.

Could not create Fileset "<name>".

The fileset could not be created. Check your rights and available disk space.

Could not create the target file.

Unable to create the new fileset during a copy. Check your rights and the available disk space.

Could not locate the Fileset for this package.

Either the fileset has been deleted, or the path to the filesets has been changed. Verify the path to filesets from Administration/Distribution/Preferences menu commands.

Error allocating memory.

General error indicating that not enough memory and/or system resources exist to accomplish a particular task. Close one or more applications and retry the operation.

Error renaming Fileset.

The fileset could not be renamed. Check your rights.

Error updating: <name>.

Unable to write to the fileset. Check your rights.

Insufficient memory available to compress files. Close one or more applications and try again.

Close one or more applications to free system resources/memory.

Name cannot be left empty.

You must enter a valid name for a fileset, script, or scope.

The fileset path is invalid. Only a valid network drive may be chosen.

You cannot store filesets on your local hard drive because your users who run SDUPDATE will not be able to locate them. Store all filesets in a location from which all users have access.

There are no log entries to display.

No users have run SDUPDATE for this package yet.

This Fileset cannot be deleted because it is part of a scheduled package.

This Scope cannot be deleted because it is part of a scheduled package.

This Log entry cannot be deleted because it is part of a scheduled package.

This Script cannot be deleted because it is part of a scheduled package.

You must first delete any packages a scope, script fileset or log entry is part of before deleting that item.

This scope has been deleted.

This scope was deleted (probably by another user) before you attempted to edit it.

Unable to delete "<name>".

The fileset could not be deleted. Check your rights.

Unable to determine Fileset name. You may be running low on memory.

Close one or more applications to free up system resources/memory.

Unable to locate inventory equipment record. This node has probably been deleted.

Another user probably deleted the specified user from your baseline.

Unable to open source file.

Could not open the source fileset for copy. Check that it exists and that you have the appropriate rights.

Unable to read files in this Fileset. The Fileset is probably corrupted.

An unexpected 'End Of File' was encountered in the fileset. Delete the fileset and create it again. This fileset was corrupted.

Write error while compressing fileset. Check to be sure that you have sufficient rights and enough disk space.

The fileset could not be compressed. Check your rights and available disk space.

You cannot delete a package that is active.

Deactivate the package, and then delete it.

You must create a Scope before you can save a package.

Because a package must consist of one scope, at least one scope must be created before creating a package.

You must select a Fileset and/or a script for this package.

Select either a fileset and/or a script to run for this package.

You must select a Scope that has at least one node.

A scope of at least one node is required in order to save a package.

You must specify a default installation path.

You must specify a path in which to install the fileset and to which the [TARGET] system variable should default.

Software Distribution Update Program Error Messages

The following error messages may display while running BrightWorks distribution Update program (SDUPDATE.EXE). The messages are listed in error number order.

ERROR 0101: Error determining boot drive letter.

This is an internal DOS error. You may be using an old DOS version. DOS 3.x is required.

ERROR 0103: Error determining first available network drive.

Check the connection to the file server, and retry the operation.

ERROR 0104: Error determining first available hard drive.

This is an internal DOS error. You may be using an old DOS version. DOS 3.x is required.

ERROR 0105: Not enough memory to create system variable: <VariableName>.

Ran out of memory. Unload some TSRs and/or device drivers.

ERROR 0106: There is no Inventory Site in the specified drive.

Before a package can be installed, you must run EQUIP on the workstation which is to receive the distributed package. You must also run an audit to include that workstation in the baseline. Be sure that you are specifying the proper path to the BrightWorks inventory database file.

ERROR 0107: Floppy Disk Error: XX

Check that the disk is in the drive, is write enabled, and is formatted properly.

ERROR 0108: Drive 'X:' not ready or invalid drive.

Check that the disk is in the drive, is write enabled, and is formatted properly.

ERROR 0109: EQUIP needs to be run first, or your disk is write protected.

Before a package can be installed, you must run EQUIP on the workstation which is to receive the distributed package. You must also run an audit to include that workstation in the baseline. Be sure that you are specifying the proper path to the BrightWorks inventory database file.

ERROR 0110: This PC isn't in the Fusion Inventory database!. Be sure to run an audit prior to running SDUPDATE!"

Before a package can be installed, you must run EQUIP on the workstation which is to receive the distributed package. You must also run an audit to include that workstation in the baseline. Be sure that you are specifying the proper path to the BrightWorks inventory database file.

ERROR 0111: There is no drive letter specified in default path: <Path>

The default path that was specified for this package is invalid. It must be in the form of "d:\[path]", "SERVER/VOLUME:\[path]", "VOLUME:\[path]", or one of the pre-defined system variables present in the combo box. D:\ is the drive letter, and [path] is the optional path. The user running SDUPDATE must be attached to SERVER, VOLUME must be a valid volume on that server, and the user must have rights to that volume.

ERROR 0112: An invalid drive letter was specified in default path: <Path>

The default path that was specified for this package is invalid. It must be in the form of "d:\[path]", "SERVER/VOLUME:\[path]", "VOLUME:\[path]", or one of the pre-defined system variables present in the combo box. D:\ is the drive letter, and [path] is the optional path. The user running SDUPDATE must be attached to SERVER, VOLUME must be a valid volume on that server, and the user must have rights to that volume.

ERROR 0113: Determining drive mapping to SERVER/USER in default path: <Path>

The default path that was specified for this package is invalid. It must be in the form of "d:\[path]", "SERVER/VOLUME:\[path]", "VOLUME:\[path]", or one of the pre-defined system variables present in the combo box. D:\ is the drive letter, and [path] is the optional path. The user running SDUPDATE must be attached to SERVER, VOLUME must be a valid volume on that server, and

the user must have rights to that volume.

ERROR 0114: Error creating default path: <Path>

The default path that was specified for this package is invalid. It must be in the form of "d:\[path]", "SERVER/VOLUME:\[path]", "VOLUME:\[path]", or one of the pre-defined system variables present in the combo box. D:\ is the drive letter, and [path] is the optional path. The user running SDUPDATE must be attached to SERVER, VOLUME must be a valid volume on that server, and the user must have rights to that volume.

ERROR 0115: Not attached to file server: <ServerName>

The user running SDUPDATE must be attached to the server.

ERROR 0116: NetWare error: 0xXXXX

NetWare error. Check connection to server, and verify your rights.

ERROR 0117: Invalid SERVER/VOLUME format in default path: <pathname>

The default path that was specified for this package is invalid. It must be in the form of "d:\[path]", "SERVER/VOLUME:\[path]", "VOLUME:\[path]", or one of the pre-defined system variables present in the combo box. D:\ is the drive letter, and [path] is the optional path. The user running SDUPDATE must be attached to SERVER, VOLUME must be a valid volume on that server, and the user must have rights to that volume.

ERROR 0118: No such volume: <VolumeName>

The update program is unable to locate the specified volume. Be sure the user running SDUPDATE has rights to the volume and that the volume exists.

ERROR 0119: There are no available drive letters to map a drive to.

The user running SDUPDATE's drive map table is full. Delete one or more drive mappings for this user.

ERROR 0120: Windows must be in your path to install this package. Aborting package install.

This package requires that SDUPDATE can locate Windows in order to install. Add the Windows directory to your path.

ERROR 0121: Unable to open Fileset: <FilesetName>

Could not open the fileset. It was either deleted or the user has no rights to the fileset path.

ERROR 0122: Unable to allocate buffers for Fileset: <FilesetName>

Ran out of memory. Unload some TSRs and/or device drivers.

ERROR 0123: File <FilesetName> is not a valid fileset!

SDUPDATE found the file, but it is not a valid fileset. From the BrightWorks console, choose the Fileset command from the Tools menu to create and edit filesets.

ERROR 0124: Unable to create file: <FileName>

Could not create a file. Check your rights and the available disk space.

ERROR 0125: Error in Fileset: <FilesetName>

An unexpected End of File was encountered in the fileset. Delete the fileset and create it again. This fileset was corrupted.

ERROR 0126: Script "<ScriptName>" has not been compiled!

The script for this package was modified and attempted to have been compiled, but the compile failed.

ERROR 0127: File <filename> doesn't exist or isn't in path.

Could not locate the specified file.

ERROR 0128: Out of memory.

Ran out of memory. Unload some TSRs and/or device drivers.

ERROR 0129: Out of disk space decompressing: <FilesetName> to <DefaultPath>."

Out of space while decompressing the fileset. This test is done before any files are decompressed.

ERROR 0201: Unable to initialize Btrieve handler.

Ran out of memory. Unload some TSRs and/or device drivers.

ERROR 0204: Unable to locate script data file: <ScriptFile>!

The compiled script file could not be located. It was probably deleted. Re-compile the script.

ERROR 0206: The script failed on line XX.

This message displays any time the script fails while executing. The message references the line number on which the script failed.

ERROR 0211: <FunctionName> didn't have enough memory to create a variable.

Ran out of memory. Unload some TSRs and/or device drivers.

ERROR 0214: Maximum nest count reached processing function: IF.

IF functions can only be nested 50 levels deep.

ERROR 0216: Invalid drive letter specified in function: <FunctionName>

A valid path is: "d:\path" where *d* is a valid drive letter and *path* is a valid path.

ERROR 0217: Invalid path specified in function: <FunctionName>.

A valid path is: "d:\path" where *d* is a valid drive letter and *path* is a valid path.

ERROR 0218: Function UPGRADEOS requires DOS boot files on your boot disk!

The disk in your boot drive does not have any DOS system files. These files are necessary for the UPGRADEOS function to proceed.

ERROR 0219: Function UPGRADEOS needs the DOS files to perform the upgrade.

Be sure to run EQUIP on a DOS 5.0 workstation, a DOS 6.0 workstation, and a DOS 6.2 workstation prior to using the UPGRADEOS function for that DOS version. EQUIP picks up DOS system information and saves it in the path where the inventory databases are located, so SDUPDATE can find them and use them to upgrade. You should also be sure you have the appropriate license for the DOS version you are installing.

ERROR 0220: Out of memory in function: <FunctionName>

Ran out of memory. Unload some TSRs and/or device drivers.

ERROR 0221: UPGRADEOS was unable to delete system files from your boot disk.

The boot disk is probably write protected or missing from the drive, or there is a problem with the drive.

ERROR 0222: DOS version X.XX is already installed on your system!

The DOS version to be installed on your boot disk is already running on the user's machine.

ERROR 0223: UPGRADEOS error upgrading system files to boot disk.

ERROR 0224: UPGRADEOS: Unable to reset disk controller.

ERROR 0225: UPGRADEOS: Unable to read boot sector on boot disk.

ERROR 0226: UPGRADEOS: Unable to write boot sector to boot disk.

ERROR 0227: UPGRADEOS: Unable to read boot sector image file.

ERROR 0228: UPGRADEOS Error 'XX' opening: <FileName>

The boot disk is probably write protected or missing from the drive, or there is a problem with the drive.

ERROR 0229: <FunctionName> returned error code: XX.

A script function returned an error in [RETVAL], and the report non-zero return codes option was checked for this package.

Using Brequest

The following topics discuss the use and configuration of the server-based Btrieve record manager, BREQUEST.

Configuring the Btrieve NLM

Using Brequest in a Login Script

Using the Btrieve VAP

NOTES ON USING BREQUEST:

a - Loading SPX - All BrightWorks modules require SPX to be loaded at each workstation. Brequest communicates with BSPXCOM.NLM via SPX. BSPXCOM, in turn, passes all requests from Brequest to the BTRIEVE NLM. If SPX is not loaded at a workstation, that workstation has no communication whatsoever with the Btrieve record manager. If you have loaded Brequest and are getting "BTRIEVE Record Manager Not Loaded" messages, make sure that SPX is loaded. This normally only becomes an issue on workstations that load the ODI drivers because the ODI drivers can optionally load SPX.

b - Brequest /D switch - The /D: parameter specifies the size of the "data message buffer length." This buffer refers to the maximum record size that the NLM will transmit to the workstation. This switch should be set to 17000 for the purposes of BrightWorks.

c - Brequest /L switch - When running SDUPDATE.EXE in a DOS box, you must load another session of BREQUEST by entering the following command:

```
BREQUEST /D:17000 /L
```

After running the program, end the additional session by issuing the ENDBTRV command.

d - Brequest Error 87 - If you receive an "Error 87" from Btrieve while using Brequest, increase the 'Number Of Open Files' setting in BSETUP.

Configuring the Btrieve NLM

The Btrieve record manager must be loaded before running the BrightWorks EQUIP program. It is highly recommended that you use the server-based BREQUEST program while running EQUIP.

NOTE:

a - When using BREQUEST, version 6.10 or greater is required.

b - When using BREQUEST, BSPXCOM and BROUTER must also be loaded on the file server. For details on loading these programs, refer to your Novell documentation.

Using BREQUEST, as opposed to BTRIEVE, will improve the performance of data collection, auditing and reporting by at least 50% and as much as 500%. An additional advantage is the savings in local (client) memory - BTRIEVE.EXE can occupy from 50-85K of conventional memory, whereas BREQUEST.EXE can occupy approximately 31-45K. The exact amount of memory required for both programs depend on the specified command line parameters.

Use the following procedure to configure the Btrieve NLM.

1. To configure the NLM, run BSETUP.NLM.

At the file server console prompt, type:

```
LOAD BSETUP <ENTER>
```

2. Choose Set Configuration to verify that the following options are defined.

The values provided below are the minimum values required; your current values may be set higher.

- * Number of Open Files: = 22 (default = 20)
- * Number of Transactions: = 1 (default = 0)
- * Largest Record Size: = 17000 (default = 8192)
- * Largest Page Size: = 4096 (default = 4096)

NOTE:

All other BSETUP options can remain unchanged.

3. Save the configuration, and exit BSETUP.

BSETUP writes the configuration changes to the BSTART.NCF file. The changes do not take effect until the next time the Btrieve NLM is loaded.

4. To load the NLM, run BSTART.

At the file server console prompt, type:

```
BSTART <ENTER>
```

BSTART is an NCF file which loads both BTRIEVE.NLM and BSPXCOM.NLM.

5. To unload the Btrieve NLM, issue the BSTOP command.

If the Btrieve NLM was loaded when changes were made in BSETUP, you need to unload Btrieve and then reload the NLMs in order for the changes to take effect.

For example, at the console prompt, type:

```
BSTOP <ENTER>
```

```
BSTART <ENTER>
```

See Also:

[Using Brequest in a Login Script](#)

[Using the Btrieve VAP](#)

Using Brequest in a Login Script

To ensure that EQUIP is executed on a regular basis, the EQUIP command can be placed in your system login script. Because the Btrieve database must be loaded before running EQUIP, a Btrieve command must also be placed in the login script. EQUIP executes much faster when used in conjunction with Brequest and the BTRIEVE NLM; therefore, the login script should contain the BREQUEST.EXE command.

The following example illustrates how EQUIP can be run from the system login script, using BREQUEST. Comments are indicated in the text between the angle brackets.

```
MAP F:=FS/SYS:Fusion <maps drive to Fusion dir>
DRIVE F: <changes to the Fusion dir>
#BREQUEST /D:17000
#EQUIP <specify scan parameters>
#ENDBTRV
```

See Also:

[Configuring the Btrieve NLM](#)

[Using the Btrieve VAP](#)

Using the Btrieve VAP

All of the options and suggestions regarding the Btrieve NLM also apply to the Btrieve VAP. The BSETUP parameters, Brequest parameters, and Windows requester initialization settings are identical.

NOTE:

When using the Btrieve VAP, the file server needs to be rebooted in order for any changes in BSETUP to take effect.

See Also:

[Configuring the Btrieve NLM](#)

[Using Brequest in a Login Script](#)

Btrieve Error Messages

Refer to the following topics for a list of the return status codes for Novell's Btrieve Record Manager.

[Btrieve Status Codes](#)

[Client-Based Btrieve for OS/2 and Windows Status Codes](#)

[Btrieve Requester Status Codes](#)

Btrieve Status Codes

Btrieve returns a status code after each operation an application performs. If the operation was successful, Btrieve returns status code 0. If the operation was not successful, Btrieve will return one of the nonzero status codes listed below.

If Btrieve returns a code that is not contained in this section, refer to your Btrieve Installation and Operation Manual for error codes from the utilities.

01 INVALID OPERATION

The operation parameter specified in the call is invalid.

02 I/O ERROR

An error occurred during disk read/write. This status code indicates that the file has been damaged and must be recreated, or that the file specified on the open call was not created. This status code also occurs if the application passed an invalid position block.

03 FILE NOT OPEN

The operation cannot execute because the file is not open. A successful Open operation must be performed before Btrieve can process any other operations. This status code may also occur if the application passed an invalid position block for the file.

04 KEY VALUE NOT FOUND

The specified key value in the index path was not found.

05 DUPLICATE KEY VALUE

A record with a key field containing a duplicate key value cannot be added to an index that does not allow duplicate values.

06 INVALID KEY NUMBER

The value stored in the key number parameter was not valid for the file being accessed. The key number must correspond to one of the keys defined when the file was created or to a supplemental index.

07 DIFFERENT KEY NUMBER

The key number parameter changed before a Get Next, Get Next Extended, Get Previous, Get Previous Extended, Update, or Delete operation. The operation specified requires the same key number parameter as the previous operation because Btrieve uses positioning information relative to the previous key number.

If you need to change key numbers between consecutive Get Next, Get Next Extended, Get Previous, or Get Previous Extended operations, use a Get Position operation followed by a Get Direct operation to re-establish positioning for the new index path.

08 INVALID POSITIONING

The current position must be established to update or delete a record. Perform a Get or Step operation to establish the current position. This status code may also occur if the application passed an invalid position block for the file.

09 END-OF-FILE

The operation tried to read past the file boundaries (end-of-file or start-of-file). When reading a file in ascending order according to an index path, Btrieve returns the last record in that index path. When reading a file in descending order according to an index path, Btrieve returns the first record in the index path.

The Get Extended and Step Extended operations return this status code if the number of records satisfying the filtering condition is less than the number of specified records to be returned and the reject count has not been reached.

10 MODIFIABLE KEY VALUE ERROR

The operation tried to modify a key field which is defined as non-modifiable.

11 INVALID FILENAME

The filename specified does not conform to file naming conventions.

12 FILE NOT FOUND

The filename specified does not exist. Check the key buffer parameter to make sure the pathname is terminated with a blank or a binary zero.

13 EXTENDED FILE ERROR

Btrieve could not find the extension file for an extended file which the application tried to open. Extension files must be loaded on the logical disk drive specified when the extension was created. Both the primary file and its extension file must be on-line to access an extended file.

14 PRE-IMAGE OPEN ERROR

The pre-image file could not be created or opened. There are three possible causes for this error.

- * Btrieve could not create a new pre-image file because your disk directory is full. Btrieve must be able to create a pre-image file.
- * Btrieve could not open the pre-image file to restore file integrity. If the pre-image file was erased or damaged, Btrieve cannot restore the file's integrity. In this case, either use the RECOVER command in the BUTIL utility to retrieve the damaged file's data records in a sequential file, or replace the file with its most recent backup.
- * Btrieve could not assign a handle to the pre-image file because the Btrieve was not started by a user with access rights to the pre-image file.

15 PRE-IMAGE I/O ERROR

An I/O error occurred during the pre-imaging function. Either the disk is full or the pre-image file is damaged.

- * If the disk is full, erase any unnecessary files or extend the file to gain additional disk space.
- * If the pre-image file is damaged, the integrity of the Btrieve file cannot be ensured. Either use the RECOVER command in the BUTIL utility to retrieve the damaged file's data records in a sequential file, or replace the file with its most recent backup.

16 EXPANSION ERROR

An error occurred while writing the directory structure to disk prior to the creation of the expanded file partition. Either Btrieve could not close the file, or a new page was added to the file and Btrieve could not close and reopen the file to update the directory structure. Check for a disk hardware failure.

17 CLOSE ERROR

An error occurred while writing the directory structure to disk prior to closing the file. Either Btrieve could not close the file, or a new page was added to the file and Btrieve could not close and reopen the file to update the directory structure. Check for a disk hardware failure. This status code also occurs if the application passed an invalid position block for the file.

18 DISK FULL

The disk is full and the file could not be expanded to accommodate the insertion. Either erase any unnecessary files or extend the file to gain additional disk space.

19 UNRECOVERABLE ERROR

An unrecoverable error has occurred. File integrity cannot be ensured. Either use the RECOVER command in the BUTIL utility to retrieve the damaged file's data records in a sequential file, or replace the Btrieve with its most recent backup.

20 RECORD MANAGER INACTIVE

A request has been made before the Record Manager has been started. Restart the Record Manager.

In network environments, the operation was not processed because BREQUEST was not loaded. Reload BREQUEST.

21 KEY BUFFER TOO SHORT

The key buffer parameter was not long enough to accommodate the key field for the index path requested. Verify that the length of the key buffer equals the defined length of the key specified in the key number parameter. This status code can be returned only by certain interfaces.

22 DATA BUFFER LENGTH

The data buffer parameter was not long enough to accommodate the length of the data record defined when the file was created. Verify that the length of the data buffer is at least as long as the file's defined record length.

- * For *Get* or *Step* operations, if the data buffer is too short to contain the fixed length portion of the record, Btrieve does not return any data to the data buffer. If the record is a variable length record and the data buffer is too short to contain the entire variable length portion of the record, Btrieve returns as much data as it can and a status code 22, indicating that it could not return the entire record.
- * For the *Insert* operation, Btrieve does not insert the record if the data buffer is shorter than the fixed length portion of the record.
- * For the *Update* operation, if the data buffer is too short to contain the fixed length portion of any record, Btrieve does not update the record.
- * For the *Create*, *Stat*, and *Create Supplemental Index* operations, a status code 22 indicates that the data buffer is not long enough to contain all the file and key specifications, and the alternate collating sequence definition, if specified.

23 POSITION BLOCK LENGTH

The position block parameter was not 128 bytes long. This error can only be detected using certain language interfaces.

24 PAGE SIZE ERROR

The page size was invalid. The page size must be a multiple of 512 bytes but must be no larger than 4096 bytes.

25 CREATE I/O ERROR

The file specified could not be created. Possible causes are a full disk directory or a full disk. If you are creating a file over an existing file, Btrieve returns this status code if the existing file is open or the operating system does not allow the creation for some other reason (for example, a NetWare file is flagged transactional).

26 NUMBER OF KEYS

The number of keys specified for the page size was invalid. For standard Btrieve files with a page size of 512 bytes, the number of key segments must be between 1 and 8. For larger page sizes, the number of key segments must be between 1 and 24. You must define at least one key without the null attribute.

27 INVALID KEY POSITION

The key field position specified exceeded the defined record length for the file. Either the key position was greater than the record length or the key position plus the key length exceeded the record length. For key-only files, the key must begin in the first byte of the record (position 1).

28 INVALID RECORD LENGTH

The record length was invalid. The record length specified (plus overhead for duplicates) must be less than or equal to the page size minus 6 or greater than or equal to 4 bytes long.

29 INVALID KEY LENGTH

The key length was invalid. The key length specified must be greater than zero and cannot exceed 255. The length of a binary key must be even. Btrieve requires that each key page in the file is large enough to hold at least eight keys.

If the page size is too small to accommodate eight occurrences of the specified key length (plus overhead), either increase the file's page size, or decrease the key length.

30 NOT A BTRIEVE FILE

The filename specified is not a valid Btrieve file. Either the file was not created by Btrieve, or it was created by an earlier version of Btrieve.

Another possibility is that the first page of the file, which contains the File Control Record, is damaged.

31 FILE ALREADY EXTENDED

The file specified has already been extended. A file can be extended only once.

32 EXTEND I/O ERROR

The file could not be extended. Possible causes are that the directory is full, the disk is full, or the disk is write protected.

34 INVALID EXTENSION NAME

The filename specified for the extended partition was invalid.

35 DIRECTORY ERROR

An error occurred while changing to the directory that contains the Btrieve file. Either the drive specified in the Get Directory operation does not exist or the pathname specified in a Set Directory operation was invalid.

36 TRANSACTION ERROR

A Begin Transaction operation could not be performed because no transactions were specified when the Btrieve was initialized.

37 TRANSACTION IS ACTIVE

A Begin Transaction was issued while another transaction was active at that station. Transactions cannot be nested.

38 TRANSACTION CONTROL FILE I/O ERROR

An error occurred when Btrieve tried to write to the transaction control file. Possible causes were that the disk was full, the disk was write protected, or the transaction control file (which was created when the Btrieve was loaded) was deleted.

39 END/ABORT TRANSACTION ERROR

An End or Abort Transaction operation was issued without a corresponding Begin Transaction operation.

40 TRANSACTION MAX FILES

The application tried to update more than the maximum number of files allowed within a transaction. The maximum number of different files that can be updated during a logical transaction is set when Btrieve is configured. Refer to your *Btrieve Installation and Operation manual* for more information on configuration.

41 OPERATION NOT ALLOWED

The application tried to perform an operation that is not allowed at this time. Some operations are not allowed under certain operating conditions. For example, Btrieve returns this status code if you attempt to perform a Step, Update, or Delete operation on a key-only file or a Get operation on a data only file.

Also, certain operations are prohibited during transactions because they have too great an effect on the pre-image file or on Btrieve's performance. These operations include Close, Set or Clear Owner, Extend, Create Supplemental Index, and Drop Supplemental Index.

42 INCOMPLETE ACCELERATED ACCESS

The application tried to open a file that was previously accessed in accelerated mode and never successfully closed. The file's integrity cannot be ensured. Either use the RECOVER command in the BUTIL utility to build a new file or restore the file using the latest backup.

43 INVALID RECORD ADDRESS

The record address specified for a Get Direct operation was invalid. The address is outside of the file's boundaries; it is not on a record boundary within a data page, or on a data page. The 4-byte address you specify for a Get Direct operation should be one that was obtained by a Get Position operation.

44 NULL KEY PATH

The application tried to use the Get Direct operation to establish an index path for a key whose value is null in the corresponding record. Btrieve cannot establish positioning based on a null key value.

45 INCONSISTENT KEY FLAGS

The key flags specification on a Create operation was inconsistent. If a key has multiple segments, the duplicate, modifiable, and null attributes should be the same for each segment in the key.

46 ACCESS TO FILE DENIED

The application opened a file in read-only mode and tried to perform an Update, Delete, or Insert on that file. Another possible cause is that the owner name required for updates was not specified correctly when you opened the file.

47 MAXIMUM OPEN FILES

The number of files opened in accelerated mode exceeded the number of buffers available in Btrieve's cache. When a file is opened in accelerated mode, Btrieve reserves one of its cache buffers for the file. Btrieve always reserves five empty buffers for index manipulation. Reconfigure the Btrieve Record Manager with a smaller page size parameter to allocate more buffers.

48 INVALID ALTERNATE SEQUENCE DEFINITION

The first byte of an alternate collating sequence definition (the identification byte) did not contain the hexadecimal value AC.

49 KEY TYPE ERROR

The application tried to create a file or a supplemental index with an invalid extended key type, or tried to assign an alternate collating sequence to a binary key or key segment. You can only assign an alternate collating sequence to a string, lstring, or zstring key type.

This status code is also returned if you define a supplemental index requiring an alternate collating sequence, and no alternate collating sequence definition exists either in the file or in the key definition passed in the data buffer.

50 OWNER ALREADY SET

The application tried to perform a Set Owner operation on a file that already has an owner. Use the Clear Owner operation to remove the previous owner before specifying a new one.

51 INVALID OWNER

There are two possible causes for this status code:

- * If your application received this status code after a Set Owner operation, the owner names specified in the key buffer and data buffer did not match.
- * If your application received this status code after an Open operation, the file you tried to open has an owner name assigned to it. Your application must specify the correct owner name in the data buffer.

52 ERROR WRITING CACHE

While trying to make a cache buffer available, Btrieve tried to write data to a logical disk drive from a file that was previously opened in accelerated mode. An I/O error occurred during a write.

53 INVALID INTERFACE

An application tried to access a file containing variable length records with a language interface from Btrieve v3.15 or earlier. To access files with variable length records, you must use v4.xx or later interface.

54 VARIABLE PAGE ERROR

During a Step Direct operation, Btrieve could not read all or part of the variable length portion of a record. In this case, Btrieve returns as much data as possible to your application. This error usually indicates file damage to one or more pages in the file.

55 AUTOINCREMENT ERROR

The application tried to specify either the segmented or duplicate attribute for an autoincrement key type. An autoincrement key cannot be part of another key and cannot allow duplicates.

56 INCOMPLETE INDEX

A supplemental index was damaged. This can occur if a Create Supplemental Index operation or a Drop Supplemental Index operation is interrupted and does not run to completion. Perform a Drop Supplemental Index operation to completely remove the index from the file.

57 EXPANDED MEMORY ERROR

This error is applicable only in the client-based DOS environment. Btrieve returns this status if it receives an error from the Expanded Memory Manager. This error usually means that Btrieve was unable to save or restore the memory mapping register context, indicating an incompatibility with another application that uses expanded memory.

58 COMPRESSION BUFFER TOO SHORT

The application tried to read or write a record that is longer than the value specified for the size of the compression buffer. Reconfigure the Btrieve Record Manager, specifying a higher value for the "Maximum Compressed Record Size" option.

59 FILE ALREADY EXISTS

This status code is returned for the Create operation if you specified -1 in the key number parameter and the name of an existing file in the key buffer parameter.

60 REJECT COUNT REACHED

Btrieve rejected the number of records specified by the reject count before an Extended Get/Step operation found the requested number of records which satisfy the filtering condition. Check the first two bytes of the data buffer returned for the number of records that were retrieved.

61 WORK SPACE TOO SMALL

The Extended Get/Step operations use the pre-image buffer as work space. This error code indicates that the work space was not large enough to hold the filtering data buffer structure and the largest record to be received. The size of the work space is configurable (see the *Btrieve Installation and Operation manual* for more information). Check the first two bytes of the data buffer returned for the number of records that were retrieved.

62 INCORRECT DESCRIPTOR

The descriptor (data buffer structure), which is passed for an extended Get or Step operation, is incorrect.

63 INVALID EXTENDED INSERT BUFFER

Extended Insert provides an invalid buffer. Either the buffer length is less than five bytes, or the number of records specified is zero.

64 FILTER LIMIT REACHED

During an Extended Get Next/Previous operation, a rejected record was reached. Furthermore, this rejected record is such that no other record can satisfy the given filtering condition, going in the direction specified by the operation. This is applicable only if the key specified by the key number is also used as the filtering field.

65 INCORRECT FIELD OFFSET

The field offset in the extractor of an Extended Get/Step is invalid based on the length of the retrieved record.

74 AUTOMATIC TRANSACTION ABORT

This is an informative status code and is applicable only in the server-based environment. Btrieve replaced an End Transaction operation with an Abort Transaction because an error had been detected for a TTS file inside the transaction. In addition, Btrieve executed the Abort Transaction operation.

78 DEADLOCK DETECTED

Btrieve detected a deadlock condition. The application should clear all resources (such as aborting or ending the transaction or releasing all record locks) before proceeding. This allows the other applications to access the resources for which they are waiting.

80 CONFLICT

The Update or Delete operation could not be performed because the record was changed by another application since your application read the record. Reread the record prior to resending an Update or Delete operation.

81 LOCK ERROR

This error can result from one of two conditions:

- * The Btrieve lock table was full. Decrease the number of locks that your application uses or reconfigure the Btrieve Record Manager and specify a higher value for the "Maximum Number of Record Locks" option.
- * The application tried to unlock one record that was locked with a multiple record lock, and the record position stored in the data buffer did not correspond with any record that was locked in that file.

82 LOST POSITION

When performing a Get Next or Get Previous on a key with duplicates, the application tried to retrieve a record that was deleted or whose key value was modified by another application. Re-establish positioning using a Get Equal or a Get Direct operation.

83 READ OUTSIDE TRANSACTION

The application tried to delete or update a record within a transaction, but the record was not read within the transaction. If you are going to update or delete a record within a transaction, you must read the record within the transaction to ensure you have first obtained exclusive access to the data.

84 RECORD LOCKED

The application tried to apply a nowait lock on a record that was currently locked by another application, or to apply a nowait lock on a file while another application held active record lock(s) in that file.

If this status code is returned, your application can use either of the following two methods:

- * Retry the operation until it is successful. Under light to moderate network use, this may be the simplest and quickest solution.
- * Use the wait option (+100/+300) instead of the nowait option.

85 FILE LOCKED

The application tried to apply a nowait record or file lock while another application held the file locked. This status code is also returned when the application tries to open a file outside of a transaction and the file is locked by some other application.

If this status code is returned, your application can use either of the following two methods:

- * Retry the operation until it is successful. Under light to moderate network use, this may be the simplest and quickest solution.
- * Use the wait option (+100/+300) instead of the nowait option.

86 FILE TABLE FULL

Btrieve's file table was full. Reconfigure Btrieve and specify a higher value for the "Maximum Number of Open Files" option.

87 HANDLE TABLE FULL

This status code is applicable only in the server-based and Windows environments.

Btrieve's handle table was full. Reconfigure the Btrieve and specify a higher value for the "Maximum Number of File Handles" option.

88 INCOMPATIBLE MODE ERROR

The application tried to open a file in an incompatible mode. If the first application to access a file opens it in accelerated mode, all other applications must open it in accelerated mode. If the first application to access a file opens it in non-accelerated mode, other applications cannot open the file in accelerated mode.

90 REDIRECTED DEVICE TABLE FULL

This status code is applicable only in the server-based environment.

BREQUEST's redirection table or server routing table was full. This occurs if you attach to

additional servers or map to additional drives after you loaded BREQUEST. Reload BREQUEST, specifying a larger number for the "Number of File Servers" or "Number of Mapped Drives" options (/S and /R respectively).

This error also occurs if you detach a particular server and attach to a different server. Once a workstation has attached to a server, BREQUEST will not remove its name from the server routing table.

91 SERVER ERROR

This status code is applicable only in the server-based environment.

BREQUEST could not establish a session with the server. In NetWare environments, either the NetWare Btrieve Record Manager has not been started or the server was not active. Verify that the NetWare Btrieve Record Manager is active on the server in question.

92 TRANSACTION TABLE FULL

This status code is applicable only in the server-based environment.

The maximum number of active transactions was exceeded. Reconfigure Btrieve and specify a higher value for the "Number of Concurrent Transactions" option.

93 INCOMPATIBLE LOCK TYPE

Your application tried to mix single record locks (+100/+200) and multiple record locks (+300/+400) in the same file at the same time. All locks of one type must be released before a lock of the other type can be executed.

94 PERMISSION ERROR

Your application tried to open or create a file in a directory without the proper privileges. Btrieve does not override the network privileges assigned to users.

95 SESSION NO LONGER VALID

This status code is applicable only in the server-based environment.

The previously established session was no longer active due to an error at the workstation, the file server, or on the network. Verify that your workstation is still attached to the file server and then reload BREQUEST.

96 COMMUNICATIONS ENVIRONMENT ERROR

This status code is applicable only in the server-based environment.

This code occurs when loading Btrieve on a NetWare server. The SPX connection table is full. Reload SPX, specifying a higher value for the connection table. Refer to the NetWare system documentation for more information.

97 DATA MESSAGE TOO SMALL

This status code is applicable only in the server-based environment.

Your application tried to read or write a record which was longer than the Btrieve Record Manager or BREQUEST could handle. Reconfigure the Btrieve Record Manager and specify a higher value for the "Maximum Record Length" option. Reload BREQUEST and specify a higher value for the /D option.

- * For an Update, Insert, or Create operation, the application receives this error if the data buffer length it specifies for the record exceeds the length specified for the Btrieve Record Manager or BSERVER.
- * For a Get, Step, or Stat operation, the application receives this error if the value specified for the data buffer length is shorter than the length of the data Btrieve would return, regardless of the data buffer length specified in the program.

98 INTERNAL TRANSACTION ERROR

This status code is applicable only in the server-based environment.

An error has been detected while executing a previous operation on a NetWare TTS file. Therefore, no operation other than Abort Transaction (21) is allowed at this point.

99 THE REQUESTER CANNOT ACCESS THE NETWARE RUNTIME SERVER

The DOS Requester returns this status code when NetWare Runtime server support is enabled

(/C:1) and the Requester either detects no existing connection or cannot find a valid login username. If the Requester cannot find a login username other than SUPERVISOR, there is no valid name to pass.

100 NO CACHE BUFFERS ARE AVAILABLE

Btrieve has used all the cache buffers it allocated at load time. Using the Setup utility, you can increase the value for the Cache Allocation configuration option. Alternatively, you can change the Number of Remote Sessions configuration option to decrease the number of concurrent Btrieve users. For more information, refer to Chapter 3, "Installing and Configuring Btrieve," in the *Btrieve Installation and Operation* manual.

101 INSUFFICIENT OPERATING SYSTEM MEMORY IS AVAILABLE

There is not enough operating system memory available to perform the requested operation. Decrease the value for the Cache Allocation configuration option (using the Setup utility), decrease the number of concurrent Btrieve users (using the Number of Remote Sessions configuration option in the Setup utility), or add memory to the server. For more information on the configuration options, refer to Chapter 3, "Installing and Configuring Btrieve," in the *Btrieve Installation and Operation* manual.

102 INSUFFICIENT STACK SPACE IS AVAILABLE

Btrieve has run out of stack space. To increase the amount of stack space available to your application, relink the application, setting the stack size to a higher value. Only the NLM applications calling Btrieve on the local server get this message.

103 THE CHUNK OFFSET IS TOO BIG

A Get Direct/Chunk operation has specified an offset beyond the end of the record, either explicitly or through the use of the next-in-record bias to the subfunction value. Unless Btrieve returns this status while processing the first chunk, the operation was partially successful. Check the data buffer length parameter immediately after the call to see how much data (and therefore how many chunks) Btrieve retrieved.

This code can also be returned by the Update Chunk operation when the specified offset is more than one byte beyond the end of the record. However, in this situation, Status Code 103 indicates that Btrieve made no changes to the record.

104 THE LOCALE INFORMATION COULD NOT BE FOUND

The Create or Create Index function returns this status code to indicate that the operating system was not able to return a collation table for the country ID and code page specified. Check that the application specified the locale's country ID and code page correctly and that the operating system is configured to support the country ID and code page.

105 THE FILE CANNOT BE CREATED WITH VARIABLE-TAIL ALLOCATION TABLES (VATS)

The application specified that a Btrieve file should be created with Variable-tail Allocation Tables (VATs); however, the application failed to specify that the file was to use variable-length records (a precondition for files to use VATs). This status applies to key-only files as well as regular data files.

106 THE OPERATION CANNOT GET THE NEXT CHUNK

The application called the Get Direct/Chunk operation to retrieve a chunk from a record and used the next-in-record bias on the descriptor subfunction. However, after the application established its positioning in the record (but prior to this call), the target record was deleted.

107 CHUNK UPDATES/RETRIEVALS CANNOT BE PERFORMED ON THE FILE

The application tried to use either a Get Direct/Chunk operation or an Update Chunk operation on a pre-v6.0 formatted file.

Client-Based Btrieve for OS/2 and Windows Status Codes

Client-based Btrieve may return the following status codes in an OS/2 or Windows environment.

1001 THE MULTIPLE LOCKS OPTION IS OUT OF RANGE

The number specified for the Multiple Locks configuration option must be between 1 and 255, inclusive.

1002 BTRIEVE CANNOT ALLOCATE THE MEMORY NEEDED

Make sure that the workstation has enough memory to load all the programs it requires.

1003 THE MEMORY SIZE IS TOO SMALL

Make sure the value for the Memory Size configuration option is large enough to accommodate the required cache size.

1004 THE PAGE SIZE OPTION IS OUT OF RANGE

The value of the Page Size configuration option must be an even multiple of 512, and it must be between 512 and 4,096, inclusive.

1005 THE PRE-IMAGE FILE DRIVE OPTION IS INVALID

You must specify a valid drive letter for the Pre-Image File Drive configuration option.

Note:

Pre-image files are used only for files created by Btrieve versions earlier than v6.x, or by v6.x if it was loaded with the Create Btrieve Files in Pre v6.x Format configuration option set to Yes.

1006 THE PRE-IMAGE BUFFER SIZE OPTION IS OUT OF RANGE

The Pre-Image Buffer Size configuration option must be between 1 and 64, inclusive.

Note:

Pre-image files are used only for files created by Btrieve versions earlier than v6.x, or by v6.x if it was loaded with the Create Btrieve Files in Pre v6.x Format configuration option set to Yes.

1007 THE OPEN FILES OPTION IS OUT OF RANGE

The Open Files configuration option must be between 1 and 255, inclusive.

1008 THE CONFIGURATION OPTIONS ARE INVALID

The configuration options specified contain invalid or unidentifiable values. For more information on configuration options, refer to the installation and operation manual for your operating environment.

1009 THE TRANSACTION FILENAME OPTION IS INVALID

The filename specified for the Transaction Filename configuration option is not valid. Check to make sure that the transaction filename is correct.

1011 THE COMPRESSION BUFFER SIZE SPECIFIED IS OUT OF RANGE

The Compression Buffer Size configuration option must be between 1 and 64, inclusive.

1013 THE TASK TABLE IS FULL (WINDOWS ONLY)

The Btrieve DLL may return this status code if the task entry table is full. You can remedy this situation by increasing the number of available task entries; use the tasks initialization option (tasks=xxx) under the [BTRIEVE] or [BREQUESTDPMI] headings in NOVDB.INI. The minimum value for this option is 1; the maximum value is 255.

1014 THE APPLICATION ENCOUNTERED A STOP WARNING

WBTRVSTOP () returns this status code if the application still has open files or an active transaction. The application must close all files and end all transactions before calling WBTRVSTOP ().

1015 A POINTER PARAMETER IS INVALID

One of the pointer parameters passed into Btrieve is invalid.

1016 BTRIEVE IS ALREADY INITIALIZED

The Btrieve DLL may return this status code if an attempt is made to initialize Btrieve when it is already initialized. To reinitialize Btrieve, close all files, end/abort all transactions, and call WBTRVSTOP () before calling the initialization function.

1017 THE BTRIEVE REQUESTER FOR WINDOWS CANNOT FIND WBTRVRES.DLL

WBTRCALL.DLL returns this status code when it cannot find the resource file WBTRVRES.DLL. You can remedy this situation by placing a copy of the WBTRVRES.DLL file in the same directory as the WBTRCALL.DLL file.

Btrieve Requester Status Codes

This section lists the status codes that the Btrieve Requesters may generate.

2001 THE MEMORY ALLOCATION IS INSUFFICIENT

In an OS/2 environment, the Requester cannot allocate enough memory for the parameters specified with the BRQPARMS environment variable. In a DOS environment, reduce the value specified for the /D configuration option.

2002 THE OPTION IS INVALID OR OUT OF RANGE

In an OS/2 environment, either one of the options specified with the BRQPARMS environment variable is invalid (such as /P instead of /D) or the value specified for a parameter is out of range. Check the SET BRQPARMS statements to make sure it is correct.

2003 THE REQUESTER DOES NOT ALLOW LOCAL ACCESS TO THE SPECIFIED FILE

The application attempted to access a file stored on a local drive. The version of WBTRCALL.DLL installed at the workstation does not allow access to local files.

2004 SPX IS NOT INSTALLED

Install the NetWare SPX v1.3 or later communications software for OS/2.

2005 AN INCORRECT VERSION OF SPX IS INSTALLED

Install the NetWare SPX v1.3 or later communications software for OS/2.

2006 THERE IS NO AVAILABLE SPX CONNECTION

SPX has already established the maximum number of sessions it can handle. To increase the maximum, edit the NET.CFG file. Refer to your NetWare documentation for more information on NET.CFG.

2007 A POINTER PARAMETER IS INVALID

One of the pointer parameters passed to Btrieve is invalid. Check the program to ensure that the pointer parameters are correct.

NMS Smart-Launch Support

Many McAfee customers have expressed an interest in utilizing the McAfee network management applications as snap-in applications within the Novell NetWare Management System framework. In response to this request, McAfee has implemented support for NMS smart-launch with four of its currently shipping applications: BrightWorks, LAN Automatic Inventory, SiteMeter and NETremote+.

After following the instructions below, the user will be able to point at an object on an NMS map, select a McAfee command from the NMS Tools menu, and launch the selected McAfee application in a context-sensitive fashion.

Activating the Smart-Launch Feature

Use the following procedure to activate the smart-launch feature.

1. Obtain the McAfee NMS Explorer disk.

Contact your McAfee representative to obtain a copy of the "McAfee NMS Explorer" Disk. It is available free of charge.

2. Copy all files from the NMS Explorer disk.

Two sets of files must be copied from this disk for later use, plus one other file must be created.

- a. Copy all of the program files (EXE, DLL) to a new directory (e.g., NMS\BDI2NMS) on your NMS console (not into the NMS directories!).

NOTE:

It is important to NOT copy these files into the NMS\BIN directory as some of the Novell DLLs that are shipped on this disk may not be the latest DLLs available from Novell for NMS.

- b. Copy the FUSION.OLF file from this disk into the NMS\OLF directory. The McAfee disk includes all .OLF files that are necessary for the smart-launch ready McAfee applications. The .OLF file prefixes match the McAfee product EXE names (e.g., for SiteMeter the relevant files are SITEMETR.EXE and, therefore, SITEMETR.OLF).
- c. Copy any icon in the NMS\ICONS directory to NULL.ICO. When the Novell OLF Introducer (see below) is run it will require a named icon (.ICO file) for each class introduction in case the icon were to show up on an NMS map. Since McAfee does not force one of its icons onto the map, no icons are shipped on this floppy. Any of the icons in that directory may be used as the source icon.

3. Introduce NMS to McAfee.

Note that the Novell N-OLFI.EXE file must be in the NMS\BIN directory.

- a. Use Novell's OLF Introducer to add the McAfee classes to the NMS database. The Introducer takes .OLF files as input. As mentioned above, .OLF files for all smart-launch ready McAfee applications can be found on this disk. It is necessary only to introduce those applications that are installed to NMS. Choose the classes that you need for the McAfee applications that are desired.
- b. Run the OLF Introducer (N-OLFI.EXE). This latter program has no menu; by default it displays a standard Windows common dialog for opening a file. Browse to find the target McAfee .OLF file (e.g., FUSION.OLF, WLAI.OLF, SITEMETR.OLF, WNR.OLF), as appropriate.
- c. Run the Introducer once for each .OLF file, specifically for FUSION.OLF.

HINT:

By default the Introducer has no main window; therefore, any success/failure indications cannot be seen. Invoke N-OLFI.EXE with a -s switch (for "show") to see status messages regarding class introductions. The syntax is as follows:

syntax: N-OLFI.EXE -s

4. Explore the NMS databases for nodes and servers.

Run the McAfee BDI2NMS.EXE program included on this disk. This program requires no "setup," but it should be copied (as mentioned above) to its own directory. It requires several DLLs that are included on this diskette. These include the NWNETAPI.DLL and several of the N-*.DLLs from the NMS BIN directory.

When you run BDI2NMS.EXE, you may either run the "Enable All" command or select a specific McAfee product, such as BrightWorks. BDI2NMS.EXE uses the NMS database APIs to search the NMS data for workstations and servers. Smart-launch is enabled for each McAfee product in the following manner:

- * **SiteMeter** - smart-launch is enabled for servers only
- * **BrightWorks and LAI** - smart-launch is enabled for all workstations and servers
- * **NETremote+** - smart-launch is enabled for workstations only

NOTE:

McAfee COULD have implemented BDI2NMS.EXE so that it would cross-check the existence of inventory data or an installed SiteMeter NLM set before enabling the smart-launch on an individual basis. However, doing so would have made the software slower and would have also required the user to perform the NMS database query far more often than is necessary for normal use.

To remove these settings within the NMS database, use the Disable All command to erase them.

This software has been tested against NMS 1.15 and 2.0. If the NMS database is large, the enabling exploration may take a while - this is due strictly to the speed of the NMS database callback API.

5. Smart-launch BrightWorks or the BrightWorks module.

- * **BrightWorks** - Select the target PC or server where either EQUIP or an audit have been run, and then select the Fusion command in the NMS Tools menu. BrightWorks will skip a few steps and invoke the inventory details dialog box for the target node. The network number and NIC address are used as the synchronization key between BrightWorks and NMS.
- * **LAI** - same as BrightWorks.
- * **SiteMeter** - Select the server within an NMS map where SiteMeter is installed, and then select the SiteMeter command in the NMS Tools menu. The SiteMeter administration program is launched with an Application Usage Monitor showing information on application usage on that server.
- * **NETremote+** - Select the target workstation, and then choose the NETremote command in the NMS Tools menu. NETremote+ will be invoked with the NETremote options that are configured for the target PC. If the target PC is configured for remote control and has "listen" enabled, then NETremote+ will automatically take control of the target PC.

Should you have any questions regarding any of these steps or regarding the smart-launch of McAfee applications within NMS, please do not hesitate to contact McAfee Technical Support.

What is the Metering Reporting Module?

BrightWorks reporting module is a flexible tool designed to help you manage your network more effectively. It puts important information about application usage and software security at your fingertips.

Using its simple Windows interface, you can generate the report you need quickly and easily. And with a variety of report types and formats, you can choose the one best-suited to your business needs. For example, run a report on a single user to see what applications he or she uses. Or run a report on all network applications that you meter to see which applications are being used the most to determine additional software needs. All of this flexibility is at your fingertips with metering.

BrightWorks also lets you export coded data into standard database formats. You can choose the database format best-suited to your needs.

See Also:

[How Does It Work?](#)

[Report Types](#)

How Does the Metering Reporting Module Work?

The reporting module has two components--exporting files and generating reports. Once you access the reporting console, you can choose either of these two features.

If you choose to export files, the Reporting Module prompts you for the source, format and destination of the desired files. Once you specify all of this information, it exports the file(s) to the desired location.

If you choose to generate a report, the module requests information about the source files, report type, report filters and report destination. Once you specify all of this information, BrightWorks generates the report and displays it on the screen.

See Also:

[Accessing the Reporting Module](#)

Accessing the Metering Reporting Module

The Report function is accessed by choosing the Metering and Security command from the Reports menu.

See Also:

[Exporting Files](#)

[Generating Reports](#)

Exporting Files

The reporting module allows you to export coded data to a standard database format.

Use the following procedure to export coded data:

1. From the Reports main menu, choose File.

2. Choose the Export command.

The Export dialog box appears.

3. From the Current Server list box, specify the server from which you wish to export the coded file.

The Current Server list box automatically displays your current server. You can select another server by choosing one from the drop-down list box.

You can also use the Attach and Detach buttons to attach to/detach from other file servers.

4. Select one of the options to specify the source of the data you are exporting.

The four options are

- * Sitedata
- * Virusdta
- * Metering Definitions
- * Security Definitions

Sitedata and Virusdta are the files where metering stores its information. The Metering and Security Definitions options retrieve the information directly from the bindery. To select an option, simply click in the appropriate radio button.

NOTES:

a - If you choose Sitedata or Virusdta, the Default and Browse buttons are enabled. The filename text box automatically displays the default file name. For Sitedata the default is SYS:

\SYSTEM\ITEMETR\SITEDATA; and for Virusdta, the default is SYS:
\SYSTEM\ITEMETR\VIRUSDTA.

b - You can also specify a different file name (if Sitedata or Virusdta was stored in a different place) by entering the name in the text box or by choosing the Browse button. Choosing the Browse button produces the standard Windows Browse dialog box. Once you select the desired file and choose the OK button, the file name you selected displays in the Filename text box.

c - Because the Metering and Security Definitions options communicate directly with the bindery to access the information, these options are not needed and therefore are not enabled.

5. Choose the format for the exported data from the Format list box.

The available formats include:

- * Lotus 123 2.1
- * Lotus 123 3.x
- * Quattro
- * Lotus 123 1.x
- * Symphony 1.0
- * Symphony 1.1-2.2

- * Excel 2.0
- * Excel 3.0
- * Excel 4.0
- * dBase II
- * dBase III
- * dBase III Plus
- * dBase IV
- * Data Interchange Format
- * ASCII (Tab delimiter)
- * ASCII (Tab delimiter-guess numeric values)
- * ASCII (comma delimiter-min quoted)
- * ASCII (comma delimiter- strings quoted)
- * ASCII (comma delimiter-guess numeric values)
- * Btrieve

6. Specify the file name for the new format.

Enter the desired file name in the Filename text box or choose the Save As button. Choosing the Save As button produces the standard Windows Save As dialog box. Once you specify the path and file name and choose the OK button, you are returned to the Export dialog box. The name you specified displays in the Filename text box.

7. Once you have specified all the above information, choose the OK button.

8. At the prompt that displays, choose the Yes button to export the data.

You can also choose the No button to discontinue the export process.

If you choose the Yes button, the Export Status window displays.

This dialog box contains the following information:

- * Server
- * Input Source
- * Input File
- * Output File
- * Output Format

The percentage complete bar tracks the progress of the exporting.

During the export, you can choose the Abort button to halt the exporting. This button changes to Close once the export is complete. By choosing Close, the output file is put into the current directory.

Generating Metering Reports

Use the following procedure to generate metering reports:

1. Choose Reports from the main menu.

2. Choose the Choose Reports command.

The SiteMeter Reports dialog box displays.

This dialog box contains the following information:

- * Current Server
- * Report Type
- * Source
- * Print Destination
- * Activity Date Range
- * Filter Criteria

The dialog box also has Attach, Detach, OK, and Cancel buttons.

The functionality of all these options is explained in the following steps.

3. From the Current Server list box, select the desired server.

The Current Server list box automatically displays your current server. You can select another server by choosing one from the drop down list box.

You can also use the Attach and Detach buttons to attach to/detach from other file servers.

To attach to another file server, choose the Attach button. The Attach to Server dialog box appears. Choose the desired server from the drop down list box, enter your username and enter your password (if applicable). When finished, choose the OK button.

4. From the Report Types list box, select a report type.

The list box provides these options:

- * Metering Definitions (based on application usage)
- * Application Summary (based on application usage)
- * Application with User Detail (based on application usage)
- * Application with User Summary (based on application usage)
- * User Summary (based on user information)
- * User with Application Detail (based on user information)
- * User with Application Summary (based on user information)
- * File Integrity Activity (based on protected files information)
- * SPA Compliance (based on application usage)
- * Software Purchase Forecast (based on application usage)
- * Upgrade Purchase Forecast (based on application usage)

Your report selection drives the source file name, the activity date range and the filter criteria. Based on what you choose here, defaults will appear in these three sections of the dialog box.

5. Specify the source of the report in the Filename text box.

If the report type you selected in Step 4 uses data from a file, the file automatically displays in this list

box. For example, if the report type uses the Sitedata file, the text box will read:
SYS\SYSTEM\SITEMETR\SITEDATA.

You can choose another file either by entering one directly in the text box or by using the Browse button. The Browse button produces the standard Windows Browse dialog box. Once you select the path and file name and choose the OK button, you are returned to the Reports dialog box. The selected file name displays in the Filename text box.

Choose the Default button to define the default file as the report source.

6. In the Print Destination section of the dialog box, specify where you want to send the report.

The three options are Print to Window, Print to Printer, and Print to File.

If you want to send the report to a file, select the appropriate file from the File list box.

The options are:

- * Space Separated
- * Tab Separated

7. Set the Activity Date Range.

The From: and To: fields list the default data range (which is the entire date range of the file that you have specified as the source of the data). To change either date to view a particular range of data, double-click in the field. The calendar displays.

The title of this box reflects the date currently set. Underneath this is a bar listing the year; the arrows at either end of the date bar allow you to go forward and backward to find a different year. Underneath that is a similar bar with the month. Underneath that are buttons--one for each day of the month. Simply choose a button to select the appropriate date.

As you change the date, the title bar changes to reflect the new date you are specifying. Choose the OK button to save your change and exit back to the other dialog box.

The new dates appear in the Reports dialog box.

NOTE:

There is no date range for Metering Definitions.

8. In the Filter Criteria list box, specify the filter criteria you wish to use for your report.

The options include users, applications and protected files. You can select either one specific user or all users, and you can select either one specific application or all applications. The default will appear based on the report type you selected in Step 4. The list in Step 4 specifies if the default will be based on application data, user data or protected files data.

9. Once you have completed the above steps, choose the OK button.

The Export Status dialog box appears.

This dialog box contains the following information:

- * Server
- * Input Source
- * Input File
- * Output File
- * Output Format

NOTE:

If the data file you are creating already exists, then a message box will display asking "Do you wish to update your baseline?" If you wish to create another data file, choose the Yes button and the Export window displays. If you do not choose to update your baseline, then the Export window will not

display.

The metering capability exports 3 files for one report. During the export, you can choose the Abort button to halt the exporting. This button changes to Close once the export is complete. By choosing Close, the output file is put into the current directory.

10. When you close this dialog box, a window appears with the report name in the title bar.

See Also:

[Report Types](#)

[SiteMeter Report Screen](#)

The Report Screen

Once the Export dialog box closes, the SiteMeter Reports window displays. This section describes the parts of the report window.

The title bar displays the report type being generated. While the report is being generated on this window, a ribbon of buttons displays to the left. The two left most buttons are arrows pointing to the left. The left-most sends you to the beginning of the report; the other goes to the previous page. The next two buttons are arrows pointing right. The first goes to the next page, and the other to the end of the report. The fourth button is the stop button. Once the first page of the report is generated, this button becomes enabled. You can use it to halt the report generation at that point. After that there are two additional buttons. The first is page view; the second is a print button that prints the report. Most of these buttons are grayed during report generation.

The next section is a box displaying the following: Read (indicating the records read from the database), Selected (indicating those records in the specified date range or meeting other criteria), Total (indicating the total records in the database), and % (indicating the percentage of the report that is completed). All of these fields except Total change to reflect the real time report generation. The Next box tells you which page you are on, for example 1 of 18 or 2 of 18.

Beneath all of this is the actual report.

See Also:

[Report Types](#)

Printing from the Reporting Module

You have three options to print a generated report:

- * Print to Window
- * Print to Printer
- * Print to File

Use the following procedure to print reports:

1. From the File menu, choose Print Setup.

The Print dialog box appears.

This dialog box lets you customize the following information:

- * Printer
- * Print Range
- * Print Quality
- * Print to File
- * Copies
- * Copies Collated

Choose the Setup button to produce the standard Windows Print Setup dialog box to specify additional options. The Cancel button exits this dialog box without printing.

2. Specify any of the above criteria to customize your print job.

3. If you did not choose to print to a file, you can choose from the following two methods to send a print job to a printer:

1) To print to a printer, choose Reports in the Reports menu bar and then choose the Print command to print to a printer.

2) To print to a window, choose Reports in the Report menu bar and select print to window. Once the report is generated on the screen and the print button becomes enabled, choose the Print button to print.

Report Types

BrightWorks offers a number of different report types for application usage and software security information. Choose the report type best suited to your business and decision-making needs.

The following report types are offered:

- * [Metering Definitions Report](#)
- * [Application Summary Report](#)
- * [Application with User Detail Report](#)
- * [Application with User Summary Report](#)
- * [User Summary Report](#)
- * [User with Application Detail Report](#)
- * [User with Application Summary Report](#)
- * [File Integrity Activity Report](#)
- * [SPA Compliance Report](#)
- * [Software Purchase Forecast Report](#)
- * [Upgrade Purchase Forecast Report](#)

Metering Definitions Report

The following describes the report generated if you chose Metering Definitions as your report type:

The report heading details:

- *Date of report generation
- *Report type (i.e., Metering Definitions, etc.)
- *Applications on which report was generated

The report provides the following information about each application:

- *Path and file name being metered
- *Metered application name
- *Number of licenses available
- *Length of the queue back time
- *Whether or not a password is required for this metered application

Application Summary Report

The following describes the report generated if you chose Application Summary as your report type:

The report heading details:

- *Date of report generation
- *Report type
- *Date range for which report was generated
- *The server on which the report was generated
- *The applications on which the report was generated (will be either one application name or All)

The report provides the following details about each application:

- *Path and file name being metered
- *Metered application name
- *Number of licenses available
- *Peak number of concurrent users
- *Peak number of queued users
- *Total number of users
- *Total usage time
- *Total queued users
- *Percent utilization

This report also provides a grand summary of each of the above categories for all the applications included in the report.

Application with User Detail Report

The following describes the report generated if you chose Application with User Detail as your report type:

The report heading details:

- *Date of report generation
- *Report type
- *Date range for which report was generated
- *The server on which the report was generated
- *The users on which the report was generated (will be either one user name or All)

The report provides the following information for each application included in the report:

- *Path and file name being metered
- *Metered application name
- *Number of licenses available
- *Peak number of concurrent users
- *Peak number of queued users
- *Total number of users
- *Total usage time
- *Total queued users
- *Percent utilization

Beneath all of the above information about each application, the following detailed information is provided about each user who used the application:

- *Date and time of usage
- *User name
- *Status (whether the user was granted access or queued)
- *Total time in minutes of usage

NOTE:

The above information is provided each time a user accesses the application.

Application with User Summary Report

The following describes the report generated if you chose Application with User Summary as your report type:

The report heading details:

- *Date of report generation
- *Report type
- *Date range for which report was generated
- *The server on which the report was generated
- *The applications on which the report was generated (will be either one application name or All)

The report provides the following information for each application included in the report:

- *Path and file name being metered
- *Metered application name
- *Number of licenses available
- *Peak number of concurrent users
- *Peak number of queued users
- *Total number of users
- *Total usage time
- *Total queued users
- *Percent utilization

Beneath all of the above information about each application, the following summary information is provided about each user who used the application:

- *User name
- *Total usage
- *Total usage time
- *Total number of times queued
- *Total average queue time

User Summary Report

The following describes the report generated if you chose User Summary as your report type:

The report heading details:

- *Date of report generation
- *Report type
- *Date range for which report was generated
- *The server on which the report was generated
- *The users on which the report was generated (will be either one user name or All)

The report provides the following information about each user on which the report was generated:

- *User name
- *Total number of times queued
- *Total usage
- *Total usage time
- *Percent utilization

This report also provides a grand summary for each of the above categories for each user included in the report.

User with Application Detail Report

The following describes the report generated if you chose User with Application Detail as your report type:

The report heading details:

- *Date of report generation
- *Report type
- *Date range for which report was generated
- *The server on which the report was generated
- *The users on which the report was generated (will be either one user name or All)

The report provides the following information about each user on which the report was generated:

- *User name
- *Total number of times queued
- *Total usage time
- *Percent utilization

Beneath all of the above information about each user, the following detailed information is provided about each application this user used:

- *Date and time of usage
- *Application name
- *Status (whether the user was granted access or queued)
- *Total time in minutes of usage

NOTE:

The above information is provided for each time the user accessed an application.

User with Application Summary Report

The following describes the report generated if you chose User with Application Summary as your report type:

The report heading details:

- *Date of report generation
- *Report type
- *Date range for which report was generated
- *The server on which the report was generated
- *The users on which the report was generated (will be either one user name or All)

The report provides the following information about each user on which the report was generated:

- *User name
- *Total number of times queued
- *Total usage
- *Total usage time
- *Percent utilization

Beneath all of the above information about each user, the following summary information is provided about each application the user used:

- *Path and file name being metered
- *Metered application name
- *Number of licenses available
- *Total number of times queued
- *Total usage
- *Total usage time
- *Percent utilization

File Integrity Activity Report

The following describes the report generated if you chose File Integrity Activity as your report type:

The report heading details:

- *Date of report generation
- *Report type
- *Date range for which report was generated
- *The server on which the report was generated
- *The users on which the report was generated (will be either one user name or All)
- *The applications on which the report was generated (will be either one application name or All)

This report provides the following information about the software security violations:

- *Action detected
- *User
- *File name
- *Date and time
- *Network name
- *Station

SPA Compliance Report

The following describes the report generated if you chose SPA Compliance as your report type:

The report heading details:

- *Date of report generation
- *Report type
- *The server on which the report was generated
- *The applications on which the report was generated (will be either one application name or All)

The report provides the following information about metered applications:

- *Metered application name
- *File name
- *Number of licenses available
- *Peak queued users
- *Peak usage

Software Purchase Forecast Report

The following describes the report generated if you chose Software Purchase Forecast as your report type:

The report heading details:

- *Date of report generation
- *Report type
- *The server on which the report was generated
- *The applications on which the report was generated (will be either one application name or All)

The report provides the following information about each application on which the report was generated:

- *Metered application name
- *File name
- *Number of licenses installed
- *Peak queued users
- *Number of surplus licenses (number of purchases to accommodate current need)
- *10% Surplus Recommendation (purchase required to accommodate the current need plus a 10% increase)
- *20% Surplus Recommendation (purchase required to accommodate the current need plus a 20% increase)

Upgrade Purchase Forecast Report

The following describes the report generated if you chose Upgrade Purchase Forecast as your report type:

The report heading details:

- *Date of report generation
- *Report type
- *The server on which the report was generated
- *The applications on which the report was generated (will be either one application name or All)

The report provides the following information about each application on which the report was generated:

- *Metered application name
- *File name
- *Number of licenses installed
- *Peak usage
- *Number of surplus licenses (number of purchases to accommodate current need)
- *10% Surplus Recommendation (purchase required to accommodate the current need plus a 10% increase)
- *20% Surplus Recommendation (purchase required to accommodate the current need plus a 20% increase)

Glossary

A

Activity Date Range

Application

Application Summary

Application with User Detail

Application with User Summary

Authorized Files

C

Current Home Directory

Current Server

D

Directories in Which Trustee Rights are Granted During Execution

Disable Local Drives

E

Export

F

File Integrity Activity

File Integrity Status

File Scan Interval

Files to Meter

Filter Criteria

Full Name

H

Home Directory

I

Input File

Input Source

L

Logical Station

Login Name

M

Maximum Number of Concurrent Users

Metered Application Name

Metering Definitions

N

Network

O

Output File

Output Format

P

Password

Print Destination

Print to File

Print to Window

Print to Printer

Q

Queue Back Time

R

Report Type

Restrict Local Execution

Rights for Named File Only

Run Unauthorized Files

S

[Security Exceptions](#)

[Security Scan Interval](#)

[Server](#)

[SiteMeter NLM](#)

[SiteMeter Proxy NLM](#)

[Software Purchase Forecast](#)

[Source](#)

[SPA Compliance](#)

[Space Separated](#)

[Station Address](#)

[Status of Swatcher](#)

[Swatcher Users](#)

T

[Tab Separated](#)

[Time into Application](#)

[Time into Network](#)

U

[Unauthorized Files](#)

[Upgrade Purchase Forecast](#)

[User Summary](#)

[User with Application Detail](#)

[User with Application Summary](#)

W

[Window](#)

Activity Date Range

In the Report dialog box, Activity Date Range specifies the start and end dates for the data used in the report being generated. The default start date is the very first date of the stored data; the default end date is today's date. You can specify either date if you do not wish to use these defaults.

Application

In the Current User Information and the Queued User Information dialog boxes, application indicates which application the user is either currently using or waiting to use.

Application Summary

The Application Summary report provides a summary of all the activity for each metered application. From this report, a user can get an overview of activity for each metered application.

Application with User Detail

The Application with User Detail report provides detailed information about a user's activity for each metered application. For example, this report details when a user accessed an application and for how long he or she used it.

Application with User Summary

The Application with User Summary report provides a summary of all the activity for each metered application, as well as an overview of a network user's use of each metered application.

Authorized Files

An authorized file is a file that has been registered for file integrity scanning. If a file is not an authorized file and you do NOT allow unauthorized files to be run, it will not be allowed to execute.

Metering records the characteristics of each authorized file and stores this value in the NetWare bindery.

Current Home Directory

This metering setting indicates the directory where the SITEDATA, VIRUSDTA, and SMRPROXY files are located.

Current Server

Current Server reflects the server to which you are currently attached. You can attach to other file servers using the Attach button.

Directories in Which Trustee Rights are Granted During Execution

Directories in Which Trustee Rights are Granted During Execution option that lets you grant temporary rights (trustee assignments) while an application (or suite) is running.

Disable Local Drives

This option allows you to specify whether or not to disable local drives entirely, essentially rendering the PC diskless. Disable Local Drives is only available if you use Swatcher, the workstation security agent method for metering and file protecting your LAN.

Export

This SREPORT option lets you export coded data into standard database formats. In the Export dialog box you specify the source file, the destination and which database format you wish to use.

File Integrity Activity

The File Integrity Activity report provides a listing of all the activity on the virus secured files and applications. It also gives details about who used the secured files, where these files were used and the actions taken by SiteMeter on these files.

File Integrity Status

This metering setting indicates how often authorized files are checked for any changes. The file is compared with a fingerprint of the file to detect any alterations.

File Scan Interval

File Scan Interval tells BrightWorks how often to check the executable program against the registered copy of that file.

To check a file every time it is requested, set the File Scan Interval to zero (0). If your file server has heavy network traffic, however, you may want to adjust this value.

The value can range from 0 to 1440 minutes (once every 24 hours). The value you set applies to ALL authorized files.

Regardless of the value, the file is always checked against the registered copy the first time it is requested. If the field is set to 15 minutes, however, no matter how many times the file is executed it will not be checked again for 15 minutes after the first check. The first attempt to run the software after the 15 minute interval will reset the time interval.

For example, you run LOTUS for the first time at 11:00 a.m. at which time BrightWorks metering checks the file. The next time the file will be checked will be the first time it is requested after 11:15 a.m. (if the File Scan Interval has been set to 15).

Files to Meter

Files to Meter indicates the list of files to be registered. For example, WP.EXE. You can also meter suites of applications, such as Microsoft Office, to ensure accurate license compliance. The executables for the applications in a suite would appear in this section.

Filter Criteria

In the Reports dialog box, Filter Criteria lets you restrict your report to a certain application or a certain user. You can also run the report on all applications, all users or all protected files, depending upon the report type you choose.

Full Name

In the Add Metered Application dialog box, Full Name indicates the entire name of the product or application. This field is ideal for entering descriptive information. For example, WordPerfect Version 5.1 for DOS.

In either the Current Users Information or Queued Users Information dialog boxes, Full Name indicates the full name of the current or queued user who was selected.

Home Directory

This option allows you to define where the SiteMeter files reside.

The default directory determined at installation time is SYS:\SYSTEM\SITEMETR. However, you can define the directory of your choice using this option.

Users need Read, Open, Search, Write, and Create rights to this directory.

Input File

In the Export dialog box, Input File indicates if the data source was Virusdta or Sitedata. (Input File will be blank if Metering Definitions was selected.)

Input Source

In the Export dialog box, Input Source indicates if the data source was Metering Definitions. (Input Source will be blank if Sitedata or Virusdta was selected.)

Logical Station

In either the Current Users Information or Queued Users Information dialog boxes, logical station indicates the station number that is arbitrarily assigned to a workstation.

Login Name

In either the Current Users Information or Queued Users Information dialog boxes, login name indicates the user's login name on the network.

Maximum Number of Concurrent Users

Maximum Number of Users indicates the total number of licenses purchased for this application.

Metered Application Name

Metered Application Name indicates the name of the application(s) to be registered. For example, WordPerfect.

Metering Definitions

The Metering Definitions report provides the network manager with a listing of all the applications currently being metered. For each application, it also provides the number of licenses, whether or not it is password-protected, and any other definitions specific to this application.

Network

In either the Current Users Information or the Queued Users Information dialog boxes, network indicates the network number of the workstation where this user is located.

Output File

In the Export dialog box, Output File indicates the name of the file to which the data is being exported.

Output Format

In the Export dialog box, Output Format indicates in what format the exported data will appear.

Password

Password is an optional password that restricts access to the metering configuration for this metered application. If you decide to assign a password to an application or to a suite of applications, you must supply this password before configuring this metered application (or suite) again.

Print Destination

In the Reports dialog box, Print Destination specifies where you want your report to go. You can send a report to a file, window or printer.

Print to File

This SREPORT printing option sends the report directly to a file that you specify.

Print to Window

This SREPORT printing option sends the report directly to a the window; from this point you can send it to a printer using the Print button.

Print to Printer

This SREPORT printing option sends the report directly to a printer, which you specify.

Queue Back Time

Queue Back Time indicates the amount of time that the application (or suite) is held exclusively for a user waiting in the queue.

Report Type

SREPORT offers a number of different report types for your data. You can choose from twelve different formats for your report; each offers data about metered applications and their usage on the network. Select the report type best-suited to your business or decision-making needs.

Restrict Local Execution

BrightWorks metering provides an option that restricts execution of applications from local drives. By using this option, you can disallow network users from running applications or other programs from the hard drive. Users will still be able to access their local drives, but will not be able to run any applications locally. This feature provides an added layer of control over software usage on your network.

This option is only available if you are using Swatcher, the workstation security agent method of metering and file protecting your LAN.

Rights for Named File Only

Secured Directory rights are available only for the file whose execution is being tracked. For example, if the application is WordPerfect (WP.EXE), you have Secured Directory rights while running this application. However, if you issue the DOS Shell command and exit into DOS, all rights are revoked until you EXIT back to WordPerfect.

Run Unauthorized Files

This option instructs BrightWorks whether or not to permit execution of currently unauthorized files on the network. Using this option prevents unauthorized software from being run on the network. When this option is enabled, only the listed application files are allowed to run. The Specify Policy dialog box lets you specify on which file server(s) you wish to allow or disallow unauthorized files to run.

Security Exceptions

If you are using Swatcher to meter and file protect your LAN, this option allows you to specify those users who are not required to load Swatcher.

Security Exceptions are those users who are not required to load the Swatcher TSR when logging in to the network.

When using the Swatcher TSR method of metering and file protecting, you can set a Security Scan Interval. This interval instructs the metering capability how often it should check to verify that all users have loaded the Swatcher TSR, except those specified in the Security Exceptions list.

If a user is a member of this list, he or she will not be disconnected from the network if the metering capability finds that he or she does not have the Swatcher TSR loaded.

Security Scan Interval

The Security Scan Interval is the length of time between Security Scan checks. This value indicates how frequently the metering capability scans the network to be sure users on the network either have loaded the Swatcher TSR or are listed as Security Exceptions.

If a user is not a Security Exception and has not loaded Swatcher, the metering capability sends a NetWare Send message to the user indicating that he or she will be logged off the file server in 30 seconds. This allows the user enough time to save his or her work before being disconnected automatically from the network. The user must load Swatcher before logging in to the network again.

Server

In the Export dialog box, Server indicates the server from which the data is being exported. This changes to reflect real time if more than one file server is being read for the export.

In either the Current Users Information or Queued Users Information dialog boxes, server indicates the file server to which this user is attached.

SiteMeter NLM

This metering setting indicates whether the SiteMeter NLM is active (loaded) or inactive.

SiteMeter Proxy NLM

This metering setting indicates whether the SiteMeter Proxy NLM is active (loaded) or inactive.

Software Purchase Forecast

The Software Purchase Forecast report recommends software purchases for metered applications based on user activity and the perceived user need as indicated by the number of queued users.

Source

In the Reports dialog box, Source indicates from where you want to take the data for your report. The available options are Sitedata, Virusdta and Metering Definitions.

SPA Compliance

The SPA Compliance report is accepted by the Software Publisher's Association in conjunction with software purchase records as proof of software license compliance in networked environments.

Space Separated

When you choose this format as the print destination, SREPORT saves the data in ASCII text format with all values separated by spaces.

Station Address

In either the Current Users Information or Queued Users Information dialog boxes, station address indicates the node address of the workstation where this user is located.

Status of Swatcher

This metering setting indicates whether or not Swatcher is loaded on the workstation.

Swatcher Users

This metering setting indicates whether users are checked to verify that the Swatcher TSR is loaded and also indicates the time interval in which they are checked.

Tab Separated

When you choose this format as the print destination, SREPORT saves the data in ASCII text format with all values separated by tabs.

Time into Application

In either the Current Users Information or the Queued Users Information dialog boxes, time into application indicates the date and time the user selected launched (or tried to launch and became queued) the application.

Time into Network

In either the Current Users Information or the Queued users Information dialog boxes, time into network indicates the length of time this user has been logged in to the network.

Unauthorized Files

Unauthorized Files are those files that are not checked for viruses by the file integrity scanning feature. In the View Metering Status dialog box, unauthorized files indicate whether such files are allowed to run on the network.

Upgrade Purchase Forecast

The Upgrade Purchase Forecast report recommends software upgrades for metered applications based on user activity and the perceived user need as indicated by the number of queued users.

User Summary

The User Summary report provides the network manager with a listing of all the users who used metered applications. It offers crucial information, such as the number of times a user was queued.

User with Application Detail

The User with Application Detail report provides information about each time a user accessed a metered application.

User with Application Summary

The User with Application Summary report provides summarized information for each metered application that a user used.

Administration

The Administration menu lists sub-menus for each BrightWorks capability. When selected, each sub-menu displays a list of commands used for configuring and managing the capability parameters.

Alerting

The Alerting tool bar button displays the Alerting Options dialog box used for defining and scheduling auditing alerts.

Distribute

The Distribute tool bar button displays the Available Packages dialog box used for viewing, creating and managing packages for distribution.

Edit

The Edit menu lists the standard editing commands used when creating and editing software distribution scripts.

File

The File menu lists file oriented commands, including those for configuring global print settings and exiting BrightWorks. For several of the File menu commands, the command text and its associated action when selected depend on the currently active window in the BrightWorks application window. For example, when the List of Available Scripts window is active, the New Script and Open Script commands are listed under the File menu.

Help

The Help menu lists commands to access BrightWorks online Windows hypertext help.

Inventory

The Inventory tool bar button displays the View Inventory dialog box used for viewing and managing the inventory of each audited workstation.

Metering

The Metering tool bar button displays the Define Metered Applications dialog box used for adding, modifying and deleting applications to be metered.

Monitor

The Monitor tool bar button displays the View Application Usage dialog box used for determining which applications are being used and by whom.

Remote

The Remote tool bar button provides access to optional McAfee NETremote+ software for automated user support.

Reports

The Reports menu lists commands for accessing specific categories of BrightWorks reports.

The Reports tool bar button displays the report generator that was last selected from the Report menu. For example, if the Inventory and Distribution command was last chosen from the Reports menu, then the Choose Report dialog box will display, enabling you to generate inventory and distribution reports.

Security

The Security tool bar button displays the Define Authorized Files dialog box used for specifying files that are secure.

Tickets

The Tickets tool bar button provides access to optional McAfee LAN Support Center software for help desk automation.

Tools

The Tools menu lists commands for viewing and managing capability-specific information. For example, choose the Inventory command to view and manage the inventory collected by BrightWorks Inventory capability.

Window

The Window menu lists commands for positioning the open document windows in the BrightWorks application window.

