

Contents

Windows Commands

[Command-Line Syntax Key](#)

[Commands by Server Role](#)

[Print Command Reference](#)

[Services for Network File System Command Reference](#)

[Remote Desktop Services \(Terminal Services\) Command Reference](#)

[Windows Server Backup Command Reference](#)

[append](#)

[arp](#)

[assoc](#)

[at](#)

[atmadm](#)

[attrib](#)

[auditpol](#)

[autochk](#)

[autoconv](#)

[autofmt](#)

[bcdboot](#)

[bcdedit](#)

[bdehdcfg](#)

[bitsadmin](#)

[bitsadmin addfile](#)

[bitsadmin addfileset](#)

[bitsadmin addfilewithranges](#)

[bitsadmin cancel](#)

[bitsadmin complete](#)

[bitsadmin create](#)

[bitsadmin getaclflags](#)

[bitsadmin getbytestotal](#)

bitsadmin getbytestransferred
bitsadmin getcompletiontime
bitsadmin getcreationtime
bitsadmin getdescription
bitsadmin getdisplayname
bitsadmin geterror
bitsadmin geterrorcount
bitsadmin getfilestotal
bitsadmin getfilestransferred
bitsadmin getminretrydelay
bitsadmin getmodificationtime
bitsadmin getnoprogresstimeout
bitsadmin getnotifycmdline
bitsadmin getnotifyflags
bitsadmin getnotifyinterface
bitsadmin getowner
bitsadmin get priority
bitsadmin getproxybypasslist
bitsadmin getproxylist
bitsadmin getproxyusage
bitsadmin getreplydata
bitsadmin getreplyfilename
bitsadmin getreplyprogress
bitsadmin getstate
bitsadmin gettype
bitsadmin help
bitsadmin info
bitsadmin list
bitsadmin listfiles
bitsadmin monitor
bitsadmin nowrap
bitsadmin rawreturn

- bitsadmin removecredentials
- bitsadmin replaceremoteprefix
- bitsadmin reset
- bitsadmin resume
- bitsadmin setaclflag
- bitsadmin setcredentials
- bitsadmin setdescription
- bitsadmin setdisplayname
- bitsadmin setminretrydelay
- bitsadmin setnoprogresstimeout
- bitsadmin setnotifycmdline
- bitsadmin setnotifyflags
- bitsadmin setpriority
- bitsadmin setproxysettings
- bitsadmin setreplyfilename
- bitsadmin suspend
- bitsadmin takeownership
- bitsadmin Transfer
- bitsadmin util
- bitsadmin wrap

bootcfg

- bootcfg addsw
- bootcfg copy
- bootcfg dbg1394
- [bootcfg debug](bootcfg-debug.md)
- bootcfg default
- bootcfg delete
- bootcfg ems
- bootcfg query
- bootcfg raw
- bootcfg rmsw
- bootcfg timeout

break
cacs
call
cd
certreq
certutil
change
 change logon
 change port
 change user
chcp
chdir
chglogon
chgport
chgusr
chkdsk
chkntfs
choice
cipher
clip
cls
Cmd
cmdkey
cmstp
color
comp
compact
convert
copy
cprofile
cscript
date

dcgpofix
defrag
del
dfsrmig
diantz
dir
diskcomp
diskcopy
diskpart
diskperf
diskraid
diskshadow
dispdiag
Dnscmd
doskey
driverquery
echo
edit
endlocal
erase
eventcreate
eventquery
eventtriggers
Evntcmd
exit
expand
extract
fc
find
findstr
finger
flattemp

fondue

for

forfiles

format

freedisk

fsutil

fsutil 8dot3name

fsutil behavior

fsutil file

fsutil fsinfo

fsutil hardlink

fsutil objectid

fsutil quota

fsutil repair

fsutil reparsepoint

fsutil resource

fsutil sparse

fsutil tiering

fsutil transaction

fsutil usn

fsutil volume

fsutil wim

ftype

fveupdate

getmac

gettype

goto

gpfixup

gpresult

gpupdate

grftabl

help

helpctr

hostname

icacls

if

inuse

ipconfig

ipxroute

irftp

jetpack

klist

ksetup

Ksetup:setrealm

Ksetup:mapuser

Ksetup:addkdc

Ksetup:delkdc

Ksetup:addkpasswd

Ksetup:delkpasswd

Ksetup:server

Ksetup:setcomputerpassword

Ksetup:removerealm

Ksetup:domain

Ksetup:changepassword

Ksetup:listrealmflags

Ksetup:setrealmflags

Ksetup:addrealmflags

Ksetup:delrealmflags

Ksetup:dumpstate

Ksetup:addhosttorealmmap

Ksetup:delhosttorealmmap

Ksetup:setenctypeattr

Ksetup:getenctypeattr

Ksetup:addenctypeattr

- Ksetup:delenctypeattr
- ktmutil
- ktpass
- label
- lodctr
- logman
 - logman create
 - logman query
 - logman start | stop
 - logman delete
 - logman update
 - logman import | export
- logoff
- lpq
- lpr
- macfile
- makecab
- manage-bde
 - Manage-bde: status
 - Manage-bde: on
 - Manage-bde: off
 - Manage-bde: pause
 - Manage-bde: resume
 - Manage-bde: lock
 - Manage-bde: unlock
 - Manage-bde: autounlock
 - Manage-bde: protectors
 - Manage-bde: tpm
 - Manage-bde: setidentifier
 - Manage-bde: ForceRecovery
 - Manage-bde: changepassword
 - Manage-bde: changepin

Manage-bde: changekey

Manage-bde: KeyPackage

Manage-bde: upgrade

Manage-bde: WipeFreeSpace

mapadmin

Md

mkdir

mklink

mmc

mode

more

mount

mountvol

move

mqbkup

mqsvc

mqtgsvc

msdt

msg

msiexec

msinfo32

mstsc

nbtstat

netcfg

netsh

netstat

Net print

nfsadmin

nfsshare

nfsstat

nlbmgr

nslookup

nslookup exit Command
nslookup finger Command
nslookup help
nslookup ls
nslookup lserver
nslookup root
nslookup server
nslookup set
nslookup set all
nslookup set class
nslookup set d2
nslookup set debug
nslookup set domain
nslookup set port
nslookup set querytype
nslookup set recurse
nslookup set retry
nslookup set root
nslookup set search
nslookup set srchlist
nslookup set timeout
nslookup set type
nslookup set vc
nslookup view
ntbackup
ntcmdprompt
ntfrsutl
openfiles
pagefileconfig
path
pathping
pause

pbadmin

pentnt

perfmon

ping

pnpunattend

pnputil

popd

PowerShell

PowerShell_ise

print

prncnfg

prndrvr

prnjobs

prnmngr

prnport

prnqctl

prompt

pubprn

pushd

pushprinterconnections

qappsrv

qprocess

query

quser

qwinsta

rcp

rd

rdpsign

recover

reg

Reg add

Reg compare

- Reg copy
- Reg delete
- Reg export
- Reg import
- Reg load
- Reg query
- Reg restore
- Reg save
- Reg unload
- regini
- regsvr32
- relog
- rem
- ren
- rename
- repair-bde
- replace
- reset session
- rexec
- risetup
- rmdir
- robocopy
- route_ws2008
- rpcinfo
- rpcping
- rsh
- rundll32
- rwinsta
- schtasks
- Scwcmd
 - Scwcmd: analyze
 - Scwcmd: configure

Scwcmd: register

[Scwcmd: rollback](scwcmd-rollback.md)

[Scwcmd: transform](scwcmd-transform.md)

[Scwcmd: view](scwcmd-view.md)

secedit

Secedit:analyze

Secedit:configure

Secedit:export

Secedit:generaterollback

Secedit:import

Secedit:validate

serverceipoptin

Servermanagercmd

set

setlocal

setx

sfc

shadow

shift

showmount

shutdown

sort

start

subst

sxstrace

sysocmgr

systeminfo

takeown

tapicfg

taskkill

tasklist

tcmsetup

telnet

tftp

time

timeout

title

tlntadmn

tpmvscmgr

tracert

tracert

tree

tscon

tsdiscon

tsecimp

tskill

tsprof

type

typeperf

tzutil

vssadmin

vssadmin delete shadows

vssadmin list shadows

vssadmin list writers

unlodctr

ver

verifier

verify

vol

waitfor

wbadmin

Wbadmin enable backup

Wbadmin disable backup

Wbadmin start backup

Wbadmin stop job

Wbadmin get versions

Wbadmin get items

Wbadmin start recovery

Wbadmin get status

Wbadmin get disks

Wbadmin start systemstaterecovery

Wbadmin start systemstatebackup

Wbadmin delete systemstatebackup

Wbadmin start sysrecovery

Wbadmin restore catalog

Wbadmin delete catalog

wdsutil

wecutil

wevtutil

where

whoami

winnt

winnt32

winpop

winrs

wmic

wscript

xcopy

Windows Commands

11/14/2018 • 6 minutes to read • [Edit Online](#)

All supported versions of Windows (server and client) have a set of Win32 console commands built in.

This set of documentation describes the Windows Commands you can use to automate tasks by using scripts or scripting tools.

To find information about a specific command, in the following A-Z menu, click the letter that the command starts with, and then click the command name.

[A](#) | [B](#) | [C](#) | [D](#) | [E](#) | [F](#) | [G](#) | [H](#) | [I](#) | [J](#) | [K](#) | [L](#) | [M](#) | [N](#) | [O](#) | [P](#) | [Q](#) | [R](#) | [S](#) | [T](#) | [U](#) | [V](#) | [W](#) | [X](#) | [Y](#) | [Z](#)

Prerequisites

The information that is contained in this PDF applies to:

- Windows Server 2019
- Windows Server (Semi-Annual Channel)
- Windows Server 2016
- Windows Server 2012 R2
- Windows Server 2012
- Windows Server 2008 R2
- Windows Server 2008
- Windows 10
- Windows 8.1

Command shell overview

The Command shell was the first shell built into Windows to automate routine tasks, like user account management or nightly backups, with batch (.bat) files. With Windows Script Host you could run more sophisticated scripts in the Command shell. For more information, see [cscript](#) or [wscript](#). You can perform operations more efficiently by using scripts than you can by using the user interface. Scripts accept all Commands that are available at the command line.

Windows has two command shells: The Command shell and [PowerShell](#). Each shell is a software program that provides direct communication between you and the operating system or application, providing an environment to automate IT operations.

PowerShell was designed to extend the capabilities of the Command shell to run PowerShell commands called cmdlets. Cmdlets are similar to Windows Commands but provide a more extensible scripting language. You can run Windows Commands and PowerShell cmdlets in Powershell, but the Command shell can only run Windows Commands and not PowerShell cmdlets.

For the most robust, up-to-date Windows automation, we recommend using PowerShell instead of Windows Commands or Windows Script Host for Windows automation.

NOTE

You can also download and install [PowerShell Core](#), the open source version of PowerShell.

Incorrectly editing the registry may severely damage your system. Before making the following changes to the registry, you should back up any valued data on the computer.

NOTE

To enable or disable file and directory name completion in the Command shell on a computer or user logon session, run **regedit.exe** and set the following **reg_DWORD** value:

HKEY_LOCAL_MACHINE\Software\Microsoft\Command Processor\completionChar\reg_DWORD

To set the **reg_DWORD** value, use the hexadecimal value of a control character for a particular function (for example, **0 9** is Tab and **0 08** is Backspace). User-specified settings take precedence over computer settings, and command-line options take precedence over registry settings.

Command-line reference A-Z

To find information about a specific Windows Command, in the following A-Z menu, click the letter that the Command starts with, and then click the Command name.

[A](#) | [B](#) | [C](#) | [D](#) | [E](#) | [F](#) | [G](#) | [H](#) | [I](#) | [J](#) | [K](#) | [L](#) | [M](#) | [N](#) | [O](#) | [P](#) | [Q](#) | [R](#) | [S](#) | [T](#) | [U](#) | [V](#) | [W](#) | [X](#) | [Y](#) | [Z](#)

A

- [append](#)
- [arp](#)
- [assoc](#)
- [at](#)
- [atmadm](#)
- [attrib](#)
- [auditpol](#)
- [autochk](#)
- [autoconv](#)
- [autofmt](#)

B

- [bcdboot](#)
- [bcdedit](#)
- [bdehdcfg](#)
- [bitsadmin](#)
 - [bitsadmin addfile](#)
 - [bitsadmin addfilesset](#)
 - [bitsadmin addfilewithranges](#)
 - [bitsadmin cancel](#)
 - [bitsadmin complete](#)
 - [bitsadmin create](#)
 - [bitsadmin getaclflags](#)
 - [bitsadmin getbytestotal](#)
 - [bitsadmin getbytestransferred](#)
 - [bitsadmin getcompletiontime](#)
 - [bitsadmin getcreationtime](#)
 - [bitsadmin getdescription](#)
 - [bitsadmin getdisplayname](#)

- `bitsadmin geterror`
- `bitsadmin geterrorcount`
- `bitsadmin getfilestotal`
- `bitsadmin getfilestransferred`
- `bitsadmin getminretrydelay`
- `bitsadmin getmodificationtime`
- `bitsadmin getnopprogresstimeout`
- `bitsadmin getnotifycmdline`
- `bitsadmin getnotifyflags`
- `bitsadmin getnotifyinterface`
- `bitsadmin getowner`
- `bitsadmin get priority`
- `bitsadmin getproxybypasslist`
- `bitsadmin getproxylst`
- `bitsadmin getproxyusage`
- `bitsadmin getreplydata`
- `bitsadmin getreplyfilename`
- `bitsadmin getreplyprogress`
- `bitsadmin getstate`
- `bitsadmin gettype`
- `bitsadmin help`
- `bitsadmin info`
- `bitsadmin list`
- `bitsadmin listfiles`
- `bitsadmin monitor`
- `bitsadmin nowrap`
- `bitsadmin rawreturn`
- `bitsadmin removecredentials`
- `bitsadmin replaceremoteprefix`
- `bitsadmin reset`
- `bitsadmin resume`
- `bitsadmin setaclflag`
- `bitsadmin setcredentials`
- `bitsadmin setdescription`
- `bitsadmin setdisplayname`
- `bitsadmin setminretrydelay`
- `bitsadmin setnopprogresstimeout`
- `bitsadmin setnotifycmdline`
- `bitsadmin setnotifyflags`
- `bitsadmin setpriority`
- `bitsadmin setproxysettings`
- `bitsadmin setreplyfilename`
- `bitsadmin suspend`
- `bitsadmin takeownership`
- `bitsadmin Transfer`
- `bitsadmin util`

- bitsadmin wrap
- bootcfg
 - bootcfg addsw
 - bootcfg copy
 - bootcfg dbg1394
 - bootcfg debug
 - bootcfg default
 - bootcfg delete
 - bootcfg ems
 - bootcfg query
 - bootcfg raw
 - bootcfg rmsw
 - bootcfg timeout
- break

C

- cacs
- call
- cd
- certreq
- certutil
- change
 - change logon
 - change port
 - change user
- chcp
- chdir
- chglogon
- chgport
- chgusr
- chkdisk
- chkntfs
- choice
- cipher
- clip
- cls
- Cmd
- cmdkey
- cmstp
- color
- comp
- compact
- convert
- copy
- cprofile
- cscript

D

- [date](#)
- [dcgpofix](#)
- [defrag](#)
- [del](#)
- [dfsrmig](#)
- [diantz](#)
- [dir](#)
- [diskcomp](#)
- [diskcopy](#)
- [diskpart](#)
- [diskperf](#)
- [diskraid](#)
- [diskshadow](#)
- [dispdiag](#)
- [dnscmd](#)
- [doskey](#)
- [driverquery](#)

E

- [echo](#)
- [edit](#)
- [endlocal](#)
- [erase](#)
- [eventcreate](#)
- [eventquery](#)
- [eventtriggers](#)
- [evntcmd](#)
- [exit](#)
- [expand](#)
- [extract](#)

F

- [fc](#)
- [find](#)
- [findstr](#)
- [finger](#)
- [flattemp](#)
- [fondue](#)
- [for](#)
- [forfiles](#)
- [format](#)
- [freedisk](#)
- [fsutil](#)
 - [fsutil 8dot3name](#)
 - [fsutil behavior](#)
 - [fsutil file](#)
 - [fsutil fsinfo](#)

- fsutil hardlink
- fsutil objectid
- fsutil quota
- fsutil repair
- fsutil reparsepoint
- fsutil resource
- fsutil sparse
- fsutil tiering
- fsutil transaction
- fsutil usn
- fsutil volume
- fsutil wim
- ftp
- ftype
- fveupdate

G

- getmac
- gettype
- goto
- gpfixup
- gpresult
- gpupdate
- graftabl

H

- help
- helpctr
- hostname

I

- icacLS
- if
- inuse
- ipconfig
- ipxroute
- irftp

J

- jetpack

K

- klist
- ksetup
 - ksetup:setrealm
 - ksetup:mapuser
 - ksetup:addkdc
 - ksetup:delkdc
 - ksetup:addkpasswd

- [ksetup:delkpasswd](#)
- [ksetup:server](#)
- [ksetup:setcomputerpassword](#)
- [ksetup:removerealm](#)
- [ksetup:domain](#)
- [ksetup:changepassword](#)
- [ksetup:listrealmflags](#)
- [ksetup:setrealmflags](#)
- [ksetup:addrealmflags](#)
- [ksetup:delrealmflags](#)
- [ksetup:dumpstate](#)
- [ksetup:addhosttorealmmap](#)
- [ksetup:delhosttorealmmap](#)
- [ksetup:setenctypeattr](#)
- [ksetup:getenctypeattr](#)
- [ksetup:addenctypeattr](#)
- [ksetup:delenctypeattr](#)
- [ktmutil](#)
- [ktpass](#)

L

- [label](#)
- [lodctr](#)
- [logman](#)
 - [logman create](#)
 - [logman query](#)
 - [logman start &124; stop](#)
 - [logman delete](#)
 - [logman update](#)
 - [logman import &124; export](#)
- [logoff](#)
- [lpq](#)
- [lpr](#)

M

- [macfile](#)
- [makecab](#)
- [manage-bde](#)
 - [manage-bde: status](#)
 - [manage-bde: on](#)
 - [manage-bde: off](#)
 - [manage-bde: pause](#)
 - [manage-bde: resume](#)
 - [manage-bde: lock](#)
 - [manage-bde: unlock](#)
 - [manage-bde: autounlock](#)
 - [manage-bde: protectors](#)

- manage-bde: tpm
- manage-bde: setidentifier
- manage-bde: ForceRecovery
- manage-bde: changepassword
- manage-bde: changepin
- manage-bde: changekey
- manage-bde: KeyPackage
- manage-bde: upgrade
- manage-bde: WipeFreeSpace
- mapadmin
- Md
- mkdir
- mklink
- mmc
- mode
- more
- mount
- mountvol
- move
- mqbkup
- mqsvc
- mqtgsvc
- msdt
- msg
- msixexec
- msinfo32
- mstsc

N

- nbtstat
- netcfg
- netsh
- netstat
- Net print
- nfsadmin
- nfsshare
- nfsstat
- nlbmgr
- nslookup
 - nslookup exit command
 - nslookup finger command
 - nslookup help
 - nslookup ls
 - nslookup lserver
 - nslookup root
 - nslookup server
 - nslookup set

- nslookup set all
- nslookup set class
- nslookup set d2
- nslookup set debug
- nslookup set domain
- nslookup set port
- nslookup set querytype
- nslookup set recurse
- nslookup set retry
- nslookup set root
- nslookup set search
- nslookup set srchlist
- nslookup set timeout
- nslookup set type
- nslookup set vc
- nslookup view
- ntbackup
- ntcmdprompt
- ntfrsutl

O

- openfiles

P

- pagefileconfig
- path
- pathping
- pause
- pbadmin
- pentnt
- perfmon
- ping
- pnpunattend
- pnputil
- popd
- PowerShell
- PowerShell_ise
- print
- prncnfg
- prndrvr
- prnjobs
- prnmngr
- prnport
- prnqctl
- prompt
- pubprn
- pushd

- [pushprinterconnections](#)

Q

- [qappsrv](#)
- [qprocess](#)
- [query](#)
- [quser](#)
- [qwinsta](#)

R

- [rcp](#)
- [rd](#)
- [rdpsign](#)
- [recover](#)
- [reg](#)
 - [reg add](#)
 - [reg compare](#)
 - [reg copy](#)
 - [reg delete](#)
 - [reg export](#)
 - [reg import](#)
 - [reg load](#)
 - [reg query](#)
 - [reg restore](#)
 - [reg save](#)
 - [reg unload](#)
- [regini](#)
- [regsvr32](#)
- [relog](#)
- [rem](#)
- [ren](#)
- [rename](#)
- [repair-bde](#)
- [replace](#)
- [reset session](#)
- [rexec](#)
- [risetup](#)
- [rmdir](#)
- [robocopy](#)
- [route_ws2008](#)
- [rpcinfo](#)
- [rpcping](#)
- [rsh](#)
- [rundll32](#)
- [rwinsta](#)

S

- [schtasks](#)

- [scwcmd](#)
 - [scwcmd: analyze](#)
 - [scwcmd: configure](#)
 - [scwcmd: register](#)
 - [scwcmd: rollback](#)
 - [scwcmd: transform](#)
 - [scwcmd: view](#)
- [secedit](#)
 - [secedit:analyze](#)
 - [secedit:configure](#)
 - [secedit:export](#)
 - [secedit:generaterollback](#)
 - [secedit:import](#)
 - [secedit:validate](#)
- [serverceipoptin](#)
- [Servermanagercmd](#)
- [serverweroptin](#)
- [set](#)
- [setlocal](#)
- [setx](#)
- [sfc](#)
- [shadow](#)
- [shift](#)
- [showmount](#)
- [shutdown](#)
- [sort](#)
- [start](#)
- [subst](#)
- [sxstrace](#)
- [sysocmgr](#)
- [systeminfo](#)

T

- [takeown](#)
- [tapicfg](#)
- [taskkill](#)
- [tasklist](#)
- [tcmsetup](#)
- [telnet](#)
- [tftp](#)
- [time](#)
- [timeout](#)
- [title](#)
- [tlntadm](#)
- [tpmvscmgr](#)
- [tracert](#)
- [tracert](#)

- [tree](#)
- [tscon](#)
- [tsdiscon](#)
- [tsecimp](#)
- [tskill](#)
- [tsprof](#)
- [type](#)
- [typeperf](#)
- [tzutil](#)

U

- [unlodctr](#)

V

- [ver](#)
- [verifier](#)
- [verify](#)
- [vol](#)
- [vssadmin-](#)

W

- [waitfor](#)
- [wbadmin](#)
 - [wbadmin enable backup](#)
 - [wbadmin disable backup](#)
 - [wbadmin start backup](#)
 - [wbadmin stop job](#)
 - [wbadmin get versions](#)
 - [wbadmin get items](#)
 - [wbadmin start recovery](#)
 - [wbadmin get status](#)
 - [wbadmin get disks](#)
 - [wbadmin start systemstaterecovery](#)
 - [wbadmin start systemstatebackup](#)
 - [wbadmin delete systemstatebackup](#)
 - [wbadmin start sysrecovery](#)
 - [wbadmin restore catalog](#)
 - [wbadmin delete catalog](#)
- [wdsutil](#)
- [wecutil](#)
- [wevtutil](#)
- [where](#)
- [whoami](#)
- [winnt](#)
- [winnt32](#)
- [winpop](#)
- [winrs](#)
- [wlbs](#)

- [wmic](#)
- [wscript](#)

X

- [xcopy](#)

Command-Line Syntax Key

4/13/2018 • 2 minutes to read • [Edit Online](#)

The following table describes the notation used to indicate command-line syntax.

| NOTATION | DESCRIPTION |
|---------------------------------|--|
| Text without brackets or braces | Items you must type as shown |
| <Text inside angle brackets> | Placeholder for which you must supply a value |
| [Text inside square brackets] | Optional items |
| {Text inside braces} | Set of required items; choose one |
| Vertical bar () | Separator for mutually exclusive items; choose one |
| Ellipsis (...) | Items that can be repeated |

Commands by Server Role

4/13/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

A server role describes the primary function of the server. Administrators can choose to dedicate an entire server to one role, or install multiple server roles and sub roles on a single computer. Each role may include additional command-line tools, installed as part of the role. The following topics provide a list of commands associated with each server role.

- [Print Command Reference](#)
- [Services for Network File System Command Reference](#)
- [Remote Desktop Services \(Terminal Services\) Command Reference](#)
- [Windows Server Backup Command Reference](#)

print Command Reference

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

The following is a list of print command-line tools.

| COMMAND | DESCRIPTION |
|---|--|
| [lpq][pq.md] | Displays the status of a print queue on a computer running Line printer Daemon (LPD). |
| lpr | Sends a file to a computer or printer sharing device running the Line printer Daemon (LPD) service in preparation for printing. |
| Net print | Displays information about a specified printer queue, displays information about a specified print job, or controls a specified print job. |
| print | Sends a text file to a printer. |
| prncnfg | Configures or displays configuration information about a printer. |
| prndrvr | adds, deletes, and lists printer drivers. |
| prnjobs | pauses, resumes, cancels, and lists print jobs. |
| prnmngr | adds, deletes, and lists printers or printer connections, in addition to setting and displaying the default printer. |
| prnport | creates, deletes, and lists standard TCP/IP printer ports, in addition to displaying and changing port configuration. |
| prnqctl | prints a test page, pauses or resumes a printer, and clears a printer queue. |
| pubprn | Publishes a printer to the active directory directory service. |
| rundll32 printui.dll,printUIEntry | Enables you to automate the installation and configuration of printers using scripts or the command prompt. |

Services for Network File System Command Reference

4/13/2018 • 2 minutes to read • [Edit Online](#)

Services for Network File System (NFS) provides a file sharing solution that enables you to transfer files between computers running Windows Server 2008 and UNIX operating systems using the NFS protocol.

The following is a list of NFS command-line tools.

| COMMAND | DESCRIPTION |
|---------------------------|--|
| mapadmin | Manage User Name Mapping for Microsoft Services for Network File System. |
| Mount | Mount Network File System (NFS) network shares. |
| Nfsadmin | Manage Server for NFS and Client for NFS. |
| Nfsshare | Control Network File System (NFS) shares. |
| Nfsstat | Display or reset counts of calls made to Server for NFS. |
| Rpcinfo | List programs on remote computers. |
| Showmount | Display mounted directories. |

Remote Desktop Services (Terminal Services) Command Reference

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

The following is a list of Remote Desktop Services command-line tools.

NOTE

In Windows Server 2008 R2, Terminal Services was renamed Remote Desktop Services. To find out what's new in the latest version, see [What's New in Remote Desktop Services in Windows Server 2012](#) in the Windows Server TechNet Library.

| COMMAND | DESCRIPTION |
|------------------------------|--|
| change | changes Remote Desktop Session Host (rd Session Host) server settings for logons, COM port mappings, and install mode. |
| change logon | Enables or disables logons from client sessions on an rd Session Host server, or displays current logon status. |
| change port | lists or changes the COM port mappings to be compatible with MS-DOS applications. |
| change user | changes the install mode for the rd Session Host server. |
| chglogon | Enables or disables logons from client sessions on an rd Session Host server, or displays current logon status. |
| chgport | lists or changes the COM port mappings to be compatible with MS-DOS applications. |
| chgusr | changes the install mode for the rd Session Host server. |
| flattemp | Enables or disables flat temporary folders. |
| logoff | Logs off a user from a session on an rd Session Host server and deletes the session from the server. |
| msg | Sends a message to a user on an rd Session Host server. |
| mstsc | creates connections to rd Session Host servers or other remote computers. |
| qappsrv | Displays a list of all rd Session Host servers on the network. |
| qprocess | Displays information about processes that are running on an rd Session Host server. |
| query | Displays information about processes, sessions, and rd Session Host servers. |

| COMMAND | DESCRIPTION |
|----------------------------------|---|
| query process | Displays information about processes that are running on an rd Session Host server. |
| query session | Displays information about sessions on an rd Session Host server. |
| query termserver | Displays a list of all rd Session Host servers on the network. |
| query user | Displays information about user sessions on an rd Session Host server. |
| quser | Displays information about user sessions on an rd Session Host server. |
| qwinsta | Displays information about sessions on an rd Session Host server. |
| rdpsign | Enables you to digitally sign a Remote Desktop Protocol (.rdp) file. |
| reset session | Enables you to reset (delete) a session on an rd Session Host server. |
| rwinsta | Enables you to reset (delete) a session on an rd Session Host server. |
| shadow | Enables you to remotely control an active session of another user on an rd Session Host server. |
| tscon | Connects to another session on an rd Session Host server. |
| tsdiscon | Disconnects a session from an rd Session Host server. |
| tskill | Ends a process running in a session on an rd Session Host server. |
| tsprof | Copies the Remote Desktop Services user configuration information from one user to another. |

Windows Server Backup Command Reference

4/13/2018 • 2 minutes to read • [Edit Online](#)

The following subcommands for **wbadmin** provide backup and recovery functionality from a command prompt.

To configure a backup schedule, you must be a member of the **Administrators** group. To perform all other tasks with this command, you must be a member of the **Backup Operators** or the **Administrators** group, or you must have been delegated the appropriate permissions.

You must run **wbadmin** from an elevated command prompt. (To open an elevated command prompt, click **Start**, right-click **Command Prompt**, and then click **Run as administrator**.)

| SUBCOMMAND | DESCRIPTION |
|---|---|
| Wbadmin enable backup | Configures and enables a daily backup schedule. |
| Wbadmin disable backup | Disables your daily backups. |
| Wbadmin start backup | Runs a one-time backup. If used with no parameters, uses the settings from the daily backup schedule. |
| Wbadmin stop job | Stops the currently running backup or recovery operation. |
| Wbadmin get versions | Lists details of backups recoverable from the local computer or, if another location is specified, from another computer. |
| Wbadmin get items | Lists the items included in a specific backup. |
| Wbadmin start recovery | Runs a recovery of the volumes, applications, files, or folders specified. |
| Wbadmin get status | Shows the status of the currently running backup or recovery operation. |
| Wbadmin get disks | Lists disks that are currently online. |
| Wbadmin start systemstaterecovery | Runs a system state recovery. |
| Wbadmin start systemstatebackup | Runs a system state backup. |
| Wbadmin delete systemstatebackup | Deletes one or more system state backups. |
| Wbadmin start sysrecovery | Runs a recovery of the full system (at least all the volumes that contain the operating system's state). This subcommand is only available if you are using the Windows Recovery Environment. |
| Wbadmin restore catalog | Recovers a backup catalog from a specified storage location in the case where the backup catalog on the local computer has been corrupted. |

| SUBCOMMAND | DESCRIPTION |
|--|---|
| Wbadmin delete catalog | Deletes the backup catalog on the local computer. Use this command only if the backup catalog on this computer is corrupted and you have no backups stored at another location that you can use to restore the catalog. |

append

5/7/2018 • 2 minutes to read • [Edit Online](#)

Allows programs to open data files in specified directories as if they were in the current directory. If used without parameters, **append** displays the appended directory list.

NOTE

This command not supported in Windows 10.

For examples of how to use this command, see [Examples](#).

Syntax

```
append [[<Drive>:]<Path>[;...]] [/x[:on|:off]] [/path[:on|:off]] [/e]
append ;
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|---|
| [<Drive>:] | Specifies a drive and directory to append. |
| /x:on | Applies appended directories to file searches and launching applications. |
| /x:off | Applies appended directories only to requests to open files. /x:off is the default setting. |
| /path:on | Applies appended directories to file requests that already specify a path. /path:on is the default setting. |
| /path:off | Turns off the effect of /path:on . |
| /e | Stores a copy of the appended directory list in an environment variable named APPEND. /e may be used only the first time you use append after starting your system. |
| ; | Clears the appended directory list. |
| /? | Displays help at the command prompt. |

Examples

To clear the appended directory list, type:

```
append ;
```

To store a copy of the appended directory to an environment variable named APPEND, type:

```
append /e
```

Additional references

[Command-Line Syntax Key](#)

arp

10/18/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Displays and modifies entries in the Address Resolution Protocol (ARP) cache. The ARP cache contains one or more tables that are used to store IP addresses and their resolved Ethernet or Token Ring physical addresses. There is a separate table for each Ethernet or Token Ring network adapter installed on your computer. Used without parameters, **arp** displays help information.

Syntax

```
arp [/a [<Inetaddr>] [/n <ifaceaddr>]] [/g [<Inetaddr>] [-n <ifaceaddr>]] [/d <Inetaddr> [<ifaceaddr>]] [/s <Inetaddr> <Etheraddr> [<ifaceaddr>]]
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------|---|
| /a [] [/n] | <p>Displays current arp cache tables for all interfaces. The /n parameter is case-sensitive.</p> <p>To display the arp cache entry for a specific IP address, use arp /a with the <i>Inetaddr</i> parameter, where <i>Inetaddr</i> is an IP address. If <i>Inetaddr</i> is not specified, the first applicable interface is used.</p> <p>To display the arp cache table for a specific interface, use the <i>/nifaceaddr</i> parameter in conjunction with the <i>/a</i> parameter where <i>ifaceaddr</i> is the IP address assigned to the interface.</p> |
| /g [] [/n] | Identical to /a . |
| /d [] | <p>Deletes an entry with a specific IP address, where <i>Inetaddr</i> is the IP address.</p> <p>To delete an entry in a table for a specific interface, use the <i>ifaceaddr</i> parameter where <i>ifaceaddr</i> is the IP address assigned to the interface.</p> <p>To delete all entries, use the asterisk (*) wildcard character in place of <i>Inetaddr</i>.</p> |
| /s [] | <p>adds a static entry to the arp cache that resolves the IP address <i>Inetaddr</i> to the physical address <i>Etheraddr</i>.</p> <p>To add a static arp cache entry to the table for a specific interface, use the <i>ifaceaddr</i> parameter where <i>ifaceaddr</i> is an IP address assigned to the interface.</p> |
| /? | Displays help at the command prompt. |

Remarks

- The IP addresses for *Inetaddr* and *ifaceaddr* are expressed in dotted decimal notation.
- The physical address for *Etheraddr* consists of six bytes expressed in hexadecimal notation and separated by hyphens (for example, 00-AA-00-4F-2A-9C).
- Entries added with the **/s** parameter are static and do not time out of the arp cache. The entries are removed if the TCP/IP protocol is stopped and started. To create permanent static arp cache entries, place the appropriate **arp** commands in a batch file and use Scheduled Tasks to run the batch file at startup. ## Examples To display the arp cache tables for all interfaces, type: `arp /a` To display the arp cache table for the interface that is assigned the IP address 10.0.0.99, type: `arp /a /n 10.0.0.99` To add a static arp cache entry that resolves the IP address 10.0.0.80 to the physical address 00-AA-00-4F-2A-9C, type: `arp /s 10.0.0.80 00-AA-00-4F-2A-9C ##` additional references
- [Command-Line Syntax Key](#)

assoc

6/11/2018 • 2 minutes to read • [Edit Online](#)

Displays or modifies file name extension associations. If used without parameters, **assoc** displays a list of all the current file name extension associations.

NOTE

This command is only supported within CMD.EXE and is not available from PowerShell.

For examples of how to use this command, see [Examples](#).

Syntax

```
assoc [<.ext>[=[<FileType>]]]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|--|
| <.ext> | Specifies the file name extension. |
| <FileType> | Specifies the file type to associate with the specified file name extension. |
| /? | Displays help at the command prompt. |

Remarks

- To remove the file type association for a file name extension, add a white space after the equal sign by pressing the SPACEBAR.
- To view current file types that have open command strings defined, use the **ftype** command.
- To redirect the output of **assoc** to a text file, use the > redirection operator.

Examples

To view the current file type association for the file name extension .txt, type:

```
assoc .txt
```

To remove the file type association for the file name extension .bak, type:

```
assoc .bak=
```

NOTE

Be sure to add a space after the equal sign.

To view the output of **assoc** one screen at a time, type:

```
assoc | more
```

To send the output of **assoc** to the file assoc.txt, type:

```
assoc>assoc.txt
```

Additional references

[Command-Line Syntax Key](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Schedules commands and programs to run on a computer at a specified time and date. You can use **at** only when the Schedule service is running. Used without parameters, **at** lists scheduled commands.

Syntax

```
at [\\computername] [[id] [/delete] | /delete [/yes]]
at [\\computername] <time> [/interactive] [/every:date[,...]] | /next:date[,...] <command>
```

Parameters

| PARAMETER | DESCRIPTION |
|------------------|---|
| \\<computerName> | Specifies a remote computer. If you omit this parameter, at schedules the commands and programs on the local computer. |
| <id> | Specifies the identification number assigned to a scheduled command. |
| /delete | Cancels a scheduled command. If you omit <i>ID</i> , all of the scheduled commands on the computer are canceled. |
| /yes | Answers yes to all queries from the system when you delete scheduled events. |
| <time> | Specifies the time when you want to run the command. time is expressed as Hours:Minutes in 24-hour notation (that is, 00:00 [midnight] through 23:59). |
| /interactive | Allows <i>command</i> to interact with the desktop of the user who is logged on at the time <i>Command</i> runs. |
| /every: | Runs <i>command</i> on every specified day or days of the week or month (for example, every Thursday, or the third day of every month). |
| <date> | Specifies the date when you want to run the command. You can specify one or more days of the week (that is, type M,T,W,Th,F,S,Su) or one or more days of the month (that is, type 1 through 31). Separate multiple date entries with commas. If you omit <i>date</i> , at uses the current day of the month. |
| /next: | Runs <i>command</i> on the next occurrence of the day (for example, next Thursday). |

| PARAMETER | DESCRIPTION |
|-----------|--|
| <command> | Specifies the Windows command, program (that is, .exe or .com file), or batch program (that is, .bat or .cmd file) that you want to run. When the command requires a path as an argument, use the absolute path (that is, the entire path beginning with the drive letter). If the command is on a remote computer, specify Universal Naming Convention (UNC) notation for the server and share name, rather than a remote drive letter. |
| /? | Displays help at the command prompt. |

Remarks

- **schtasks** is another command-line scheduling tool that you can use to create and manage scheduled tasks. For more information about **schtasks**, see Related Topics.

- Using **at**

To use **at**, you must be a member of the local Administrators group.

- Loading Cmd.exe

at does not automatically load Cmd.exe, the command interpreter, before running commands. If you are not running an executable (.exe) file, you must explicitly load Cmd.exe at the beginning of the command as follows:

cmd /c dir > c:\test.out

- Viewing scheduled commands

When you use **at** without command-line options, scheduled tasks appear in a table formatted similar to the following:

```
Status ID Day time Command Line OK 1 Each F 4:30 PM net send group leads status due OK 2 Each M 12:00 AM
chkstor > check.file OK 3 Each F 11:59 PM backup2.bat
```

- Including identification number (*ID*)

When you include identification number (*ID*) with **at** at a command prompt, information for a single entry appears in a format similar to the following:

```
Task ID: 1 Status: OK Schedule: Each F time of Day: 4:30 PM Command: net send group leads status due
```

After you schedule a command with **at**, especially a command that has command-line options, check that the command syntax is correct by typing **at** without command-line options. If the information in the Command Line column is incorrect, delete the command and retype it. If it is still incorrect, retype the command with fewer command-line options.

- Viewing results

Commands scheduled with **at** run as background processes. Output is not displayed on the computer screen. To redirect output to a file, use the redirection symbol (>). If you redirect output to a file, you need to use the escape symbol (^) before the redirection symbol, whether you are using **at** at the command line or in a batch file. For example, to redirect output to Output.txt, type:

```
at 14:45 c:\test.bat ^>c:\output.txt
```

The current directory for the executing command is the systemroot folder.

- Changing system time

If you change the system time at a computer after you schedule a command to run with **at**, synchronize the **at** scheduler with the revised system time by typing **at** without command-line options.

- Storing commands

Scheduled commands are stored in the registry. As a result, you do not lose scheduled tasks if you restart the Schedule service.

- Connecting to network drives

Do not use a redirected drive for scheduled jobs that access the network. The Schedule service might not be able to access the redirected drive, or the redirected drive might not be present if a different user is logged on at

the time the scheduled task runs. Instead, use UNC paths for scheduled jobs. For example:

```
at 1:00pm my_backup \\server\share
```

Do not use the following syntax, where **x:** is a connection made by the user:

```
at 1:00pm my_backup x:
```

If you schedule an **at** command that uses a drive letter to connect to a shared directory, include an **at** command to disconnect the drive when you are finished using the drive. If the drive is not disconnected, the assigned drive letter is not available at the command prompt.

- Tasks stopping after 72 hours

By default, tasks scheduled using the **at** command stop after 72 hours. You can modify the registry to change this default value.

1. start registry editor (regedit.exe).

2. Locate and click the following key in the registry:

HKEY_LOCAL_MACHINE\SYSTEM\CurrentControlSet\Services\Schedule

3. On the edit menu, click add Value, and then add the following registry value: Value Name:

atTaskMaxHours Data type: reg_DWORD Radix: Decimal Value Data: 0. A value of 0 in the value data field indicates no limit, does not stop. Values from 1 through 99 indicates the number of hours. **Caution**

- Incorrectly editing the registry may severely damage your system. Before making changes to the registry, you should back up any valued data on the computer.

- Task Scheduler and the **at** command

You can use the Scheduled Tasks folder to view or modify the settings of a task that was created by using the **at** command. When you schedule a task using the **at** command, the task is listed in the Scheduled Tasks folder, with a name such as the following: **at3478**. However, if you modify an at task through the Scheduled Tasks folder, it is upgraded to a normal scheduled task. The task is no longer visible to the **at** command, and the at account setting no longer applies to it. You must explicitly enter a user account and password for the task. ## Examples To display a list of commands scheduled on the Marketing server, type:

```
at \\marketing
```

To learn more about a command with the identification number 3 on the Corp server, type:

```
at \\corp 3
```

To schedule a net share command to run on the Corp server at 8:00 A.M. and redirect the listing to the Maintenance server, in the Reports shared directory, and the Corp.txt file, type:

```
at \\corp 08:00 cmd /c "net share reports=d:\marketing\reports >> \\maintenance\reports\corp.txt"
```

To back up the hard drive of the Marketing server to a tape drive at midnight every five days, create a batch program called Archive.cmd, which contains the backup commands, and then schedule the batch program to run, type:

```
at \\marketing 00:00 /every:5,10,15,20,25,30 archive
```

To cancel all commands scheduled on the current server, clear the **at** schedule information as follows:

```
at /delete
```

To run a command that is not an executable (that is, .exe) file, precede the command with **cmd /c** to load Cmd.exe as follows:

```
cmd /c dir > c:\test.out
```

atmadm

9/28/2018 • 3 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Monitors connections and addresses that are registered by the atM call Manager on an asynchronous transfer mode (atM) network. You can use **atmadm** to display statistics for incoming and outgoing calls on atM adapters. Used without parameters, **atmadm** displays statistics for monitoring the status of active atM connections.

Syntax

```
atmadm [/c][/a][/s]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| /c | Displays call information for all current connections to the atM network adapter installed on this computer. |
| /a | Displays the registered atM network service access point (NSAP) address for each adapter installed in this computer. |
| /s | Displays statistics for monitoring the status of active atM connections. |
| /? | Displays help at the command prompt. |

Remarks

- The **atmadm /c** command produces output similar to the following:

Windows atM call Manager Statistics

atM Connections on Interface : [009] Olicom atM PCI 155 Adapter

| Connection | VPI/VCI | remote address/ Media Parameters (rates in bytes/sec) |
|-------------|---------|--|
| In PMP SVC | 0/193 | 47000580FFE100000F21A2E180020481A2E180B Tx:UBR,Peak 0,Avg 0,MaxSdu 1516 Rx:UBR,Peak 16953936,Avg 16953936,MaxSdu 1516 |
| Out P-P SVC | 0/192 | 47000580FFE100000F21A2E180020481A2E180B Tx:UBR,Peak 16953936,Avg 16953936,MaxSdu 1516 Rx:UBR,Peak 16953936,Avg 16953936,MaxSdu 1516 |
| In PMP SVC | 0/191 | 47000580FFE100000F21A2E180020481A2E180B Tx:UBR,Peak 0,Avg 0,MaxSdu 1516 Rx:UBR,Peak 16953936,Avg 16953936,MaxSdu 1516 |
| Out P-P SVC | 0/190 | 47000580FFE100000F21A2E180020481A2E180B Tx:UBR,Peak 16953936,Avg 16953936,MaxSdu 1516 Rx:UBR,Peak 16953936,Avg 16953936,MaxSdu 1516 |
| In P-P SVC | 0/475 | 47000580FFE100000F21A2E180000C110081501 Tx:UBR,Peak 16953984,Avg 16953984,MaxSdu 9188 Rx:UBR,Peak 16953936,Avg 16953936,MaxSdu 9188 |
| Out PMP SVC | 0/194 | 47000580FFE100000F21A2E180000C110081501 (0) Tx:UBR,Peak 16953984,Avg 16953984,MaxSdu 9180 Rx:UBR,Peak 0,Avg 0,MaxSdu 0 |
| Out P-P SVC | 0/474 | 4700918100000000613E5BFE010000C110081500 Tx:UBR,Peak 16953984,Avg 16953984,MaxSdu 9188 Rx:UBR,Peak 16953984,Avg 16953984,MaxSdu 9188 |
| In PMP SVC | 0/195 | 47000580FFE100000F21A2E180000C110081500 Tx:UBR,Peak 0,Avg 0,MaxSdu 0 Rx:UBR,Peak 16953936,Avg 16953936,MaxSdu 9180 |

The following table contains descriptions of each element in the **atmadm /c** sample output.

| TYPE OF DATA | SCREEN DISPLAY | DESCRIPTION |
|---------------------------------|--|---|
| Connection Information | In/Out | direction of the call. In is to the atM network adapter from another device. Out is from the atM network adapter to another device. |
| | PMP | Point-to-multipoint call. |
| | P-P | Point-to-point call. |
| | SVC | Connection is on a switched virtual circuit. |
| | PVC | Connection is on a permanent virtual circuit. |
| VPI/VCI Information | VPI/VCI | Virtual path and virtual channel of the incoming or outgoing call. |
| remote address/Media Parameters | 47000580FFE100000F21A2E18000C110081500 | NSAP address of the calling (In) or called (Out) atM device. |

| TYPE OF DATA | SCREEN DISPLAY | DESCRIPTION |
|--------------|----------------|---|
| | Tx | <p>The Tx parameter includes the following three elements:</p> <ul style="list-style-type: none"> - Default or specified bit-rate type (UBR, CBR, VBR, or ABR) - Default or specified line speed - Specified service data unit (SDU) size |
| | Rx | <p>The Rx parameter includes the following three elements:</p> <ul style="list-style-type: none"> - Default or specified bit-rate type (UBR, CBR, VBR, or ABR) - Default or specified line speed - Specified SDU size |

- The **atmadm /a** command produces output similar to the following:

```
Windows atM call Manager Statistics
atM addresses for Interface : [009] Olicom atM PCI 155 Adapter
47000580FFE100000F21A2E18000C110081500
```

- The **atmadm /s** command produces output similar to the following:

```
Windows atM call Manager Statistics
atM call Manager statistics for Interface : [009] Olicom atM PCI 155 Adapter
Current active calls                = 4
Total successful Incoming calls     = 1332
Total successful Outgoing calls     = 1297
Unsuccessful Incoming calls        = 1
Unsuccessful Outgoing calls        = 1
calls Closed by remote             = 1302
calls Closed Locally               = 1323
Signaling and ILMI Packets Sent    = 33655
Signaling and ILMI Packets Received = 34989
```

The following table contains descriptions of each element in the **atmadm /s** sample output.

| CALL MANAGER STATISTIC | DESCRIPTION |
|---------------------------------|---|
| Current active calls | calls currently active on the atM adapter installed on this computer. |
| Total successful Incoming calls | calls successfully received from other devices on this atM network. |
| Total successful Outgoing calls | calls successfully completed to other atM devices on this network from this computer. |
| Unsuccessful Incoming calls | Incoming calls that failed to connect to this computer. |
| Unsuccessful Outgoing calls | Outgoing calls that failed to connect to another device on the network. |

| CALL MANAGER STATISTIC | DESCRIPTION |
|-------------------------------------|--|
| calls Closed by remote | calls closed by a remote device on the network. |
| calls Closed Locally | calls closed by this computer. |
| Signaling and ILMI Packets Sent | Number of integrated local management interface (ILMI) packets sent to the switch to which this computer is attempting to connect. |
| Signaling and ILMI Packets Received | Number of ILMI packets received from the atM switch. |

Examples

To display call information for all current connections to the atM network adapter installed on this computer, type:

```
atmadm /c
```

To display the registered atM network service access point (NSAP) address for each adapter installed in this computer, type:

```
atmadm /a
```

To display statistics for monitoring the status of active atM connections, type:

```
atmadm /s
```

Additional references

- [Command-Line Syntax Key](#)

attrib

6/11/2018 • 2 minutes to read • [Edit Online](#)

Displays, sets, or removes attributes assigned to files or directories. If used without parameters, **attrib** displays attributes of all files in the current directory.

For examples of how to use this command, see [Examples](#).

Syntax

```
attrib [{+|-}r] [{+|-}a] [{+|-}s] [{+|-}h] [{+|-}i] [<Drive>:][<Path>][<FileName>] [/s [/d] [/l]]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------|--|
| {+ -}r | Sets (+) or clears (-) the Read-only file attribute. |
| {+ -}a | Sets (+) or clears (-) the Archive file attribute. |
| {+ -}s | Sets (+) or clears (-) the System file attribute. |
| {+ -}h | Sets (+) or clears (-) the Hidden file attribute. |
| {+ -}i | Sets (+) or clears (-) the Not Content Indexed file attribute. |
| <Drive>:[] [] | Specifies the location and name of the directory, file, or group of files for which you want to display or change attributes. You can use the ? and * wildcard characters in the <i>FileName</i> parameter to display or change the attributes for a group of files. |
| /s | Applies attrib and any command-line options to matching files in the current directory and all of its subdirectories. |
| /d | Applies attrib and any command-line options to directories. |
| /l | Applies attrib and any command-line options to the Symbolic Link, rather than the target of the Symbolic Link. |
| /? | Displays help at the command prompt. |

Remarks

- You can use wildcard characters (?) and (*) with the *FileName* parameter to display or change the attributes for a group of files.
- If a file has the System (**s**) or Hidden (**h**) attribute set, you must clear the attribute before you can change any other attributes for that file.
- The Archive attribute (**a**) marks files that have changed since the last time they were backed up. Note that the

xcopy command uses archive attributes.

Examples

To display the attributes of a file named News86 that is located in the current directory, type:

```
attrib news86
```

To assign the Read-only attribute to the file named Report.txt, type:

```
attrib +r report.txt
```

To remove the Read-only attribute from files in the Public directory and its subdirectories on a disk in drive B, type:

```
attrib -r b:\public\*.* /s
```

To set the Archive attribute for all files on drive A, and then clear the Archive attribute for files with the .bak extension, type:

```
attrib +a a: *.* & attrib -a a:*.bak
```

auditpol

4/13/2018 • 2 minutes to read • [Edit Online](#)

Displays information about and performs functions to manipulate audit policies.

For examples of how this command can be used, see the Examples section in each topic.

Syntax

```
Auditpol command [<sub-command><options>]
```

Parameters

| SUB-COMMAND | DESCRIPTION |
|---------------|---|
| /get | Displays the current audit policy. See Auditpol get for syntax and options. |
| /set | Sets the audit policy. See Auditpol set for syntax and options. |
| /list | Displays selectable policy elements. See Auditpol list for syntax and options. |
| /backup | Saves the audit policy to a file. See Auditpol backup for syntax and options. |
| /restore | Restores the audit policy from a file that was previously created by using auditpol /backup. See Auditpol restore for syntax and options. |
| /clear | Clears the audit policy. See Auditpol clear for syntax and options. |
| /remove | Removes all per-user audit policy settings and disables all system audit policy settings. See Auditpol remove for syntax and options. |
| /resourceSACL | Configures global resource system access control lists (SACLs). Note: Applies only to Windows 7 and Windows Server 2008 R2. See Auditpol resourceSACL . |
| /? | Displays help at the command prompt. |

Remarks

The audit policy command-line tool can be used to:

- Set and query a system audit policy.

- Set and query a per-user audit policy.
- Set and query auditing options.
- Set and query the security descriptor used to delegate access to an audit policy.
- Report or back up an audit policy to a comma-separated value (CSV) text file.
- Load an audit policy from a CSV text file.
- Configure global resource SACLs.

Additional references

[Command-Line Syntax Key](#)

autochk

4/13/2018 • 2 minutes to read • [Edit Online](#)

Runs when the computer is started and prior to Windows Server® 2008 R2 starting to verify the logical integrity of a file system.

Autochk.exe is a version of **Chkdsk** that runs only on NTFS disks and only before Windows Server 2008 R2 starts. **Autochk** cannot be run directly from the command-line. Instead, **Autochk** runs in the following situations:

- If you try to run **Chkdsk** on the boot volume
- If **Chkdsk** cannot gain exclusive use of the volume
- If the volume is flagged as dirty

Remarks

- [!WARNING] The **Autochk** command-line tool cannot be directly run from the command-line. Instead, use the **Chkntfs** command-line tool to configure the way you want **Autochk** to run at startup.
- You can use **Chkntfs** with the **/x** parameter to prevent **Autochk** from running on a specific volume or multiple volumes.
- Use the **Chkntfs.exe** command-line tool with the **/t** parameter to change the Autochk delay from 0 seconds to up to 3 days (259,200 seconds). However, a long delay means that the computer does not start until the time elapses or until you press a key to cancel **Autochk**.

Additional references

[Command-Line Syntax Key](#)

[Chkdsk](#)

[Chkntfs](#)

[Troubleshooting Disks and File Systems](#)

autoconv

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

converts file allocation table (Fat) and Fat32 volumes to the NTFS file system, leaving existing files and directories intact at startup after **autochk** runs. volumes converted to the NTFS file system cannot be converted back to Fat or Fat32.

Remarks

You cannot run **autoconv** on the command-line. This will only be run at startup, if set through **convert.exe**.

additional references

[Command-Line Syntax Key autochk convert Working with File Systems](#)

autofmt

4/13/2018 • 2 minutes to read • [Edit Online](#)

Formats a drive or partition when called from the Windows Recovery Console.

Remarks

You cannot run **Autofmt** directly from the command-line.

Additional references

[Command-Line Syntax Key](#)

[Working with File Systems](#)

bcdboot

4/13/2018 • 2 minutes to read • [Edit Online](#)

Enables you to quickly set up a system partition, or to repair the boot environment located on the system partition. The system partition is set up by copying a simple set of Boot Configuration Data (BCD) files to an existing empty partition.

For more information about BCDboot, including information on where to find BCDboot and examples of how to use this command, see the [BCDboot Command-Line Options](#) topic.

Syntax

```
bcdboot <source> [/l] [/s]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| source | Specifies the location of the Windows directory to use as the source for copying boot environment files. |
| /l | Specifies the locale. The default locale is US English. |
| /s | Specifies the volume letter of the system partition. The default is the system partition identified by the firmware. |

Examples

For more examples of how to use this command, see the [BCDboot Command-Line Options](#) topic.

Additional references

[Command-Line Syntax Key](#)

bcdedit

4/13/2018 • 4 minutes to read • [Edit Online](#)

Boot Configuration Data (BCD) files provide a store that is used to describe boot applications and boot application settings. The objects and elements in the store effectively replace Boot.ini.

BCDEdit is a command-line tool for managing BCD stores. It can be used for a variety of purposes, including creating new stores, modifying existing stores, adding boot menu parameters, and so on. BCDEdit serves essentially the same purpose as Bootcfg.exe on earlier versions of Windows, but with two major improvements:

- Exposes a wider range of boot parameters than Bootcfg.exe.
- Has improved scripting support.

NOTE

Administrative privileges are required to use BCDEdit to modify BCD.

BCDEdit is the primary tool for editing the boot configuration of Windows Vista and later versions of Windows. It is included with the Windows Vista distribution in the %WINDIR%\System32 folder.

BCDEdit is limited to the standard data types and is designed primarily to perform single common changes to BCD. For more complex operations or nonstandard data types, consider using the BCD Windows Management Instrumentation (WMI) application programming interface (API) to create more powerful and flexible custom tools.

Syntax

```
BCDEdit /Command [<Argument1>] [<Argument2>] ...
```

Parameters

General BCDEdit Command-Line Option

| OPTION | DESCRIPTION |
|--------|--|
| /? | Displays a list of BCDEdit commands. Running this command without an argument displays a summary of the available commands. To display detailed help for a particular command, run bcdedit /? <command> , where is the name of the command you are searching for more information about. For example, bcdedit /? createstore displays detailed help for the Createstore command. |

Parameters that Operate on a Store

| OPTION | DESCRIPTION |
|--------------|---|
| /createstore | Creates a new empty boot configuration data store. The created store is not a system store. |

| OPTION | DESCRIPTION |
|---------|---|
| /export | Exports the contents of the system store into a file. This file can be used later to restore the state of the system store. This command is valid only for the system store. |
| /import | Restores the state of the system store by using a backup data file previously generated by using the /export option. This command deletes any existing entries in the system store before the import takes place. This command is valid only for the system store. |
| /store | This option can be used with most BCDedit commands to specify the store to be used. If this option is not specified, then BCDEdit operates on the system store. Running the bcdedit /store command by itself is equivalent to running the bcdedit /enum active command. |

Parameters that Operate on Entries in a Store

| PARAMETER | DESCRIPTION |
|-----------|--|
| /copy | Makes a copy of a specified boot entry in the same system store. |
| /create | Creates a new entry in the boot configuration data store. If a well-known identifier is specified, then the /application , /inherit , and /device parameters cannot be specified. If an identifier is not specified or not well known, an /application , /inherit , or /device option must be specified. |
| /delete | Deletes an element from a specified entry. |

Parameters that Operate on Entry Options

| PARAMETER | DESCRIPTION |
|--------------|--|
| /deletevalue | Deletes a specified element from a boot entry. |
| /set | Sets an entry option value. |

Parameters that Control Output

| PARAMETER | DESCRIPTION |
|-----------|--|
| /enum | Lists entries in a store. The /enum option is the default value for BCDedit, so running the bcdedit command without parameters is equivalent to running the bcdedit /enum active command. |
| /v | Verbose mode. Usually, any well-known entry identifiers are represented by their friendly shorthand form. Specifying /v as a command-line option displays all identifiers in full. Running the bcdedit /v command by itself is equivalent to running the bcdedit /enum active /v command. |

Parameters that Control the Boot Manager

| PARAMETER | DESCRIPTION |
|--------------------|---|
| /bootsequence | Specifies a one-time display order to be used for the next boot. This command is similar to the /displayorder option, except that it is used only the next time the computer starts. Afterwards, the computer reverts to the original display order. |
| /default | Specifies the default entry that the boot manager selects when the timeout expires. |
| /displayorder | Specifies the display order that the boot manager uses when displaying boot parameters to a user. |
| /timeout | Specifies the time to wait, in seconds, before the boot manager selects the default entry. |
| /toolsdisplayorder | Specifies the display order for the boot manager to use when displaying the Tools menu. |

Parameters that Control Emergency Management Services

| PARAMETER | DESCRIPTION |
|--------------|--|
| /bootems | Enables or disables Emergency Management Services (EMS) for the specified entry. |
| /ems | Enables or disables EMS for the specified operating system boot entry. |
| /emssettings | Sets the global EMS settings for the computer. /emssettings does not enable or disable EMS for any particular boot entry. |

Parameters that Control Debugging

| PARAMETER | DESCRIPTION |
|--------------|---|
| /bootdebug | Enables or disables the boot debugger for a specified boot entry. Although this command works for any boot entry, it is effective only for boot applications. |
| /dbgsettings | Specifies or displays the global debugger settings for the system. This command does not enable or disable the kernel debugger; use the /debug option for that purpose. To set an individual global debugger setting, use the bcdedit /set <dbgsettings> command. |
| /debug | Enables or disables the kernel debugger for a specified boot entry. |

Examples

For examples of BCDEdit, see the [BCDEdit Options Reference](#).

bdehdcfg

5/14/2018 • 2 minutes to read • [Edit Online](#)

Prepares a hard drive with the partitions necessary for BitLocker Drive Encryption. Most installations of Windows 7 will not need to use this tool because BitLocker setup includes the ability to prepare and repartition drives as required.

WARNING

There is a known conflict with the **Deny write access to fixed drives not protected by BitLocker** Group Policy setting located in **Computer Configuration\Administrative Templates\Windows Components\BitLocker Drive Encryption\Fixed Data Drives**.

- > If Bdehdcfg is run on a computer when this policy setting is enabled, you may encounter the following issues:
- > - If you attempted to shrink the drive and create the system drive, the drive size will be successfully reduced and a raw partition will be created. However, the raw partition will not be formatted. The following error message is displayed: "The new active Drive cannot be formatted. You may need to manually prepare your drive for BitLocker."
- > - If you attempted to use unallocated space to create the system drive, a raw partition will be created. However, the raw partition will not be formatted. The following error message is displayed: "The new active Drive cannot be formatted. You may need to manually prepare your drive for BitLocker."
- > - If you attempted to merge an existing drive into the system drive, the tool will fail to copy the required boot file onto the target drive to create the system drive. The following error message is displayed: "BitLocker setup failed to copy boot files. You may need to manually prepare your drive for BitLocker."
- > If this policy setting is being enforced, a hard drive cannot be repartitioned because the drive is protected. If you are upgrading computers in your organization from a previous version of Windows and those computers were configured with a single partition, you should create the required BitLocker system partition before applying the policy setting to the computers.

For examples of how this command can be used, see [Examples](#).

Syntax

```
bdehdcfg [-driveinfo <DriveLetter>] [-target {default|unallocated|<DriveLetter> shrink|<DriveLetter> merge}] [-newdriveletter] [-size <SizeinMB>] [-quiet]
```

Parameters

| PARAMETER | DESCRIPTION |
|--|--|
| Bdehdcfg: driveinfo | Displays the drive letter, the total size, the maximum free space, and the partition characteristics of the partitions on the drive specified. Only valid partitions are listed. Unallocated space is not listed if four primary or extended partitions already exist. |
| Bdehdcfg: target | Defines which portion of a drive to use as the system drive and makes the portion active. |
| Bdehdcfg: newdriveletter | Assigns a new drive letter to the portion of a drive used as the system drive. |

| PARAMETER | DESCRIPTION |
|-----------------------------------|---|
| Bdehdcfg: size | Determines the size of the system partition when a new system drive is being created. |
| Bdehdcfg: quiet | Prevents the display of all actions and errors in the command-line interface and directs Bdehdcfg to use the "Yes" answer to any Yes/No prompts that may occur during subsequent drive preparation. |
| Bdehdcfg: restart | Directs the computer to restart after the drive preparation has finished. |
| /? | Displays Help at the command prompt. |

Examples

The following example depicts Bdehdcfg being used with the default drive to create a system partition of 500 MB. Because no drive letter is specified, the new system partition will not have a drive letter.

```
bdehdcfg -target default -size 500
```

The following example depicts Bdehdcfg being used with the default drive to create a system partition (P:) of the default size of 300 MB out of unallocated space on the drive. The tool will not prompt the user for any further input nor will any errors be displayed. After the system drive has been created, the computer will automatically restart.

```
bdehdcfg -target unallocated -newdriveletter P: -quiet -restart
```

Additional references

- [Command-Line Syntax Key](#)

bitsadmin

5/7/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012, Windows 10

bitsadmin is a command-line tool that you can use to create download or upload jobs and monitor their progress.

Commands

bitsadmin addfile
bitsadmin addfileset
bitsadmin addfilewithranges
bitsadmin cancel
bitsadmin complete
bitsadmin create
bitsadmin getaclflags
bitsadmin getbytestotal
bitsadmin getbytestransferred
bitsadmin getcompletiontime
bitsadmin getcreationtime
bitsadmin getdescription
bitsadmin getdisplayname
bitsadmin geterror
bitsadmin geterrorcount
bitsadmin getfiletotal
bitsadmin getfilestransferred
bitsadmin getminretrydelay
bitsadmin getmodificationtime
bitsadmin getnoprogresstimeout
bitsadmin getnotifycmdline
bitsadmin getnotifyflags
bitsadmin getnotifyinterface
bitsadmin getowner
bitsadmin get priority
bitsadmin getproxybypasslist
bitsadmin getproxylist
bitsadmin getproxyusage
bitsadmin getreplydata
bitsadmin getreplyfilename
bitsadmin getreplyprogress
bitsadmin getstate
bitsadmin gettype
bitsadmin help
bitsadmin info
bitsadmin list
bitsadmin listfiles
bitsadmin monitor

bitsadmin nowrap
bitsadmin rawreturn
bitsadmin removecredentials
bitsadmin replaceremoteprefix
bitsadmin reset
bitsadmin resume
bitsadmin setaclflag
bitsadmin setcredentials
bitsadmin setdescription
bitsadmin setdisplayname
bitsadmin setminretrydelay
bitsadmin setnoprogresstimeout
bitsadmin setnotifycmdline
bitsadmin setnotifyflags
bitsadmin setpriority
bitsadmin setproxysettings
bitsadmin setreplyfilename
bitsadmin suspend
bitsadmin takeownership
bitsadmin Transfer
bitsadmin util
bitsadmin wrap

bitsadmin addfile

4/13/2018 • 2 minutes to read • [Edit Online](#)

Adds a file to the specified job.

Syntax

```
bitsadmin /AddFile <Job> <RemoteURL> <LocalName>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| Job | The job's display name or GUID |
| RemoteURL | The URL of the file on the server. |
| LocalName | The name of the file on the local computer. <i>LocalName</i> must contain an absolute path to the file. |

Examples

Add a file to the job. Repeat this call for each file you want to add. If multiple jobs use *myDownloadJob* as their name, you must replace *myDownloadJob* with the job's GUID to uniquely identify the job.

```
C:\>bitsadmin /addfile myDownloadJob http://downloadsrv/10mb.zip c:\10mb.zip
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin addfileset

4/13/2018 • 2 minutes to read • [Edit Online](#)

Adds one or more files to the specified job.

Syntax

```
bitsadmin /addfileset <Job> <TextFile>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| Job | The job's display name or GUID |
| TextFile | A text file containing remote and local file names. Note: The names are space-delimited. Lines that begin with a # character are treated as a comment. |

Examples

```
C:\>bitsadmin /addfileset files.txt
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin addfilewithranges

4/13/2018 • 2 minutes to read • [Edit Online](#)

Adds a file to the specified job. BITS downloads the specified ranges from the remote file.

Syntax

```
bitsadmin /AddFileWithRanges <Job> <RemoteURL> <LocalName> <RangeList>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| Job | The job's display name or GUID |
| RemoteURL | <i>RemoteURL</i> is the URL of the file on the server. |
| LocalName | <i>LocalName</i> is the name of the file on the local computer. <i>LocalName</i> must contain an absolute path to the file. |
| RangeList | <i>RangeList</i> is a comma-delimited list of offset and length pairs. Use a colon to separate the offset value from the length value. |

More Information

- The token **eof** is a valid length value within the offset and length pairs in the *<RangeList>*. It instructs the service to read to the end of the specified file.
- Note that AddFileWithRanges will fail with error code 0x8020002c when a zero-length range is specified along with another range with same offset, such as: C:\bits>bitsadmin /addfilewithranges j2 <http://bitsdc/dload/1k.zip> c:\1k.zip 100:0,100:5

Error message: Unable to add file to job - 0x8020002c. The list of byte ranges contains some overlapping ranges, which are not supported.

Workaround: do not specify the zero-length range first. For example: bitsadmin /addfilewithranges j2 <http://bitsdc/dload/1k.zip> c:\1k.zip 100:5,100:0.

Examples

The following example tells BITS to transfer 100 bytes from offset 0, 100 bytes from offset 2000, and the remaining bytes from offset 5000 to the end of the file.

```
C:\>bitsadmin /addfilewithranges http://downloadsrv/10mb.zip c:\10mb.zip "0:100,2000:100,5000:eof"
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin cancel

4/13/2018 • 2 minutes to read • [Edit Online](#)

Removes the job from the transfer queue and deletes all temporary files associated with the job.

Syntax

```
bitsadmin /cancel <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Examples

The following example removes the *myDownloadJob* job from the transfer queue.

```
C:\>bitsadmin /cancel myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin complete

4/13/2018 • 2 minutes to read • [Edit Online](#)

Completes the job. The downloaded files are not available to you until you use this switch. Use this switch after the job moves to the transferred state. Otherwise, only those files that have been successfully transferred are available.

Syntax

```
bitsadmin /complete <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Examples

When the state of the job is TRANSFERRED, BITS has successfully transferred all files in the job. However, the files are not available until you use the **/complete** switch. If multiple jobs use *myDownloadJob* as their name, you must replace *myDownloadJob* with the job's GUID to uniquely identify the job.

```
C:\>bitsadmin /complete myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin create

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

creates a transfer job with the given display name.

Syntax

```
bitsadmin /create [type] DisplayName
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------|--|
| type | <ul style="list-style-type: none">- /Download transfers data from a server to a local file.- /Upload transfers data from a local file to a server.- /Upload-Reply transfers data from a local file to a server and receives a reply file from the server.- This parameter defaults to /Download when not specified on the command line. |
| DisplayName | The display name assigned to the newly created job. |

Use the [bitsadmin resume](#) switch to activate the job in the transfer queue.

Examples

creates a download job named *myDownloadJob*.

```
C:\>bitsadmin /create myDownloadJob
```

additional references

[Command-Line Syntax Key](#)

bitsadmin getaclflags

4/13/2018 • 2 minutes to read • [Edit Online](#)

Retrieves the access control list propagations flags.

Syntax

```
bitsadmin /GetAcLFlags <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Remarks

Displays one or more of the following flag values:

- O: Copy owner information with file.
- G: Copy group information with file.
- D: Copy DACL information with file.
- S: Copy SACL information with file.

Examples

The following example retrieves the access control list propagation flags for the job named *myDownloadJob*.

```
C:\>bitsadmin /getaclflags myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin getbytestotal

4/13/2018 • 2 minutes to read • [Edit Online](#)

Retrieves the size of the specified job

Syntax

```
bitsadmin /GetBytesTotal <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Examples

The following example retrieves the size of the job named *myDownloadJob*.

```
C:\>bitsadmin /GetBytesTotal myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin getbytestransferred

4/13/2018 • 2 minutes to read • [Edit Online](#)

Retrieves the number of bytes transferred for the specified job.

Syntax

```
bitsadmin /GetBytesTransferred <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Examples

The following example retrieves the number of bytes transferred for the job named *myDownloadJob*.

```
C:\>bitsadmin /GetBytesTransferred myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin getcompletiontime

4/13/2018 • 2 minutes to read • [Edit Online](#)

Retrieves the time that the job finished transferring data.

Syntax

```
bitsadmin /GetCompletionTime <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Examples

The following example retrieves the time that the job named *myDownloadJob* finished transferring data.

```
C:\>bitsadmin /GetCompletionTime myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin getcreationtime

4/13/2018 • 2 minutes to read • [Edit Online](#)

Retrieves the creation time for the specified job.

Syntax

```
bitsadmin /GetCreationTime <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Examples

The following example retrieves the creation time for the job named *myDownloadJob*.

```
C:\>bitsadmin /GetCreationTime myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin getdescription

4/13/2018 • 2 minutes to read • [Edit Online](#)

Retrieves the description of the specified job.

Syntax

```
bitsadmin /GetDescription <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Examples

The following example retrieves the description for the job named *myDownloadJob*.

```
C:\>bitsadmin /GetDescription myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin getdisplayname

4/13/2018 • 2 minutes to read • [Edit Online](#)

Retrieves the display name of the specified job.

Syntax

```
bitsadmin /GetDisplayName <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Examples

The following example retrieves the display name for the job named *myDownloadJob*.

```
C:\>bitsadmin /GetDisplayName myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin geterror

4/13/2018 • 2 minutes to read • [Edit Online](#)

Retrieves detailed error information for the specified job.

Syntax

```
bitsadmin /GetError <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Examples

The following example retrieves the error information for the job named *myDownloadJob*.

```
C:\>bitsadmin /GetError myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin geterrorcount

4/13/2018 • 2 minutes to read • [Edit Online](#)

Retrieves a count of the number of times the specified job generated a transient error.

Syntax

```
bitsadmin /GetErrorCount <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Examples

The following example retrieves error count information for the job named *myDownloadJob*.

```
C:\>bitsadmin /GetErrorCount myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin getfilestotal

4/13/2018 • 2 minutes to read • [Edit Online](#)

Retrieves the number of files in the specified job.

Syntax

```
bitsadmin /GetFilesTotal <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Examples

The following example retrieves the number of files included in the job named *myDownloadJob*.

```
C:\>bitsadmin /GetFilesTotal myDownloadJob
```

#

[Command-Line Syntax Key](#) See Also

bitsadmin getfilestransferred

4/13/2018 • 2 minutes to read • [Edit Online](#)

Retrieves the number of files transferred for the specified job.

Syntax

```
bitsadmin /GetFilesTransferred <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Examples

The following example retrieves the number of files transferred in the job named *myDownloadJob*.

```
C:\>bitsadmin /GetFilesTransferred myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin getminretrydelay

4/13/2018 • 2 minutes to read • [Edit Online](#)

Retrieves the length of time, in seconds, that the service waits after encountering a transient error before trying to transfer the file.

Syntax

```
bitsadmin /GetMinRetryDelay <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Examples

The following example retrieves the minimum retry delay for the job named *myDownloadJob*.

```
C:\>bitsadmin /GetMinRetryDelay myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin getmodificationtime

4/13/2018 • 2 minutes to read • [Edit Online](#)

Retrieves the last time the job was modified or data was successfully transferred.

Syntax

```
bitsadmin /GetModificationTime <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Examples

The following example retrieves the last modified time for the job named *myDownloadJob*.

```
C:\>bitsadmin /GetModificationTime myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin getnoprogresstimeout

4/13/2018 • 2 minutes to read • [Edit Online](#)

Retrieves the length of time, in seconds, that the service tries to transfer the file after a transient error occurs.

Syntax

```
bitsadmin /GetNoProgressTimeout <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Examples

The following example retrieves the progress time out value for the job named *myDownloadJob*.

```
C:\>bitsadmin /GetNoProgressTimeout myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin getnotifycmdline

4/13/2018 • 2 minutes to read • [Edit Online](#)

Retrieves the command-line command that is ran when the job finishes transferring data.

Syntax

```
bitsadmin /GetNotifyCmdLine <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Examples

The following example retrieves the command-line command used by the service when the job named *myDownloadJob* completes.

```
C:\>bitsadmin /GetNotifyCmdLine myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin getnotifyflags

4/13/2018 • 2 minutes to read • [Edit Online](#)

Retrieves the notify flags for the specified job.

Syntax

```
bitsadmin /GetNotifyFlags <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Remarks

The job can contain one or more of the following notification flags.

|-----|-----| |0x001|Generate an event when all files in the job have been transferred.| |0x002|Generate an event when an error occurs.| |0x004|Disable notifications.| |0x008|Generate an event when the job is modified or transfer progress is made.|

Examples

The following example retrieves the notify flags for the job named *myDownloadJob*.

```
C:\>bitsadmin /GetNotifyFlags myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin getnotifyinterface

4/13/2018 • 2 minutes to read • [Edit Online](#)

Determines if another program has registered a COM callback interface for the specified job.

Syntax

```
bitsadmin /GetNotifyInterface <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Remarks

Displays REGISTERED or UNREGISTERED.

NOTE

It is not possible to determine the program that registered the callback interface.

Examples

The following example retrieves the notify interface for the job named *myDownloadJob*.

```
C:\>bitsadmin /GetNotifyInterface myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin getowner

4/13/2018 • 2 minutes to read • [Edit Online](#)

Retrieves the owner of the specified job.

Syntax

```
bitsadmin /GetOwner <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Examples

The following example displays the owner for the job named *myDownloadJob*.

```
C:\>bitsadmin /GetOwner myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin get priority

4/13/2018 • 2 minutes to read • [Edit Online](#)

Retrieves the priority of the specified job.

Syntax

```
bitsadmin /GetPriority <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Remarks

- The priority is either FOREGROUND, HIGH, NORMAL, LOW, or UNKNOWN.

Examples

The following example retrieves the priority for the job named *myDownloadJob*.

```
C:\>bitsadmin /GetPriority myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin getproxybypasslist

4/13/2018 • 2 minutes to read • [Edit Online](#)

Retrieves the proxy bypass list for the specified job.

Syntax

```
bitsadmin /GetProxyBypassList <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Remarks

- The bypass list contains the host names or IP addresses, or both, that are not to be routed through a proxy. The list can contain "<local>" to refer to all servers on the same LAN. The list can be semicolon or space-delimited.

Examples

The following example retrieves the proxy bypass list for the job named *myDownloadJob*.

```
C:\>bitsadmin /GetProxyBypassList myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin getproxylist

4/13/2018 • 2 minutes to read • [Edit Online](#)

Retrieves the proxy list for the specified job.

Syntax

```
bitsadmin /GetProxyList <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Remarks

- The proxy list is the list of proxy servers to use. The list is comma-delimited.

Examples

The following example retrieves the proxy list for the job named *myDownloadJob*.

```
C:\>bitsadmin /GetProxyList myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin getproxyusage

4/13/2018 • 2 minutes to read • [Edit Online](#)

Retrieves the proxy usage setting for the specified job.

Syntax

```
bitsadmin /GetProxyUsage <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Remarks

The possible values are:

- PRECONFIG—use the owner's Internet Explorer defaults.
- NO_PROXY—do not use a proxy server.
- OVERRIDE—Use an explicit proxy list.
- AUTODETECT—Automatically detect the proxy settings.

Examples

The following example retrieves the proxy usage for the job named *myDownloadJob*.

```
C:\>bitsadmin /GetProxyUsage myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin getreplydata

4/13/2018 • 2 minutes to read • [Edit Online](#)

Retrieves the server's reply data in hexadecimal format.

Syntax

```
bitsadmin /GetReplyData <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Remarks

Valid only for upload-reply jobs.

Examples

The following example retrieves the reply data for the job named *myDownloadJob*.

```
C:\>bitsadmin /GetReplyData myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin getreplyfilename

4/13/2018 • 2 minutes to read • [Edit Online](#)

Gets the path of the file that contains the server reply.

Syntax

```
bitsadmin /GetReplyFileName <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Remarks

Valid only for upload-reply jobs.

Examples

The following example retrieves the reply filename for the job named *myDownloadJob*.

```
C:\>bitsadmin /GetReplyFileName myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin getreplyprogress

4/13/2018 • 2 minutes to read • [Edit Online](#)

Retrieves the size and progress of the server reply.

Syntax

```
bitsadmin /GetReplyProgress <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Remarks

Valid only for upload-reply jobs.

Examples

The following example retrieves the reply progress for the job named *myDownloadJob*.

```
C:\>bitsadmin /GetReplyProgress myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin getstate

4/13/2018 • 2 minutes to read • [Edit Online](#)

Retrieves the state of the specified job.

Syntax

```
bitsadmin /GetState <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Remarks

The possible states are:

|-----|-----| |QUEUED|The job is waiting to run.| |CONNECTING|BITS is contacting the server.|
|TRANSFERRING|BITS is transferring data.| |SUSPENDED|The job is paused.| |ERROR|A non-recoverable error
occurred; the transfer will not be retried.| |TRANSIENT_ERROR|A recoverable error occurred; the transfer retries
when the minimum retry delay expires.| |ACKNOWLEDGED|The job was completed.| |CANCELED|The job was
canceled.|

Examples

The following example retrieves the state for the job named *myDownloadJob*.

```
C:\>bitsadmin /GetState myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin gettype

4/13/2018 • 2 minutes to read • [Edit Online](#)

Retrieves the job type of the specified job.

Syntax

```
bitsadmin /GetType <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Remarks

The type can be DOWNLOAD, UPLOAD, UPLOAD-REPLY, or UNKNOWN.

Examples

The following example retrieves the job type for the job named *myDownloadJob*.

```
C:\>bitsadmin /GetType myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin help

4/13/2018 • 2 minutes to read • [Edit Online](#)

Displays the command-line usage.

Syntax

```
bitsadmin /help | /?
```

Examples

The following example retrieves the command-line help.

```
C:\>bitsadmin /help
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin info

4/13/2018 • 2 minutes to read • [Edit Online](#)

Displays summary information about the specified job.

Syntax

```
bitsadmin /Info <Job> [/verbose]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Remarks

Use the /verbose parameter to provide detailed information about the job.

Examples

The following example retrieves information about the job named *myDownloadJob*.

```
C:\>bitsadmin /Info myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin list

4/13/2018 • 2 minutes to read • [Edit Online](#)

Lists the transfer jobs owned by the current user.

Syntax

```
bitsadmin /List [/allusers][/verbose]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| /Allusers | Optional—lists jobs for all users |
| /Verbose | Optional—provides detail information for each job. |

Remarks

You must have administrator privileges to use the /allusers parameter

Examples

The following example retrieves information about jobs owned by the current user.

```
C:\>bitsadmin /List
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin listfiles

4/13/2018 • 2 minutes to read • [Edit Online](#)

Lists the files in the specified job.

Syntax

```
bitsadmin /ListFiles <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Examples

The following example retrieves the list of files for the job named *myDownloadJob*.

```
C:\>bitsadmin /GetNotifyFlags myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin monitor

4/13/2018 • 2 minutes to read • [Edit Online](#)

Monitors jobs in the transfer queue that the current user owns.

Syntax

```
bitsadmin /Monitor [/allusers] [/refresh <Seconds>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| Allusers | Optional—monitors jobs for all users |
| Refresh | Optional—refreshes the data at an interval specified by <i>Seconds</i> . The default refresh interval is five seconds. |

Remarks

- You must have administrator privileges to use the **Allusers** parameter.
- Use CTRL+C to stop the refresh.

Examples

The following example monitors the transfer queue for jobs owned by the current user and refreshes the information every 60 seconds.

```
C:\>bitsadmin /Monitor /refresh 60
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin nowrap

4/13/2018 • 2 minutes to read • [Edit Online](#)

Truncates any line of output text extending beyond the rightmost edge of the command window.

Syntax

```
bitsadmin /NoWrap
```

Remarks

By default, all commands, except the **Monitor** command, wrap the output. Specify the **NoWrap** command before other commands.

Examples

The following example retrieves the state for the job named *myDownloadJob* and does not wrap the output

```
C:\>bitsadmin /NoWrap /GetState myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin rawreturn

4/13/2018 • 2 minutes to read • [Edit Online](#)

Returns data suitable for parsing.

Syntax

```
bitsadmin /RawReturn
```

Remarks

Strips new line characters and formatting from the output.

Typically, you use this command in conjunction with the **Create** and **Get*** commands to receive only the value. You must specify this command before other commands.

Examples

The following example retrieves the raw data for the state of the job named *myDownloadJob*.

```
C:\>bitsadmin /RawReturn /GetState myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin removecredentials

4/13/2018 • 2 minutes to read • [Edit Online](#)

Removes credentials from a job.

Syntax

```
bitsadmin /RemoveCredentials <Job> <Target> <Scheme>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| Job | The job's display name or GUID |
| Target | SERVER or PROXY |
| Scheme | One of the following: <ul style="list-style-type: none">- BASIC—authentication scheme in which the user name and password are sent in clear-text to the server or proxy.- DIGEST—a challenge-response authentication scheme that uses a server-specified data string for the challenge.- NTLM—a challenge-response authentication scheme that uses the credentials of the user for authentication in a Windows network environment.- NEGOTIATE—also known as the Simple and Protected Negotiation protocol (Snego) is a challenge-response authentication scheme that negotiates with the server or proxy to determine which scheme to use for authentication. Examples are the Kerberos protocol and NTLM.- PASSPORT—a centralized authentication service provided by Microsoft that offers a single logon for member sites. |

Examples

The following example removes credentials from the job named *myDownloadJob*.

```
C:\>bitsadmin /RemoveCredentials myDownloadJob SERVER BASIC
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin replaceremoteprefix

4/13/2018 • 2 minutes to read • [Edit Online](#)

All files in the job whose remote URL begins with *OldPrefix* are changed to use *NewPrefix*.

Syntax

```
bitsadmin /ReplaceRemotePrefix <Job> <OldPrefix> <NewPrefix>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |
| OldPrefix | Existing URL prefix |
| NewPrefix | New URL prefix |

Examples

The following example changes all files in job named *myDownloadJob* whose remote URL begins with [http://stageserver](#) to [http://prodserver](#).

```
C:\>bitsadmin /ReplaceRemotePrefix myDownloadJob http://stageserver http://prodserver
```

Additional information

[Command-Line Syntax Key](#)

bitsadmin reset

4/13/2018 • 2 minutes to read • [Edit Online](#)

Cancels all jobs in the transfer queue that the current user owns.

Syntax

```
bitsadmin /Reset [/AllUsers]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| AllUsers | Optional— cancels all jobs in the queue. |

Remarks

You must have administrator privileges to use the **AllUsers** parameter.

NOTE

Administrators cannot reset jobs created by Local System. Use the task scheduler to schedule this command as task using the Local System credentials.

Examples

The following example cancels all the jobs in the transfer queue for the current user.

```
C:\>bitsadmin /Reset
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin resume

4/13/2018 • 2 minutes to read • [Edit Online](#)

Activates a new or suspended job in the transfer queue.

Syntax

```
bitsadmin /Resume <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Examples

The following example resumes the job named *myDownloadJob*.

```
C:\>bitsadmin /Resume myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin setaclflag

4/13/2018 • 2 minutes to read • [Edit Online](#)

Sets the access control list propagations flags.

Syntax

```
bitsadmin /SetAclFlags <Job> <Flags>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| Job | The job's display name or GUID |
| Flags | Specify one or more of the following flag values: <ul style="list-style-type: none">- O: Copy owner information with file.- G: Copy group information with file.- D: Copy DACL information with file.- S: Copy SACL information with file. |

Remarks

The SetAclFlags command is used to maintain Owner and access control list information when a job is downloading data from a Windows (SMB) share.

Examples

The following example sets the access control list propagation flags for the job named *myDownloadJob* to maintain the owner and group information with the downloaded files.

```
C:\>bitsadmin /setaclflags myDownloadJob OG
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin setcredentials

4/13/2018 • 2 minutes to read • [Edit Online](#)

Adds credentials to a job.

Syntax

```
bitsadmin /SetCredentials <Job> <Target> <Scheme> <Username> <Password>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| Job | The job's display name or GUID |
| Target | SERVER or PROXY |
| Scheme | One of the following: <ul style="list-style-type: none">- BASIC—authentication scheme in which the user name and password are sent in clear-text to the server or proxy.- DIGEST—a challenge-response authentication scheme that uses a server-specified data string for the challenge.- NTLM—a challenge-response authentication scheme that uses the credentials of the user for authentication in a Windows network environment.- NEGOTIATE—also known as the Simple and Protected Negotiation protocol (Snego) is a challenge-response authentication scheme that negotiates with the server or proxy to determine which scheme to use for authentication. Examples are the Kerberos protocol and NTLM.- PASSPORT—a centralized authentication service provided by Microsoft that offers a single logon for member sites. |
| Username | The name of the provided credentials |
| Password | The password associated with the provided <i>Username</i> |

Examples

The following example Adds credentials to the job named *myDownloadJob*.

```
C:\>bitsadmin /RemoveCredentials myDownloadJob SERVER BASIC Edward Password20
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin setdescription

4/13/2018 • 2 minutes to read • [Edit Online](#)

Sets the description of the specified job.

Syntax

```
bitsadmin /SetDescription <Job> <Description>
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------|--------------------------------|
| Job | The job's display name or GUID |
| Description | Text used to describe the job. |

Examples

The following example retrieves the description for the job named *myDownloadJob*.

```
C:\>bitsadmin /SetDescription myDownloadJob "Music Downloads"
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin setdisplayname

4/13/2018 • 2 minutes to read • [Edit Online](#)

Sets the display name of the specified job.

Syntax

```
bitsadmin /SetDisplayName <Job> <DisplayName>
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------|--|
| Job | The job's display name or GUID |
| DisplayName | Text used for the display name of the specified job. |

Examples

The following example sets the display name for the job named *myDownloadJob* to *myDownloadJob2*.

```
C:\>bitsadmin /SetDisplayName myDownloadJob "Download Music Job"
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin setminretrydelay

4/13/2018 • 2 minutes to read • [Edit Online](#)

Sets the length of time, in seconds, that the service waits after encountering a transient error before retrying to transfer the file.

Syntax

```
bitsadmin /SetMinRetryDelay <Job> <RetryDelay>
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|----------------------------------|
| Job | The job's display name or GUID |
| RetryDelay | A number represented in seconds. |

Examples

The following example sets the minimum retry delay for the job named *myDownloadJob* to 35 seconds.

```
C:\>bitsadmin /SetMinRetryDelay myDownloadJob 35
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin setnoprogresstimeout

4/13/2018 • 2 minutes to read • [Edit Online](#)

Sets the length of time, in seconds, that the service tries to transfer the file after a transient error occurs.

Syntax

```
bitsadmin /SetNoProgressTimeout <Job> <TimeOutvalue>
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------|----------------------------------|
| Job | The job's display name or GUID |
| TimeOutvalue | A number represented in seconds. |

Remarks

- The no progress timeout interval begins when the job encounters a transient error.
- The timeout interval stops or resets when a byte of data is successfully transferred.
- If no progress timeout interval exceeds the *TimeOutvalue*, then the job is placed in a fatal error state.

Examples

The following example sets the no progress time out value for the job named *myDownloadJob* to 20 seconds

```
C:\>bitsadmin /SetNoProgressTimeout myDownloadJob 20
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin setnotifycmdline

4/13/2018 • 2 minutes to read • [Edit Online](#)

Sets the command-line command that will run when the job finishes transferring data or when a job enters a state..

Syntax

```
bitsadmin /SetNotifyCmdLine <Job> <ProgramName> [ProgramParameters]
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------------|--|
| Job | The job's display name or GUID |
| ProgramName | Name of the command to run when the job completes. |
| ProgramParameters | Parameters that you want to pass to <i>ProgramName</i> . |

Remarks

You can specify NULL for *ProgramName* and *ProgramParameters*. If *ProgramName* is NULL, *ProgramParameters* must be NULL.

IMPORTANT

If *ProgramParameters* is not NULL, then the first parameter in *ProgramParameters* must match *ProgramName*.

Examples

The following example sets the command-line command used by the service to run notepad when the job named *myDownloadJob* completes.

```
C:\>bitsadmin /SetNotifyCmdLine myDownloadJob c:\winnt\system32\notepad.exe NULL
```

```
C:\>bitsadmin /SetNotifyCmdLine myDownloadJob c:\winnt\system32\notepad.exe "notepad c:\eula.txt"
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin setnotifyflags

4/13/2018 • 2 minutes to read • [Edit Online](#)

Sets the event notification flags for the specified job.

Syntax

```
bitsadmin /SetNotifyFlags <Job> <NotifyFlags>
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------|--------------------------------|
| Job | The job's display name or GUID |
| NotifyFlags | See Remarks |

Remarks

The **NotifyFlags** parameter can contain one or more of the following notification flags.

|-----|-----| |1|Generate an event when all files in the job have been transferred.| |2|Generate an event when an error occurs.| |4|Disable notifications.|

Examples

The following example sets the notify flags for transferred and error events job for job named *myDownloadJob*.

```
C:\>bitsadmin /SetNotifyFlags myDownloadJob 3
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin setpriority

4/13/2018 • 2 minutes to read • [Edit Online](#)

Sets the priority of the specified job.

Syntax

```
bitsadmin /SetPriority <Job> <Priority>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| Job | The job's display name or GUID |
| Priority | One of the following values: <ul style="list-style-type: none">- FOREGROUND- HIGH- NORMAL- LOW |

Examples

The following example sets the priority for the job named *myDownloadJob* to normal.

```
C:\>bitsadmin /SetPriority myDownloadJob NORMAL
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin setproxysettings

4/13/2018 • 2 minutes to read • [Edit Online](#)

Sets the proxy settings for the specified job.

Syntax

```
bitsadmin /SetProxySettings <Job> <Usage> [List] [Bypass]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| Job | The job's display name or GUID |
| Usage | One of the following values: <ul style="list-style-type: none">- PRECONFIG—use the owner's Internet Explorer defaults.- NO_PROXY—do not use a proxy server.- OVERRIDE—use an explicit proxy list and bypass list. A proxy and proxy bypass list must follow.- AUTODETECT—automatically detect proxy settings. |
| List | Used when the <i>Usage</i> parameter is set to OVERRIDE—contains a semicolon or space delimited list of proxy servers to use. |
| Bypass | Used when the <i>Usage</i> parameter is set to OVERRIDE—contains a semicolon or space-delimited list of host names or IP addresses, or both, for which transfers are not to be routed through a proxy. This can be <local> to refer to all servers on the same LAN. Values of NULL or "" may be used for an empty proxy bypass list. |

Examples

The following example sets the proxy settings for the job named *myDownloadJob*.

```
C:\>bitsadmin /SetProxySettings myDownloadJob PRECONFIG
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin setreplyfilename

4/13/2018 • 2 minutes to read • [Edit Online](#)

Specify the path of the file that contains the server reply.

Syntax

```
bitsadmin /SetReplyFileName <Job> <Path>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|------------------------------------|
| Job | The job's display name or GUID |
| Path | Location to place the server reply |

Remarks

Valid only for upload-reply jobs.

Examples

The following example sets the reply filename path for the job named *myDownloadJob*.

```
C:\>bitsadmin /SetReplyFileName myDownloadJob c:\reply
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin suspend

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Suspends the specified job.

Syntax

```
bitsadmin /Suspend <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Remarks

To restart the job, use the [bitsadmin resume](#) command.

Examples

The following example suspends the job named *myDownloadJob*.

```
C:\>bitsadmin /Suspend myDownloadJob
```

additional references

[Command-Line Syntax Key](#)

bitsadmin takeownership

4/13/2018 • 2 minutes to read • [Edit Online](#)

Lets a user with administrative privileges take ownership of the specified job.

Syntax

```
bitsadmin /TakeOwnership <Job>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Examples

The following example takes ownership of the job named *myDownloadJob*.

```
C:\>bitsadmin /TakeOwnership myDownloadJob
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin Transfer

4/13/2018 • 2 minutes to read • [Edit Online](#)

Transfers one or more files.

Syntax

```
bitsadmin /Transfer Name [Type] [/Priority Job_Priority] [/ACLFlags Flags] <RemoteFileName> <LocalFileName>
```

Parameters

| PARAMETER | DESCRIPTION |
|----------------|---|
| Name | The name of the job. |
| Type | Optional—specify the type of job. Use /Download for a download job or /Upload for an upload job. |
| Priority | Optional— set the job_priority to one of the following values: <ul style="list-style-type: none">- FOREGROUND- HIGH- NORMAL- LOW |
| ACLFlags | Specify one or more of the following flags: <ul style="list-style-type: none">- O: Copy owner information with file.- G: Copy group information with file.- D: Copy DACL information with file.- S: Copy SACL information with file. |
| RemoteFileName | The name of the file when transferred to the server |
| LocalFileName | The name of the file that resides locally. |

Remarks

By default, the BITSAdmin service creates a download job that runs at **NORMAL** priority and updates the command window with progress information until the transfer is complete or until a critical error occurs. The service completes the job if it successfully transfers all the files and cancels the job if a critical error occurs. The service does not create the job if it is unable to add files to the job or if you specify an invalid value for *Type* or *Job_Priority*. To transfer more than one file, specify multiple *RemoteFileName-LocalFileName* pairs. The pairs are space-delimited.

NOTE

The BITSAdmin command continues to run if a transient error occurs. To end the command, press CTRL+C.

Examples

The following example starts a transfer job with named *myDownloadJob*.

```
C:\>bitsadmin /Transfer myDownloadJob http://prodserver/audio.wma c:\downloads\audio.wma
```

Additional references

[Command-Line Syntax Key](#)

bitsadmin util

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Syntax

```
bitsadmin /Util /help
bitsadmin /Util /GetIEProxy
bitsadmin /Util /repairService
bitsadmin /Util /SetIEProxy
bitsadmin /Util /version
```

Parameters

| PARAMETER | DESCRIPTION |
|--|--|
| bitsadmin util and help | Displays the command-line usage for the /Util commands. |
| bitsadmin util and getieproxy | Retrieves the proxy usage for the given service account. |
| bitsadmin util and repairservice | repairs known issues with BITS service |
| bitsadmin util and setieproxy | Specifies proxy settings to use when transferring files using a service account. |
| bitsadmin util and version | Displays the version of the BITS service |

additional references

[Command-Line Syntax Key](#)

bitsadmin wrap

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Wraps any line of output text extending beyond the rightmost edge of the command window to the next line.

Syntax

```
bitsadmin /Wrap Job
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------|
| Job | The job's display name or GUID |

Remarks

Specify before other commands. By default, all commands, except the [bitsadmin monitor](#) command, wrap the output.

Examples

The following example retrieves information for the job named *myDownloadJob* and wraps the output.

```
C:\>bitsadmin /Wrap /Info myDownloadJob /verbose
```

additional references

[Command-Line Syntax Key](#)

bootcfg

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Configures, queries, or changes Boot.ini file settings.

Syntax

```
bootcfg <parameter> [arguments...]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------------------------|---|
| bootcfg addsw | adds operating system load options for a specified operating system entry. |
| bootcfg copy | Makes a copy of an existing boot entry, to which you can add command-line options. |
| bootcfg dbg1394 | Configures 1394 port debugging for a specified operating system entry. |
| bootcfg debug | adds or changes the debug settings for a specified operating system entry. |
| bootcfg default | Specifies the operating system entry to designate as the default. |
| bootcfg delete | deletes an operating system entry in the [operating systems] section of the Boot.ini file. |
| bootcfg ems | Enables the user to add or change the settings for redirection of the Emergency Management Services console to a remote computer. |
| bootcfg query | Queries and displays the [boot loader] and [operating systems] section entries from Boot.ini. |
| bootcfg raw | adds operating system load options specified as a string to an operating system entry in the [operating systems] section of the Boot.ini file. |
| bootcfg rmsw | removes operating system load options for a specified operating system entry. |
| bootcfg timeout | changes the operating system time-out value. |

bootcfg addsw

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

adds operating system load options for a specified operating system entry.

Syntax

```
bootcfg /addsw [/s <computer> [/u <Domain>\<User> /p <Password>]] [/mm <MaximumRAM>] [/bv] [/so] [/ng] /id <OSEntryLineNum>
```

Parameters

| TERM | DEFINITION |
|------|--|
| /s | Specifies the name or IP address of a remote computer (do not use backslashes). The default is the local computer. |
| /u \ | Runs the command with the account permissions of the user specified by or \. The default is the permissions of the current logged on user on the computer issuing the command. |
| /p | Specifies the password of the user account that is specified in the /u parameter. |
| /mm | Specifies the maximum amount of RAM, in megabytes, that the operating system can use. The value must be equal to or greater than 32 Megabytes. |
| /bv | adds the /basevideo option to the specified , directing the operating system to use standard VGA mode for the installed video driver. |
| /so | adds the /sos option to the specified <i>OSEntryLineNum</i> , directing the operating system to display device driver names while they are being loaded. |
| /ng | adds the /noguiboot option to the specified , disabling the progress bar that appears before the CTRL+ALT+del logon prompt. |
| /id | Specifies the operating system entry line number in the [operating systems] section of the Boot.ini file to which the operating system load options are added. The first line after the [operating systems] section header is 1. |
| /? | Displays help at the command prompt. |

Examples

The following examples show how you can use the **bootcfg /addsw** command:

```
bootcfg /addsw /mm 64 /id 2
bootcfg /addsw /so /id 3
bootcfg /addsw /so /ng /s srvmain /u hirop1n /id 2
bootcfg /addsw /ng /id 2
bootcfg /addsw /mm 96 /ng /s srvmain /u maindom\hirop1n /p p@ssw23 /id 2
```

additional references

[Command-Line Syntax Key](#)

bootcfg copy

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Makes a copy of an existing boot entry, to which you can add command-line options.

Syntax

```
bootcfg /copy [/s <computer> [/u <Domain>\<User> /p <Password>]] [/d <Description>] [/id <OSEntryLineNum>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| /s | Specifies the name or IP address of a remote computer (do not use backslashes). The default is the local computer. |
| /u \ | Runs the command with the account permissions of the user specified by or \. The default is the permissions of the current logged on user on the computer issuing the command. |
| /p | Specifies the password of the user account that is specified in the /u parameter. |
| /d | Specifies the description for the new operating system entry. |
| /id | Specifies the operating system entry line number in the [operating systems] section of the Boot.ini file to copy. The first line after the [operating systems] section header is 1. |
| /? | Displays help at the command prompt. |

Examples

The following examples show how you can use the **bootcfg /copy** command to copy boot entry 1 and enter "\ABC Server\" as the description:

```
bootcfg /copy /d "\ABC Server\" /id 1
```

additional references

[Command-Line Syntax Key](#)

bootcfg dbg1394

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Configures 1394 port debugging for a specified operating system entry.

Syntax

```
bootcfg /dbg1394 {ON | OFF} [/s <computer> [/u <Domain>\<User> /p <Password>]] [/ch <Channel>] /id <OSEntryLineNum>
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------|--|
| {ON OFF} | Specifies the value for 1394 port debugging. - ON - Enables remote debugging support by adding the /dbg1394 option to the specified . - OFF - Disables remote debugging support by removing the /dbg1394 option from the specified . |
| /s | Specifies the name or IP address of a remote computer (do not use backslashes). The default is the local computer. |
| /u \ | Runs the command with the account permissions of the user specified by or \. The default is the permissions of the current logged on user on the computer issuing the command. |
| /p | Specifies the password of the user account that is specified in the /u parameter. |
| /ch Channel | Specifies the channel to use for debugging. Valid values are integers between 1 and 64. Do not use the /ch parameter if 1394 port debugging is being disabled. |
| /id | Specifies the operating system entry line number in the [operating systems] section of the Boot.ini file to which the 1394 port debugging options are added. The first line after the [operating systems] section header is 1. |
| /? | Displays help at the command prompt. |

Examples

The following examples show how you can use the **bootcfg /dbg1394** command:

```
bootcfg /dbg1394 /id 2
bootcfg /dbg1394 on /ch 1 /id 3
bootcfg /dbg1394 edit /ch 8 /id 2
bootcfg /s srvmain /u maindom\hiropln /p p@ssw23 /dbg1394 off /id 2
```

additional references

[Command-Line Syntax Key](#)

bootcfg default

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Specifies the operating system entry to designate as the default.

Syntax

```
bootcfg /default [/s <computer> [/u <Domain>\<User> /p <Password>]] [/id <OSEntryLineNum>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| /s | Specifies the name or IP address of a remote computer (do not use backslashes). The default is the local computer. |
| /u \ | Runs the command with the account permissions of the user specified by or \. The default is the permissions of the current logged on user on the computer issuing the command. |
| /p | Specifies the password of the user account that is specified in the /u parameter. |
| /id | Specifies the operating system entry line number in the [operating systems] section of the Boot.ini file to designate as default. The first line after the [operating systems] section header is 1. |
| /? | Displays help at the command prompt. |

Examples

The following examples show how you can use the **bootcfg /default** command:

```
bootcfg /default /id 2
bootcfg /default /s srvmain /u maindom\hiropln /p p@ssw23 /id 2
```

additional references

[Command-Line Syntax Key](#)

bootcfg delete

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

deletes an operating system entry in the [operating systems] section of the Boot.ini file.

Syntax

```
bootcfg /delete [/s <computer> [/u <Domain>\<User> /p <Password>]] [/id <OSEntryLineNum>]
```

Parameters

| TERM | DEFINITION |
|------|---|
| /s | Specifies the name or IP address of a remote computer (do not use backslashes). The default is the local computer. |
| /u \ | Runs the command with the account permissions of the user specified by or \. The default is the permissions of the current logged on user on the computer issuing the command. |
| /p | Specifies the password of the user account that is specified in the /u parameter. |
| /id | Specifies the operating system entry line number in the [operating systems] section of the Boot.ini file to delete. The first line after the [operating systems] section header is 1. |
| /? | Displays help at the command prompt. |

Examples

The following examples show how you can use the **bootcfg /delete** command:

```
bootcfg /delete /id 1
bootcfg /delete /s srvmain /u maindom\hiropln /p p@ssW23 /id 3
```

additional references

[Command-Line Syntax Key](#)

bootcfg ems

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Enables the user to add or change the settings for redirection of the Emergency Management Services console to a remote computer. By enabling Emergency Management Services, you add a "redirect=Port#" line to the [boot loader] section of the Boot.ini file and a /redirect option to the specified operating system entry line. The Emergency Management Services feature is enabled only on servers.

Syntax

```
bootcfg /ems {ON | OFF | edit} [/s <computer> [/u <Domain>\<User> /p <Password>]] [/port {COM1 | COM2 | COM3 | COM4 | BIOSSET}] [/baud {9600 | 19200 | 38400 | 57600 | 115200}] [/id <OSEntryLineNum>]
```

Parameters

| PARAMETER | DESCRIPTION |
|---|--|
| {ON OFF edit} | <p>Specifies the value for Emergency Management Services redirection.</p> <p>ON - Enables remote output for the specified . adds a /redirect option to the specified and a redirect=com setting to the [boot loader] section. The value of com is set by the /port parameter.</p> <p>OFF - Disables output to a remote computer. removes the /redirect option from the specified and the redirect=com setting from the [boot loader] section.</p> <p>edit - Allows changes to port settings by changing the redirect=com setting in the [boot loader] section. The value of com is reset to the value specified by the /port parameter.</p> |
| /s | Specifies the name or IP address of a remote computer (do not use backslashes). The default is the local computer. |
| /u \ | Runs the command with the account permissions of the user specified by or \. The default is the permissions of the current logged on user on the computer issuing the command. |
| /p | Specifies the password of the user account that is specified in the /u parameter. |
| /port {COM1 COM2 COM3 COM4 BIOSSET} | Specifies the COM port to be used for redirection. BIOSSET directs Emergency Management Services to get the BIOS settings to determine which port should be used for redirection. Do not use the /port parameter if remotely administered output is being disabled. |

| PARAMETER | DESCRIPTION |
|--|--|
| /baud {9600 19200 38400 57600 115200} | Specifies the baud rate to be used for redirection. Do not use the /baud parameter if remotely administered output is being disabled. |
| /id | Specifies the operating system entry line number to which the Emergency Management Services option is added in the [operating systems] section of the Boot.ini file. The first line after the [operating systems] section header is 1. This parameter is required when the Emergency Management Services value is set to ON or OFF . |
| /? | Displays help at the command prompt. |

Examples

The following examples show how you can use the **bootcfg /ems** command:

```
bootcfg /ems on /port com1 /baud 19200 /id 2
bootcfg /ems on /port biosset /id 3
bootcfg /s srvmain /ems off /id 2
bootcfg /ems edit /port com2 /baud 115200
bootcfg /s srvmain /u maindom\hiropln /p p@ssW23 /ems off /id 2
```

additional references

[Command-Line Syntax Key](#)

bootcfg query

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Queries and displays the [boot loader] and [operating systems] section entries from Boot.ini.

Syntax

```
bootcfg /query [/s <computer> [/u <Domain>\<User> /p <Password>]]
```

Parameters

| TERM | DEFINITION |
|------|--|
| /s | Specifies the name or IP address of a remote computer (do not use backslashes). The default is the local computer. |
| /u \ | Runs the command with the account permissions of the user specified by or \. The default is the permissions of the current logged on user on the computer issuing the command. |
| /p | Specifies the password of the user account that is specified in the /u parameter. |
| /? | Displays help at the command prompt. |

Remarks

- The following is a sample of **bootcfg /query** output:

```
Boot Loader Settings ----- timeout: 30 default: multi(0)disk(0)rdisk(0)partition(1)\WINDOWS Boot
Entries ----- Boot entry ID: 1 Friendly Name: "" path: multi(0)disk(0)rdisk(0)partition(1)\WINDOWS OS Load
Options: /fastdetect /debug /debugport=com1:
```

- The Boot Loader Settings portion of the **bootcfg query** output displays each entry in the [boot loader] section of Boot.ini.
- The Boot Entries portion of the **bootcfg query** output displays the following detail for each operating system entry in the [operating systems] section of Boot.ini: Boot entry ID, Friendly Name, path, and OS Load Options.

Examples The following examples show how you can use the **bootcfg /query** command:

```
bootcfg /query bootcfg /query /s srvmain /u maindom\hiropln /p p@ssw23 bootcfg /query /u hiropln /p p@ssw23
```

additional references [Command-Line Syntax Key](#)

bootcfg raw

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

adds operating system load options specified as a string to an operating system entry in the **[operating systems]** section of the Boot.ini file.

Syntax

```
bootcfg /raw [/s <computer> [/u <Domain>\<User> /p <Password>]] <OSLoadOptionsString> [/id <OSEntryLineNum>] [/a]
```

Parameters

| TERM | DEFINITION |
|------|---|
| /s | Specifies the name or IP address of a remote computer (do not use backslashes). The default is the local computer. |
| /u \ | Runs the command with the account permissions of the user specified by or \. The default is the permissions of the current logged on user on the computer issuing the command. |
| /p | Specifies the password of the user account that is specified in the /u parameter. |
| | Specifies the operating system load options to add to the operating system entry. These load options will replace any existing load options associated with the operating system entry. No validation of is done. |
| /id | Specifies the operating system entry line number in the [operating systems] section of the Boot.ini file to update. The first line after the [operating systems] section header is 1. |
| /a | Specifies that the operating system options being added should be appended to any existing operating system options. |
| /? | Displays help at the command prompt. |

Remarks

- bootcfg raw** is used to add text to the end of an operating system entry, overwriting any existing operating system entry options. This text should contain valid OS Load Options such as **/debug**, **/fastdetect**, **/nodebug**, **/baudrate**, **/crashdebug**, and **/sos**. For example, the following command adds **"/debug /fastdetect"** to the end of the first operating system entry, replacing any previous operating system entry options:

```
bootcfg /raw "/debug /fastdetect" /id 1 ## Examples The following examples show how you can use the
```

bootcfg /raw command:

```
bootcfg /raw "/debug /sos" /id 2 bootcfg /raw /s srvmain /u maindom\hiropIn /p p@ssw23 "/crashdebug " /id 2
```

additional references [Command-Line Syntax Key](#)

bootcfg rmsw

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

removes operating system load options for a specified operating system entry.

Syntax

```
bootcfg /rmsw [/s <computer> [/u <Domain>\<User> [/p <Password>]]] [/mm] [/bv] [/so] [/ng] /id  
<OSEntryLineNum>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| /s | Specifies the name or IP address of a remote computer (do not use backslashes). The default is the local computer. |
| /u \ | Runs the command with the account permissions of the user specified by or \. The default is the permissions of the current logged on user on the computer issuing the command. |
| /p | Specifies the password of the user account that is specified in the /u parameter. |
| /mm | removes the /maxmem option and its associated maximum memory value from the specified . The /maxmem option specifies the maximum amount of RAM that the operating system can use. |
| /bv | removes the /basevideo option from the specified . The /basevideo option directs the operating system to use standard VGA mode for the installed video driver. |
| /so | removes the /sos option from the specified . The /sos option directs the operating system to display device driver names while they are being loaded. |
| /ng | removes the /noguiboot option from the specified . The /noguiboot option disables the progress bar that appears before the CTRL+ALT+del logon prompt. |
| /id | Specifies the operating system entry line number in the [operating systems] section of the Boot.ini file from which the OS Load Options are removed. The first line after the [operating systems] section header is 1. |
| /? | Displays help at the command prompt. |

Examples

The following examples show how you can use the **bootcfg /rmsw** command:

```
bootcfg /rmsw /mm 64 /id 2
bootcfg /rmsw /so /id 3
bootcfg /rmsw /so /ng /s srvmain /u hirop1n /id 2
bootcfg /rmsw /ng /id 2
bootcfg /rmsw /mm 96 /ng /s srvmain /u maindom\hirop1n /p p@ssw23 /id 2
```

additional references

[Command-Line Syntax Key](#)

bootcfg timeout

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

changes the operating system time-out value.

Syntax

```
bootcfg /timeout <timeOutValue> [/s <computer> [/u <Domain\User>/p <Password>]]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------------|--|
| /timeout | Specifies the timeout value in the [boot loader] section. The is the number of seconds the user has to select an operating system from the boot loader screen before NTLDR loads the default. Valid range for is 0-999. If the value is 0, then NTLDR immediately starts the default operating system without displaying the boot loader screen. |
| /s | Specifies the name or IP address of a remote computer (do not use backslashes). The default is the local computer. |
| /u <Domain\User> | Runs the command with the account permissions of the user specified by or <Domain\User>. The default is the permissions of the current logged on user on the computer issuing the command. |
| /p | Specifies the of the user account that is specified in the /u parameter. |
| /? | Displays help at the command prompt. |

Examples

The following examples show how you can use the **bootcfg /timeout** command:

```
bootcfg /timeout 30
bootcfg /s srvmain /u maindom\hiropln /p p@ssw23 /timeout 50
```

additional references

[Command-Line Syntax Key](#)

break

4/13/2018 • 2 minutes to read • [Edit Online](#)

Sets or clears extended CTRL+C checking on MS-DOS systems. If used without parameters, **break** displays the current setting.

NOTE

This command is no longer in use. It is included only to preserve compatibility with existing MS-DOS files, but it has no effect at the command line because the functionality is automatic.

Syntax

```
break=[on|off]
```

Remarks

If command extensions are enabled and running on the Windows platform, inserting the **break** command into a batch file enters a hard-coded breakpoint if being debugged by a debugger.

Additional references

[Command-Line Syntax Key](#)

cacls

9/28/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Displays or modifies discretionary access control lists (DACL) on specified files.

Syntax

```
cacls <filename> [/t] [/m] [/l] [/s[:sddl]] [/e] [/c] [/g user:<perm>] [/r user [...]] [/p user:<perm> [...]]  
[/d user [...]]
```

Parameters

| PARAMETER | DESCRIPTION |
|----------------|--|
| <filename> | Required. Displays Acls of specified files. |
| /t | changes Acls of specified files in the current directory and all subdirectories. |
| /m | changes Acls of volumes mounted to a directory. |
| /l | Work on the Symbolic Link itself versus the target. |
| /s:sddl | replaces the Acls with those specified in the SDDL string (not valid with /e , /g , /r , /p , or /d). |
| /e | edit ACL instead of replacing it. |
| /c | Continue on access denied errors. |
| /g user:<perm> | Grant specified user access rights. Valid values for permission: - n - none - r - read - w - write - c - change (write) - f - full control |
| /r user [...] | Revoke specified user's access rights (only valid with /e). |

| PARAMETER | DESCRIPTION |
|------------------------|--|
| [/p user:<perm> [...]] | replace specified user's access rights. Valid values for permission: - n - none - r - read - w - write - c - change (write) - f - full control |
| [/d user [...]] | Deny specified user access. |
| /? | Displays help at the command prompt. |

Remarks

- This command has been deprecated. Please use [icacls](#) instead.
- Use the following table to interpret the results:

| OUTPUT | ACCESS CONTROL ENTRY (ACE) APPLIES TO |
|-------------------|---|
| OI | Object inherit. This folder and files. |
| CI | Container inherit. This folder and subfolders. |
| IO | Inherit only. The ACE does not apply to the current file/directory. |
| No output message | This folder only. |
| (OI)(CI) | This folder, subfolders, and files. |
| (OI)(CI)(IO) | Subfolders and files only. |
| (CI)(IO) | Subfolders only. |
| (OI)(IO) | Files only. |

- You can use wildcards (**?** and *****) to specify multiple files.
- You can specify more than one user.

additional references

- [Command-Line Syntax Key](#)
- [icacls](#)

call

6/5/2018 • 3 minutes to read • [Edit Online](#)

Calls one batch program from another without stopping the parent batch program. The **call** command accepts labels as the target of the call.

NOTE

Call has no effect at the command prompt when it is used outside of a script or batch file.

For examples of how to use this command, see [Examples](#).

Syntax

```
call [Drive:][Path]<FileName> [<BatchParameters>] [:<Label> [<Arguments>]]
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------------|---|
| [<Drive>:] | Specifies the location and name of the batch program that you want to call. The <i>FileName</i> parameter is required, and it must have a .bat or .cmd extension. |
| <BatchParameters> | Specifies any command-line information required by the batch program. |
| :<Label> | Specifies the label that you want a batch program control to jump to. |
| <Arguments> | Specifies the command-line information to be passed to the new instance of the batch program, beginning at <i>:Label</i> . |
| /? | Displays help at the command prompt. |

Batch parameters

The batch script argument references (**%0**, **%1**, ...) are listed in the following tables.

%* in a batch script refers to all the arguments (for example, **%1**, **%2**, **%3**...)

You can use the following optional syntaxes as substitutions for batch parameters (**%n**):

| BATCH PARAMETER | DESCRIPTION |
|-----------------|--|
| %~1 | Expands %1 and removes surrounding quotation marks (" "). |
| %~f1 | Expands %1 to a fully qualified path. |

| BATCH PARAMETER | DESCRIPTION |
|-----------------|--|
| %~d1 | Expands %1 to a drive letter only. |
| %~p1 | Expands %1 to a path only. |
| %~n1 | Expands %1 to a file name only. |
| %~x1 | Expands %1 to a file name extension only. |
| %~s1 | Expands %1 to a fully qualified path that contains short names only. |
| %~a1 | Expands %1 to the file attributes. |
| %~t1 | Expands %1 to the date and time of file. |
| %~z1 | Expands %1 to the size of the file. |
| %~\$PATH:1 | Searches the directories listed in the PATH environment variable, and expands %1 to the fully qualified name of the first directory found. If the environment variable name is not defined or the file is not found by the search, then this modifier expands to the empty string. |

The following table shows how you can combine modifiers with the batch parameters for compound results:

| BATCH PARAMETER WITH MODIFIER | DESCRIPTION |
|-------------------------------|--|
| %~dp1 | Expands %1 to a drive letter and path only. |
| %~nx1 | Expands %1 to a file name and extension only. |
| %~dp\$PATH:1 | Searches the directories listed in the PATH environment variable for %1, and then expands to the drive letter and path of the first directory found. |
| %~ftza1 | Expands %1 to display output similar to the dir command. |

In the above examples, %1 and PATH can be replaced by other valid values. The %~ syntax is terminated by a valid argument number. The %~ modifiers cannot be used with %*.

Remarks

- Using batch parameters

Batch parameters can contain any information that you can pass to a batch program, including command-line options, file names, the batch parameters %0 through %9, and variables (for example, %baud%).

- Using the *Label* parameter

By using **call** with the *Label* parameter, you create a new batch file context and pass control to the statement after the specified label. The first time the end of the batch file is encountered (that is, after jumping to the label), control returns to the statement after the **call** statement. The second time the end of the batch file is encountered, the batch script is exited.

- Using pipes and redirection symbols

Do not use pipes (|) and redirection symbols (< or >) with **call**.

- Making a recursive call

You can create a batch program that calls itself. However, you must provide an exit condition. Otherwise, the parent and child batch programs can loop endlessly.

- Working with command extensions

If command extensions are enabled, **call** accepts *Label* as the target of the call. The correct syntax is as follows:

```
call :\<Label> <Arguments>
```

Examples

To run the Checknew.bat program from another batch program, type the following command in the parent batch program:

```
call checknew
```

If the parent batch program accepts two batch parameters and you want it to pass those parameters to Checknew.bat, type the following command in the parent batch program:

```
call checknew %1 %2
```

Additional references

[Command-Line Syntax Key](#)

cd

4/13/2018 • 2 minutes to read • [Edit Online](#)

Displays the name of or changes the current directory. If used with only a drive letter (for example, `cd C:`), **cd** displays the names of the current directory in the specified drive. If used without parameters, **cd** displays the current drive and directory.

NOTE

This command is the same as the **chdir** command.

For examples of how to use this command, see [Examples](#).

Syntax

```
cd [/d] [<Drive>:][<Path>]
cd [..]
chdir [/d] [<Drive>:][<Path>]
chdir [..]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| /d | Changes the current drive as well as the current directory for a drive. |
| <Drive>: | Specifies the drive to display or change (if different from the current drive). |
| <Path> | Specifies the path to the directory that you want to display or change. |
| [..] | Specifies that you want to change to the parent folder. |
| /? | Displays help at the command prompt. |

Remarks

If command extensions are enabled, the following conditions apply to the **cd** command:

- The current directory string is converted to use the same case as the names on the disk. For example, `cd C:\TEMP` would set the current directory to C:\Temp if that is the case on the disk.
- Spaces are not treated as delimiters, so *Path* can contain spaces without enclosing quotation marks. For example:

```
cd username\programs\start menu
```

is the same as:

```
cd "username\programs\start menu"
```

The quotation marks are required, however, if extensions are disabled.

To disable command extensions, type:

```
cmd /e:off
```

Examples

The root directory is the top of the directory hierarchy for a drive. To return to the root directory, type:

```
cd\
```

To change the default directory on a drive that is different from the one you are on, type:

```
cd [<Drive>:\[<Directory>]]
```

To verify the change to the directory, type:

```
cd [<Drive>:]
```

Additional references

[Command-Line Syntax Key](#)

Certreq can be used to request certificates from a certification authority (CA), to retrieve a response to a previous request from a CA, to create a new request from an .inf file, to accept and install a response to a request, to construct a cross-certification or qualified subordination request from an existing CA certificate or request, and to sign a cross-certification or qualified subordination request.

WARNING

- Earlier versions of certreq may not provide all of the options that are described in this document. You can see all the options that a specific version of certreq provides by running the commands shown in the Syntax notations section.
- Certreq does not support creating a new certificate request based on a Key Attestation template when in a CEP/CES environment

Contents

The major sections in this article are as follows:

1. [Verbs](#)
2. [Syntax notations](#)
3. [Options](#)
4. [Formats](#)
5. [Additional certreq examples](#)

Verbs

The following table describes the verbs that can be used with the certreq command

| SWITCH | DESCRIPTION |
|----------------------------|---|
| -Submit | Submits a request to a CA. For more information, see Certreq -submit . |
| -retrieve <i>RequestID</i> | Retrieves a response to a previous request from a CA. For more information, see Certreq -retrieve . |
| -New | Creates a new request from an .inf file. For more information, see Certreq -new . |
| -Accept | Accepts and installs a response to a certificate request. For more information, see Certreq -accept . |
| -Policy | Sets the policy for a request. For more information, see Certreq -policy . |
| -Sign | Signs a cross-certification or qualified subordination request. For more information, see Certreq -sign . |

| SWITCH | DESCRIPTION |
|-----------|--|
| -Enroll | Enrolls for or renews a certificate. For more information, see Certreq -enroll . |
| -? | Displays a list of certreq syntax, options, and descriptions. |
| <verb> -? | Displays help for the verb specified. |
| -v -? | Displays a verbose list of the certreq syntax, options, and descriptions. |

Return to [Contents](#)

Syntax notations

- For basic command line syntax, run `certreq -?`
- For the syntax on using certutil with a specific verb, run **certreq** <verb> -?
- To send all of the certutil syntax into a text file, run the following commands:
 - `certreq -v -? > certreqhelp.txt`
 - `notepad certreqhelp.txt`

The following table describes the notation used to indicate command-line syntax.

| NOTATION | DESCRIPTION |
|---------------------------------|--|
| Text without brackets or braces | Items you must type as shown |
| <Text inside angle brackets> | Placeholder for which you must supply a value |
| [Text inside square brackets] | Optional items |
| {Text inside braces} | Set of required items; choose one |
| Vertical bar () | Separator for mutually exclusive items; choose one |
| Ellipsis (...) | Items that can be repeated |

Return to [Contents](#)

Certreq -submit

This is the default certreq.exe parameter, if no option is specified explicitly at the command-line prompt, certreq.exe attempts to submit a certificate request to a CA.

```
CertReq [-Submit] [Options] [RequestFileIn [CertFileOut [CertChainFileOut [FullResponseFileOut]]]]
```

You must specify a certificate request file when using the `-submit` option. If this parameter is omitted, a common File Open window is displayed where you can select the appropriate certificate request file.

You can use these examples as a starting point to build your certificate submit request:

To submit a simple certificate request use the example below:

```
certreq -submit certRequest.req certnew.cer certnew.pfx
```

To request a certificate by specifying the SAN attribute, see the detailed steps in Microsoft Knowledge Base article 931351 [How to add a Subject Alternative Name to a secure LDAP certificate](#) in the "How to use the Certreq.exe utility to create and submit a certificate request that includes a SAN" section.

Return to [Contents](#)

Certreq -retrieve

```
certreq -retrieve [Options] RequestId [CertFileOut [CertChainFileOut [FullResponseFileOut]]]
```

- If you don't specify the CAComputerName or CAName in -config CAComputerName\CAName, a dialog box appears and displays a list of all CAs that are available.
- If you use -config - instead of -config CAComputerName\CAName, the operation is processed using the default CA.
- You can use certreq -retrieve *RequestId* to retrieve the certificate after the CA has actually issued it. The *RequestId* can be a decimal or hex with 0x prefix and it can be a certificate serial number with no 0x prefix. You can also use it to retrieve any certificate that has ever been issued by the CA, including revoked or expired certificates, without regard to whether the certificate's request was ever in the pending state.
- If you submit a request to the CA, the policy module of the CA might leave the request in a pending state and return the *RequestId* to the Certreq caller for display. Eventually, the CA's administrator will issue the certificate or deny the request.

The command below retrieves the certificate id 20 and creates the certificate file (.cer):

```
certreq -retrieve 20 MyCertificate.cer
```

Return to [Contents](#)

Certreq -new

```
certreq -new [Options] [PolicyFileIn [RequestFileOut]]
```

Since the INF file allows for a rich set of parameters and options to be specified, it is difficult to define a default template that administrators should use for all purposes. Therefore, this section describes all the options to enable you to create an INF file tailored to your specific needs. The following key words are used to describe the INF file structure.

1. A *section* is an area in the INF file that covers a logical group of keys. A section always appears in brackets in the INF file.
2. A *key* is the parameter that is to the left of the equal sign.
3. A *value* is the parameter that is to the right of the equal sign.

For example, a minimal INF file would look similar to the following:

```
[NewRequest]
; At least one value must be set in this section
Subject = "CN=W2K8-BO-DC.contoso2.com"
```

The following are some of the possible sections that may be added to the INF file:

[NewRequest]

This section is mandatory for an INF file that acts as a template for a new certificate request. This section requires at least one key with a value.

| KEY | DEFINITION | VALUE | EXAMPLE |
|---------------------|---|--|---|
| Subject | Several applications rely on the subject information in a certificate. Thus, it is recommended that a value for this key be specified. If the subject is not set here, it is recommended that a subject name be included as part of the subject alternative name certificate extension. | Relative Distinguished Name string values | Subject = "CN=computer1.contoso.com" Subject="CN=John Smith,CN=Users,DC=Contoso,DC=com" |
| Exportable | If this attribute is set to TRUE, the private key can be exported with the certificate. To ensure a high level of security, private keys should not be exportable; however, in some cases, it might be required to make the private key exportable if several computers or users must share the same private key. | true, false | Exportable = TRUE. CNG keys can distinguish between this and plaintext exportable. CAPI1 keys cannot. |
| ExportableEncrypted | Specifies whether the private key should be set to be exportable. | true, false | ExportableEncrypted = true Tip: Not all public key sizes and algorithms will work with all hash algorithms. The specified CSP must also support the specified hash algorithm. To see the list of supported hash algorithms, you can run the command <pre>certutil -oid 1 findstr pwszCNGAlgid findstr /v CryptOIDInfo</pre> |
| HashAlgorithm | Hash Algorithm to be used for this request. | Sha256, sha384, sha512, sha1, md5, md4, md2 | HashAlgorithm = sha1. To see the list of supported hash algorithms use: certutil -oid 1 findstr pwszCNGAlgid findstr /v CryptOIDInfo |
| KeyAlgorithm | The algorithm that will be used by the service provider to generate a public and private key pair. | RSA, DH, DSA, ECDH_P256, ECDH_P521, ECDSA_P256, ECDSA_P384, ECDSA_P521 | KeyAlgorithm = RSA |

| KEY | DEFINITION | VALUE | EXAMPLE |
|--------------|--|--|---|
| KeyContainer | <p>It is not recommended to set this parameter for new requests where new key material is generated. The key container is automatically generated and maintained by the system. For requests where the existing key material should be used, this value can be set to the key-container name of the existing key. Use the certutil –key command to display the list of available key containers for the machine context. Use the certutil –key –user command for the current user’s context.</p> | <p>Random string value Tip: You should use double quotes around any INF key value that has blanks or special characters to avoid potential INF parsing issues.</p> | KeyContainer = {C347BD28-7F69-4090-AA16-BC58CF4D749C} |
| KeyLength | <p>Defines the length of the public and private key. The key length has an impact on the security level of the certificate. Greater key length usually provides a higher security level; however, some applications may have limitations regarding the key length.</p> | Any valid key length that is supported by the cryptographic service provider. | KeyLength = 2048 |
| KeySpec | <p>Determines if the key can be used for signatures, for Exchange (encryption), or for both.</p> | AT_NONE, AT_SIGNATURE, AT_KEYEXCHANGE | KeySpec = AT_KEYEXCHANGE |

| KEY | DEFINITION | VALUE | EXAMPLE |
|------------------|---|--|--|
| KeyUsage | Defines what the certificate key should be used for. | CERT_DIGITAL_SIGNATURE_KEY_USAGE -- 80 (128) Tip: The values shown are hexadecimal (decimal) values for each bit definition. Older syntax can also be used: a single hexadecimal value with multiple bits set, instead of the symbolic representation. For example, KeyUsage = 0xa0. CERT_NON_REPUDIATION_KEY_USAGE -- 40 (64) CERT_KEY_ENCIPHERMENT_KEY_USAGE -- 20 (32) CERT_DATA_ENCIPHERMENT_KEY_USAGE -- 10 (16) CERT_KEY_AGREEMENT_KEY_USAGE -- 8 CERT_KEY_CERT_SIGN_KEY_USAGE -- 4 CERT_OFFLINE_CRL_SIGN_KEY_USAGE -- 2 CERT_CRL_SIGN_KEY_USAGE -- 2 CERT_ENCIPHER_ONLY_KEY_USAGE -- 1 CERT_DECIPHER_ONLY_KEY_USAGE -- 8000 (32768) | KeyUsage = "CERT_DIGITAL_SIGNATURE_KEY_USAGE CERT_KEY_ENCIPHERMENT_KEY_USAGE" Tip: Multiple values use a pipe () symbol separator. Ensure that you use double-quotes when using multiple values to avoid INF parsing issues. |
| KeyUsageProperty | Retrieves a value that identifies the specific purpose for which a private key can be used. | NCryptAllowDecryptFlag -- 1 NCryptAllowSigningFlag -- 2 NCryptAllowKeyAgreementFlag -- 4 NCryptAllowAllUsages -- ffffff (16777215) | KeyUsageProperty = "NCryptAllowDecryptFlag NCryptAllowSigningFlag" |

| KEY | DEFINITION | VALUE | EXAMPLE |
|-------------------|---|-----------------------|---|
| MachineKeySet | This key is important when you need to create certificates that are owned by the machine and not a user. The key material that is generated is maintained in the security context of the security principal (user or computer account) that has created the request. When an administrator creates a certificate request on behalf of a computer, the key material must be created in the machine's security context and not the administrator's security context. Otherwise, the machine could not access its private key since it would be in the administrator's security context. | true, false | MachineKeySet = true Tip: The default is false. |
| NotBefore | Specifies a date or date and time before which the request cannot be issued. NotBefore can be used with ValidityPeriod and ValidityPeriodUnits. | date or date and time | NotBefore = "7/24/2012 10:31 AM" Tip: NotBefore and NotAfter are for RequestType=cert only. Date parsing attempts to be locale-sensitive. Using month names will disambiguate and should work in every locale. |
| NotAfter | Specifies a date or date and time after which the request cannot be issued. NotAfter cannot be used with ValidityPeriod or ValidityPeriodUnits. | date or date and time | NotAfter = "9/23/2014 10:31 AM" Tip: NotBefore and NotAfter are for RequestType=cert only. Date parsing attempts to be locale-sensitive. Using month names will disambiguate and should work in every locale. |
| PrivateKeyArchive | The PrivateKeyArchive setting works only if the corresponding RequestType is set to "CMC" because only the Certificate Management Messages over CMS (CMC) request format allows for securely transferring the requester's private key to the CA for key archival. | true, false | PrivateKeyArchive = True |

| KEY | DEFINITION | VALUE | EXAMPLE |
|---------------------|---|---|--|
| EncryptionAlgorithm | The encryption algorithm to use. | <p>Possible options vary, depending on the operating system version and the set of installed cryptographic providers. To see the list of available algorithms, run the command</p> <pre>certutil -oid 2 findstr pwszCNGAlgid</pre> <p>The specified CSP used must also support the specified symmetric encryption algorithm and length.</p> | EncryptionAlgorithm = 3des |
| EncryptionLength | Length of encryption algorithm to use. | Any length allowed by the specified EncryptionAlgorithm. | EncryptionLength = 128 |
| ProviderName | The provider name is the display name of the CSP.. | If you do not know the provider name of the CSP you are using, run certutil – csplist from a command line. The command will display the names of all CSPs that are available on the local system | ProviderName = "Microsoft RSA SChannel Cryptographic Provider" |
| ProviderType | The provider type is used to select specific providers based on specific algorithm capability such as "RSA Full". | If you do not know the provider type of the CSP you are using, run certutil – csplist from a command-line prompt. The command will display the provider type of all CSPs that are available on the local system. | ProviderType = 1 |
| RenewalCert | If you need to renew a certificate that exists on the system where the certificate request is generated, you must specify its certificate hash as the value for this key. | <p>The certificate hash of any certificate that is available at the computer where the certificate request is created. If you do not know the certificate hash, use the Certificates MMC Snap-In and look at the certificate that should be renewed. Open the certificate properties and see the "Thumbprint" attribute of the certificate. Certificate renewal requires either a PKCS#7 or a CMC request format.</p> | RenewalCert = 4EDF274BD2919C6E9EC6A522F0F3B153E9B1582D |

| KEY | DEFINITION | VALUE | EXAMPLE |
|---|--|--|--|
| RequesterName Note: This makes the request to enroll on behalf of another user request. The request must also be signed with an Enrollment Agent certificate, or the CA will reject the request. Use the -cert option to specify the enrollment agent certificate. | The requester name can be specified for certificate requests if the RequestType is set to PKCS#7 or CMC. If the RequestType is set to PKCS#10, this key will be ignored. The Requestername can only be set as part of the request. You cannot manipulate the Requestername in a pending request. | Domain\User | Requestername = "Contoso\BSmith" |
| RequestType | Determines the standard that is used to generate and send the certificate request. | PKCS10 -- 1 PKCS7 -- 2 CMC -- 3 Cert -- 4 Tip: This option indicates a self-signed or self-issued certificate. It does not generate a request, but rather a new certificate and then installs the certificate. Self-signed is the default. Specify a signing cert by using the -cert option to create a self-issued certificate that is not self-signed. | RequestType = CMC |
| SecurityDescriptor Tip: This is relevant only for machine context non-smart card keys. | Contain the security information associated with securable objects. For most securable objects, you can specify an object's security descriptor in the function call that creates the object. | Strings based on security descriptor definition language . | SecurityDescriptor = "D:P(A;GA;;;SY)(A;GA;;;BA)" |
| AlternateSignatureAlgorithm | Specifies and retrieves a Boolean value that indicates whether the signature algorithm object identifier (OID) for a PKCS#10 request or certificate signature is discrete or combined. | true, false | AlternateSignatureAlgorithm = false Tip: For an RSA signature, false indicates a Pkcs1 v1.5. True indicates a v2.1 signature. |
| Silent | By default, this option allows the CSP access to the interactive user desktop and request information such as a smart card PIN from the user. If this key is set to TRUE, the CSP must not interact with the desktop and will be blocked from displaying any user interface to the user. | true, false | Silent = true |

| KEY | DEFINITION | VALUE | EXAMPLE |
|--|--|---|--|
| SMIME | If this parameter is set to TRUE, an extension with the object identifier value 1.2.840.113549.1.9.15 is added to the request. The number of object identifiers depends on the on the operating system version installed and CSP capability, which refer to symmetric encryption algorithms that may be used by Secure Multipurpose Internet Mail Extensions (S/MIME) applications such as Outlook. | true, false | SMIME = true |
| UseExistingKeySet | This parameter is used to specify that an existing key pair should be used in building a certificate request. If this key is set to TRUE, you must also specify a value for the RenewalCert key or the KeyContainer name. You must not set the Exportable key because you cannot change the properties of an existing key. In this case, no key material is generated when the certificate request is built. | true, false | UseExistingKeySet = true |
| KeyProtection | Specifies a value that indicates how a private key is protected before use. | XCN_NCRYPT_UI_NO_PROTECTION_FLAG -- 0 XCN_NCRYPT_UI_PROTECT_KEY_FLAG -- 1 XCN_NCRYPT_UI_FORCE_HIGH_PROTECTION_FLAG -- 2 | KeyProtection = NCRYPT_UI_FORCE_HIGH_PROTECTION_FLAG |
| SuppressDefaults | Specifies a Boolean value that indicates whether the default extensions and attributes are included in the request. The defaults are represented by their object identifiers (OIDs). | true, false | SuppressDefaults = true |
| FriendlyName | A friendly name for the new certificate. | Text | FriendlyName = "Server1" |
| ValidityPeriodUnits Note: This is used only when the request type=cert. | Specifies a number of units that is to be used with ValidityPeriod. | Numeric | ValidityPeriodUnits = 3 |
| ValidityPeriod Note: This is used only when the request type=cert. | VValidityPeriod must be an US English plural time period. | Years, Months, Weeks, Days, Hours, Minutes, Seconds | ValidityPeriod = Years |

[Extensions]

This section is optional.

| EXTENSION OID | DEFINITION | VALUE | EXAMPLE |
|-----------------|------------|-------|---|
| 2.5.29.17 | | | 2.5.29.17 = "{text}" |
| <i>continue</i> | | | <i>continue</i> = "UPN=User@Domain.com&" |
| <i>continue</i> | | | <i>continue</i> = "EMail=User@Domain.com& " |
| <i>continue</i> | | | <i>continue</i> = "DNS=host.domain.com&" |
| <i>continue</i> | | | <i>continue</i> = "DirectoryName=CN=Name, DC=Domain,DC=com&" |
| <i>continue</i> | | | <i>continue</i> = "URL= http://host.domain.com/default.html &" |
| <i>continue</i> | | | <i>continue</i> = "IPAddress=10.0.0.1&" |
| <i>continue</i> | | | <i>continue</i> = "RegisteredId=1.2.3.4.5&" |
| <i>continue</i> | | | <i>continue</i> = "1.2.3.4.6.1= {utf8}String&" |
| <i>continue</i> | | | <i>continue</i> = "1.2.3.4.6.2= {octet}AAECAwQFBgc=&" |
| <i>continue</i> | | | <i>continue</i> = "1.2.3.4.6.2= {octet}{hex}00 01 02 03 04 05 06 07&" |
| <i>continue</i> | | | <i>continue</i> = "1.2.3.4.6.3= {asn}BAgAAQIDBAUGBw== &" |
| <i>continue</i> | | | <i>continue</i> = "1.2.3.4.6.3= {hex}04 08 00 01 02 03 04 05 06 07" |
| 2.5.29.37 | | | 2.5.29.37="{text}" |
| <i>continue</i> | | | <i>continue</i> = "1.3.6.1.5.5.7. |
| <i>continue</i> | | | <i>continue</i> = "1.3.6.1.5.5.7.3.1" |

| EXTENSION OID | DEFINITION | VALUE | EXAMPLE |
|------------------|------------|-------|---|
| 2.5.29.19 | | | "{text}ca=0pathlength=3" |
| Critical | | | Critical=2.5.29.19 |
| KeySpec | | | AT_NONE -- 0 AT_SIGNATURE -- 2 AT_KEYEXCHANGE -- 1 |
| RequestType | | | PKCS10 -- 1 PKCS7 -- 2 CMC -- 3 Cert -- 4 |
| KeyUsage | | | CERT_DIGITAL_SIGNATURE_ KEY_USAGE -- 80 (128) CERT_NON_REPUDIATION_ KEY_USAGE -- 40 (64) CERT_KEY_ENCIPHERMENT_ KEY_USAGE -- 20 (32) CERT_DATA_ENCIPHERMENT_ KEY_USAGE -- 10 (16) CERT_KEY_AGREEMENT_ KEY_USAGE -- 8 CERT_KEY_CERT_SIGN_ KEY_USAGE -- 4 CERT_OFFLINE_CRL_SIGN_ KEY_USAGE -- 2 CERT_CRL_SIGN_ KEY_USAGE -- 2 CERT_ENCIPHER_ONLY_ KEY_USAGE -- 1 CERT_DECIPHER_ONLY_ KEY_USAGE -- 8000 (32768) |
| KeyUsageProperty | | | NCRYPT_ALLOW_DECRYPT_ FLAG -- 1 NCRYPT_ALLOW_SIGNING_ FLAG -- 2 NCRYPT_ALLOW_KEY_ AGREEMENT_FLAG -- 4 NCRYPT_ALLOW_ALL_ USAGES -- ffffff (16777215) |
| KeyProtection | | | NCRYPT_UI_NO_PROTECTIO N_FLAG -- 0 NCRYPT_UI_PROTECT_KEY_ FLAG -- 1 NCRYPT_UI_FORCE_HIGH_P ROTECTION_FLAG -- 2 |

| EXTENSION OID | DEFINITION | VALUE | EXAMPLE |
|------------------|------------|-------|---|
| SubjectNameFlags | template | | CT_FLAG_SUBJECT_REQUIRE _COMMON_NAME -- 40000000 (1073741824) CT_FLAG_SUBJECT_REQUIRE _DIRECTORY_PATH -- 80000000 (2147483648) CT_FLAG_SUBJECT_REQUIRE _DNS_AS_CN -- 10000000 (268435456) CT_FLAG_SUBJECT_REQUIRE _EMAIL -- 20000000 (536870912) CT_FLAG_OLD_CERT_SUPPLI ES_SUBJECT_AND_ALT_NAM E -- 8 CT_FLAG_SUBJECT_ALT_REQ UIRE_DIRECTORY_GUID -- 1000000 (16777216) CT_FLAG_SUBJECT_ALT_REQ UIRE_DNS -- 8000000 (134217728) CT_FLAG_SUBJECT_ALT_REQ UIRE_DOMAIN_DNS -- 400000 (4194304) CT_FLAG_SUBJECT_ALT_REQ UIRE_EMAIL -- 4000000 (67108864) CT_FLAG_SUBJECT_ALT_REQ UIRE_SPN -- 800000 (8388608) CT_FLAG_SUBJECT_ALT_REQ UIRE_UPN -- 2000000 (33554432) |

| EXTENSION OID | DEFINITION | VALUE | EXAMPLE |
|---------------|------------|-------|--|
| X500NameFlags | | | CERT_NAME_STR_NONE -- 0 CERT_OID_NAME_STR -- 2 CERT_X500_NAME_STR -- 3 CERT_NAME_STR_SEMICOLON_FLAG -- 40000000 (1073741824) CERT_NAME_STR_NO_PLUS_FLAG -- 20000000 (536870912) CERT_NAME_STR_NO_QUOTING_FLAG -- 10000000 (268435456) CERT_NAME_STR_CRLF_FLAG -- 8000000 (134217728) CERT_NAME_STR_COMMA_FLAG -- 4000000 (67108864) CERT_NAME_STR_REVERSE_FLAG -- 2000000 (33554432) CERT_NAME_STR_FORWARD_FLAG -- 1000000 (16777216) CERT_NAME_STR_DISABLE_IE4_UTF8_FLAG -- 10000 (65536) CERT_NAME_STR_ENABLE_T61_UNICODE_FLAG -- 20000 (131072) CERT_NAME_STR_ENABLE_UTF8_UNICODE_FLAG -- 40000 (262144) CERT_NAME_STR_FORCE_UTF8_DIR_STR_FLAG -- 80000 (524288) CERT_NAME_STR_DISABLE_UTF8_DIR_STR_FLAG -- 100000 (1048576) CERT_NAME_STR_ENABLE_PUNYCODE_FLAG -- 200000 (2097152) |

Return to [Contents](#)

NOTE

SubjectNameFlags allows the INF file to specify which Subject and SubjectAltName extension fields should be auto-populated by certreq based on the current user or current machine properties: DNS name, UPN, and so on. Using the literal "template" means the template name flags are used instead. This allows a single INF file to be used in multiple contexts to generate requests with context-specific subject information.

X500NameFlags specifies the flags to be passed directly to CertStrToName API when the Subject INF keys value is converted to an ASN.1 encoded Distinguished Name.

To request a certificate based using certreq -new use the steps from the example below:

WARNING

The content for this topic is based on the default settings for Windows Server 2008 AD CS; for example, setting the key length to 2048, selecting Microsoft Software Key Storage Provider as the CSP, and using Secure Hash Algorithm 1 (SHA1). Evaluate these selections against the requirements of your company's security policy.

To create a Policy File (.inf) copy and save the example below in Notepad and save as RequestConfig.inf:

```
[NewRequest]
Subject = "CN=<FQDN of computer you are creating the certificate>"
Exportable = TRUE
KeyLength = 2048
KeySpec = 1
KeyUsage = 0xf0
MachineKeySet = TRUE
[RequestAttributes]
CertificateTemplate="WebServer"
[Extensions]
OID = 1.3.6.1.5.5.7.3.1
OID = 1.3.6.1.5.5.7.3.2
```

On the computer for which you are requesting a certificate type the command below:

```
CertReq -New RequestConfig.inf CertRequest.req
```

The following example demonstrates implementing the [Strings] section syntax for OIDs and other difficult to interpret data. The new {text} syntax example for EKU extension, which uses a comma separated list of OIDs:

```
[Version]
Signature="$Windows NT$"

[Strings]
szOID_ENHANCED_KEY_USAGE = "2.5.29.37"
szOID_PKIX_KP_SERVER_AUTH = "1.3.6.1.5.5.7.3.1"
szOID_PKIX_KP_CLIENT_AUTH = "1.3.6.1.5.5.7.3.2"

[NewRequest]
Subject = "CN=TestSelfSignedCert"
Requesttype = Cert

[Extensions]
%szOID_ENHANCED_KEY_USAGE%="{text}%szOID_PKIX_KP_SERVER_AUTH%, "
_continue_ = "%szOID_PKIX_KP_CLIENT_AUTH%"
```

Return to [Contents](#)

Certreq -accept

```
CertReq -accept [Options] [CertChainFileIn | FullResponseFileIn | CertFileIn]
```

The -accept parameter links the previously generated private key with the issued certificate and removes the pending certificate request from the system where the certificate is requested (if there is a matching request).

You can use this example for manually accepting a certificate:

```
certreq -accept certnew.cer
```

WARNING

The `-accept` verb, the `-user` and `-machine` options indicate whether the cert being installed should be installed in user or machine context. If there's an outstanding request in either context that matches the public key being installed, then these options are not needed. If there is no outstanding request, then one of these must be specified.

Return to [Contents](#)

Certreq -policy

```
certreq -policy [-attrib AttributeString] [-binary] [-cert CertID] [RequestFileIn [PolicyFileIn [RequestFileOut  
[PKCS10FileOut]]]]
```

- The configuration file that defines the constraints that are applied to a CA certificate when qualified subordination is defined is called Policy.inf..
- You can find an example of the Policy.inf file in the [Appendix A of Planning and Implementing Cross-Certification and Qualified Subordination](#) white paper.
- If you type the `certreq -policy` without any additional parameter it will open a dialog window so you can select the requested file (req, cmc, txt, der, cer or crt). Once you select the requested file and click Open button, another dialog window will open in order to select the INF file.

You can use this example to build a cross certificate request:

```
certreq -policy Certsrv.req Policy.inf newcertsrv.req
```

Return to [Contents](#)

Certreq -sign

```
certreq -sign [Options] [RequestFileIn [RequestFileOut]]
```

- If you type the `certreq -sign` without any additional parameter it will open a dialog window so you can select the requested file (req, cmc, txt, der, cer or crt).
- Signing the qualified subordination request may require Enterprise Administrator credentials. This is a best practice for issuing signing certificates for qualified subordination.
- The certificate used to sign the qualified subordination request is created using the qualified subordination template. Enterprise Admins will have to sign the request or grant user permissions for the individuals that will sign the certificate.
- When you sign the CMC request, you may need to have multiple personnel sign this request, depending on the assurance level that is associated with the qualified subordination.
- If the parent CA of the qualified subordinate CA you are installing is offline, you must obtain the CA certificate for the qualified subordinate CA from the offline parent. If the parent CA is online, specify the CA certificate for the qualified subordinate CA during the Certificate Services Installation Wizard.

The sequence of commands below will show how to create a new certificate request, sign it and submit it:

```
certreq -new policyfile.inf MyRequest.req
certreq -sign MyRequest.req MyRequest_Sign.req
certreq -submit MyRequest_Sign.req MyRequest_cert.cer
```

Return to [Contents](#)

Certreq -enroll

To enroll to a certificate

```
certreq -enroll [Options] TemplateName
```

To renew an existing certificate

```
certreq -enroll -cert CertId [Options] Renew [ReuseKeys]
```

You can only renew certificates that are time valid. Expired certificates cannot be renewed and must be replaced with a new certificate.

Here an example of renewing a certificate using its serial number:

```
certreq -enroll -machine -cert "61 2d 3c fe 00 00 00 00 00 05" Renew
```

Here an example of enrolling to a certificate template called WebServer by using asterisk (*) to select the policy server via U/I:

```
certreq -enroll -machine -policyserver * "WebServer"
```

Return to [Contents](#)

Options

| OPTIONS | DESCRIPTION |
|------------------------------|--|
| -any | Force ICertRequest::Submit to determine encoding type. |
| -attrib <AttributeString> | Specifies the Name and Value string pairs, separated by a colon. Separate Name and Value string pairs with \n (for example, Name1:Value1\nName2:Value2). |
| -binary | Formats output files as binary instead of base64-encoded. |
| -PolicyServer <PolicyServer> | "ldap: <path>" Insert the URI or unique ID for a computer running the Certificate Enrollment Policy Web Service. To specify that you would like to use a request file by browsing, just use a minus (-) sign for <policyserver>. |

| OPTIONS | DESCRIPTION |
|-----------------------------------|---|
| -config <ConfigString> | Processes the operation by using the CA specified in the configuration string, which is CAHostName\CAName. For an https connection, specify the enrollment server URI. For the local machine store CA, use a minus (-) sign. |
| -Anonymous | Use anonymous credentials for Certificate Enrollment Web Services. |
| -Kerberos | Use Kerberos (domain) credentials for Certificate Enrollment Web Services. |
| -ClientCertificate <ClientCertId> | You can replace the <ClientCertID> with a certificate thumbprint, CN, EKU, template, email, UPN, and the new name=value syntax. |
| -UserName <UserName> | Used with Certificate Enrollment Web Services. You can substitute <UserName> with the SAM name or domain\user. This option is for use with the -p option. |
| -p <Password> | Used with Certificate Enrollment Web Services. Substitute <Password> with the actual user's password. This option is for use with the -UserName option. |
| -user | Configures the -user context for a new certificate request or specifies the context for an a certificate acceptance. This is the default context, if none is specified in the INF or template. |
| -machine | Configures a new certificate request or specifies the context for an a certificate acceptance for the machine context. For new requests it must be consistent with the MachineKeyset INF key and the template context. If this option is not specified and the template does not set a context, then the default is the user context. |
| -crl | Includes certificate revocation lists (CRLs) in the output to the base64-encoded PKCS #7 file specified by CertChainFileOut or to the base64-encoded file specified by RequestFileOut. |
| -rpc | Instructs Active Directory Certificate Services (AD CS) to use a remote procedure call (RPC) server connection instead of Distributed COM. |
| -AdminForceMachine | Use the Key Service or impersonation to submit the request from Local System context. Requires that the user invoking this option be a member of Local Administrators. |
| -RenewOnBehalfOf | Submit a renewal on behalf of the subject identified in the signing certificate. This sets CR_IN_ROBO when calling ICertRequest::Submit |
| -f | Force existing files to be overwritten. This also bypasses caching templates and policy. |
| -q | Use silent mode; suppress all interactive prompts. |

| OPTIONS | DESCRIPTION |
|--------------|---|
| -Unicode | Writes Unicode output when standard output is redirected or piped to another command, which helps when invoked from Windows PowerShell® scripts). |
| -UnicodeText | Sends Unicode output when writing base64 text encoded data blobs to files. |

Return to [Contents](#)

Formats

| FORMATS | DESCRIPTION |
|---------------------|--|
| RequestFileIn | Base64-encoded or binary input file name: PKCS #10 certificate request, CMS certificate request, PKCS #7 certificate renewal request, X.509 certificate to be cross-certified, or KeyGen tag format certificate request. |
| RequestFileOut | Base64-encoded output file name |
| CertFileOut | Base64-encoded X-509 file name. |
| PKCS10FileOut | For use with the Certreq -policy verb only. Base64-encoded PKCS10 output file name. |
| CertChainFileOut | Base64-encoded PKCS #7 file name. |
| FullResponseFileOut | Base64-encoded full response file name. |
| PolicyFileIn | For use with the Certreq -policy verb only. INF file containing a textual representation of extensions used to qualify a request. |

Additional certreq examples

The following articles contain examples of certreq usage:

- [How to Request a Certificate With a Custom Subject Alternative Name](#)
- [Test Lab Guide: Deploying an AD CS Two-Tier PKI Hierarchy](#)
- [Appendix 3: Certreq.exe Syntax](#)
- [How to create a web server SSL certificate manually](#)
- [Request an AMT Provisioning Certificate Using a Windows Server 2008 CA](#)
- [Certificate Enrollment for System Center Operations Manager Agent](#)
- [AD CS Step by Step Guide: Two Tier PKI Hierarchy Deployment](#)
- [How to enable LDAP over SSL with a third-party certification authority](#)

Return to [Contents](#)

Certutil.exe is a command-line program that is installed as part of Certificate Services. You can use Certutil.exe to dump and display certification authority (CA) configuration information, configure Certificate Services, backup and restore CA components, and verify certificates, key pairs, and certificate chains.

When certutil is run on a certification authority without additional parameters, it displays the current certification authority configuration. When cerutil is run on a non-certification authority, the command defaults to running the certutil `-dump` verb.

WARNING

Earlier versions of certutil may not provide all of the options that are described in this document. You can see all the options that a specific version of certutil provides by running the commands shown in the [Syntax notations](#) section.

Menu

The major sections in this document are:

- [Verbs](#)
- [Syntax notations](#)
- [Options](#)
- [Additional certutil examples](#)

Verbs

The following table describes the verbs that can be used with the certutil command.

| VERBS | DESCRIPTION |
|-----------------------------|--|
| <code>-dump</code> | Dump configuration information or files |
| <code>-asn</code> | Parse ASN.1 file |
| <code>-decodehex</code> | Decode hexadecimal-encoded file |
| <code>-decode</code> | Decode a Base64-encoded file |
| <code>-encode</code> | Encode a file to Base64 |
| <code>-deny</code> | Deny a pending certificate request |
| <code>-resubmit</code> | Resubmit a pending certificate request |
| <code>-setattributes</code> | Set attributes for a pending certificate request |
| <code>-setextension</code> | Set an extension for a pending certificate request |

| VERBS | DESCRIPTION |
|--------------|--|
| -revoke | Revoke a certificate |
| -isvalid | Display the disposition of the current certificate |
| -getconfig | Get the default configuration string |
| -ping | Attempt to contact the Active Directory Certificate Services Request interface |
| -pingadmin | Attempt to contact the Active Directory Certificate Services Admin interface |
| -CAInfo | Display information about the certification authority |
| -ca.cert | Retrieve the certificate for the certification authority |
| -ca.chain | Retrieve the certificate chain for the certification authority |
| -GetCRL | Get a certificate revocation list (CRL) |
| -CRL | Publish new certificate revocation lists (CRLs) [or only delta CRLs] |
| -shutdown | Shutdown Active Directory Certificate Services |
| -installCert | Install a certification authority certificate |
| -renewCert | Renew a certification authority certificate |
| -schema | Dump the schema for the certificate |
| -view | Dump the certificate view |
| -db | Dump the raw database |
| -deleterow | Delete a row from the server database |
| -backup | Backup Active Directory Certificate Services |
| -backupDB | Backup the Active Directory Certificate Services database |
| -backupKey | Backup the Active Directory Certificate Services certificate and private key |
| -restore | Restore Active Directory Certificate Services |
| -restoreDB | Restore the Active Directory Certificate Services database |
| -restoreKey | Restore the Active Directory Certificate Services certificate and private key |

| VERBS | DESCRIPTION |
|--------------------------|--|
| -importPFX | Import certificate and private key |
| -dynamicfilelist | Display a dynamic file list |
| -databaselocations | Display database locations |
| -hashfile | Generate and display a cryptographic hash over a file |
| -store | Dump the certificate store |
| -addstore | Add a certificate to the store |
| -delstore | Delete a certificate from the store |
| -verifystore | Verify a certificate in the store |
| -repairstore | Repair a key association or update certificate properties or the key security descriptor |
| -viewstore | Dump the certificates store |
| -viewdelstore | Delete a certificate from the store |
| -dsPublish | Publish a certificate or certificate revocation list (CRL) to Active Directory |
| -ADTemplate | Display AD templates |
| -Template | Display certificate templates |
| -TemplateCAs | Display the certification authorities (CAs) for a certificate template |
| -CATemplates | Display templates for CA |
| -SetCASites | Manage Site Names for CAs |
| -enrollmentServerURL | Display, add or delete enrollment server URLs associated with a CA |
| -ADCA | Display AD CAs |
| -CA | Display Enrollment Policy CAs |
| -Policy | Display Enrollment Policy |
| -PolicyCache | Display or delete Enrollment Policy Cache entries |
| -CredStore | Display, add or delete Credential Store entries |
| -InstallDefaultTemplates | Install default certificate templates |

| VERBS | DESCRIPTION |
|-------------------------|---|
| -URLCache | Display or delete URL cache entries |
| -pulse | Pulse auto enrollment events |
| -MachineInfo | Display information about the Active Directory machine object |
| -DCInfo | Display information about the domain controller |
| -EntInfo | Display information about an enterprise CA |
| -TCAInfo | Display information about the CA |
| -SCInfo | Display information about the smart card |
| -SCRoots | Manage smart card root certificates |
| -verifykeys | Verify a public or private key set |
| -verify | Verify a certificate, certificate revocation list (CRL), or certificate chain |
| -verifyCTL | Verify AuthRoot or Disallowed Certificates CTL |
| -sign | Re-sign a certificate revocation list (CRL) or certificate |
| -vroot | Create or delete web virtual roots and file shares |
| -vocsproot | Create or delete web virtual roots for an OCSP web proxy |
| -addEnrollmentServer | Add an Enrollment Server application |
| -deleteEnrollmentServer | Delete an Enrollment Server application |
| -addPolicyServer | Add a Policy Server application |
| -deletePolicyServer | Delete a Policy Server application |
| -oid | Display the object identifier or set a display name |
| -error | Display the message text associated with an error code |
| -getreg | Display a registry value |
| -setreg | Set a registry value |
| -delreg | Delete a registry value |
| -ImportKMS | Import user keys and certificates into the server database for key archival |

| VERBS | DESCRIPTION |
|----------------------------------|--|
| -ImportCert | Import a certificate file into the database |
| -GetKey | Retrieve an archived private key recovery blob |
| -RecoverKey | Recover an archived private key |
| -MergePFX | Merge PFX files |
| -ConvertEPF | Convert a PFX file into an EPF file |
| -? | Displays the list of verbs |
| -<verb> -? | Displays help for the verb specified. |
| -? -v | Displays a full list of verbs and |

Return to [Menu](#)

Syntax notations

- For basic command line syntax, run `certutil -?`
- For the syntax on using certutil with a specific verb, run **certutil** *<verb>* **-?**
- To send all of the certutil syntax into a text file, run the following commands:
 - `certutil -v -? > certutilhelp.txt`
 - `notepad certutilhelp.txt`

The following table describes the notation used to indicate command-line syntax.

| NOTATION | DESCRIPTION |
|---------------------------------|---|
| Text without brackets or braces | Items you must type as shown |
| <Text inside angle brackets> | Placeholder for which you must supply a value |
| [Text inside square brackets] | Optional items |
| {Text inside braces} | Set of required items; choose one |
| Vertical bar () | |
| Ellipsis (...) | Items that can be repeated |

Return to [Menu](#)

-dump

CertUtil [Options] [-dump]

CertUtil [Options] [-dump] File

Dump configuration information or files

[-f] [-silent] [-split] [-p Password] [-t Timeout]

Return to [Menu](#)

-asn

CertUtil [Options] -asn File [type]

Parse ASN.1 file

type: numeric CRYPT_STRING_* decoding type

Return to [Menu](#)

-decodehex

CertUtil [Options] -decodehex InFile OutFile [type]

type: numeric CRYPT_STRING_* encoding type

[-f]

Return to [Menu](#)

-decode

CertUtil [Options] -decode InFile OutFile

Decode Base64-encoded file

[-f]

Return to [Menu](#)

-encode

CertUtil [Options] -encode InFile OutFile

Encode file to Base64

[-f] [-UnicodeText]

Return to [Menu](#)

-deny

CertUtil [Options] -deny RequestId

Deny pending request

[-config Machine\CAName]

Return to [Menu](#)

-resubmit

CertUtil [Options] -resubmit RequestId

Resubmit pending request

[-config Machine\CAName]

Return to [Menu](#)

-setattributes

CertUtil [Options] -setattributes RequestId AttributeString

Set attributes for pending request

RequestId -- numeric Request Id of pending request

AttributeString -- Request Attribute name and value pairs

- Names and values are colon separated.
- Multiple name, value pairs are newline separated.
- Example: "CertificateTemplate\User\nEMail:User@Domain.com"
- Each "\n" sequence is converted to a newline separator.

[-config Machine\CAName]

Return to [Menu](#)

-setextension

CertUtil [Options] -setextension RequestId ExtensionName Flags {Long | Date | String | @InFile}

Set extension for pending request

RequestId -- numeric Request Id of a pending request

ExtensionName -- ObjectId string of the extension

Flags -- 0 is recommended. 1 makes the extension critical, 2 disables it, 3 does both.

If the last parameter is numeric, it is taken as a Long.

If it can be parsed as a date, it is taken as a Date.

If it starts with '@', the rest of the token is the filename containing binary data or an ascii-text hex dump.

Anything else is taken as a String.

[-config Machine\CAName]

Return to [Menu](#)

-revoke

CertUtil [Options] -revoke SerialNumber [Reason]

Revoke Certificate

SerialNumber: Comma separated list of certificate serial numbers to revoke

Reason: numeric or symbolic revocation reason

- 0: CRL_REASON_UNSPECIFIED: Unspecified (default)
- 1: CRL_REASON_KEY_COMPROMISE: Key Compromise
- 2: CRL_REASON_CA_COMPROMISE: CA Compromise
- 3: CRL_REASON_AFFILIATION_CHANGED: Affiliation Changed
- 4: CRL_REASON_SUPERSEDED: Superseded

- 5: CRL_REASON_CESSATION_OF_OPERATION: Cessation of Operation
- 6: CRL_REASON_CERTIFICATE_HOLD: Certificate Hold
- 8: CRL_REASON_REMOVE_FROM_CRL: Remove From CRL
- -1: Unrevoke: Unrevoke

[-config Machine\CAName]

Return to [Menu](#)

-isvalid

CertUtil [Options] -isvalid SerialNumber | CertHash

Display current certificate disposition

[-config Machine\CAName]

Return to [Menu](#)

-getconfig

CertUtil [Options] -getconfig

Get default configuration string

[-config Machine\CAName]

Return to [Menu](#)

-ping

CertUtil [Options] -ping [MaxSecondsToWait | CAMachineList]

Ping Active Directory Certificate Services Request interface

CAMachineList -- Comma-separated CA machine name list

1. For a single machine, use a terminating comma
2. Displays the site cost for each CA machine

[-config Machine\CAName]

Return to [Menu](#)

-CAInfo

CertUtil [Options] -CAInfo [InfoName [Index | ErrorCode]]

Display CA Information

InfoName -- indicates the CA property to display (see below). Use "*" for all properties.

Index -- optional zero-based property index

ErrorCode -- numeric error code

[-f] [-split] [-config Machine\CAName]

InfoName argument syntax:

- file: File version

- product: Product version
- exitcount: Exit module count
- exit [Index]: Exit module description
- policy: Policy module description
- name: CA name
- sanitizedname: Sanitized CA name
- dsname: Sanitized CA short name (DS name)
- sharedfolder: Shared folder
- error1 ErrorCode: Error message text
- error2 ErrorCode: Error message text and error code
- type: CA type
- info: CA info
- parent: Parent CA
- certcount: CA cert count
- xchgcount: CA exchange cert count
- kracount: KRA cert count
- kraused: KRA cert used count
- propidmax: Maximum CA PropId
- certstate [Index]: CA cert
- certversion [Index]: CA cert version
- certstatuscode [Index]: CA cert verify status
- crlstate [Index]: CRL
- krastate [Index]: KRA cert
- crossstate+ [Index]: Forward cross cert
- crossstate- [Index]: Backward cross cert
- cert [Index]: CA cert
- certchain [Index]: CA cert chain
- certcrlchain [Index]: CA cert chain with CRLs
- xchg [Index]: CA exchange cert
- xchgchain [Index]: CA exchange cert chain
- xchgcrlchain [Index]: CA exchange cert chain with CRLs
- kra [Index]: KRA cert
- cross+ [Index]: Forward cross cert
- cross- [Index]: Backward cross cert
- CRL [Index]: Base CRL
- deltacr [Index]: Delta CRL
- crlstatus [Index]: CRL Publish Status
- deltacrstatus [Index]: Delta CRL Publish Status
- dns: DNS Name
- role: Role Separation
- ads: Advanced Server
- templates: Templates
- ocsp [Index]: OCSP URLs
- aia [Index]: AIA URLs
- cdp [Index]: CDP URLs
- localename: CA locale name

- subjecttemplateoids: Subject Template OIDs

Return to [Menu](#)

-ca.cert

CertUtil [Options] -ca.cert OutCACertFile [Index]

Retrieve the CA's certificate

OutCACertFile: output file

Index: CA certificate renewal index (defaults to most recent)

[-f] [-split] [-config Machine\CAName]

Return to [Menu](#)

-ca.chain

CertUtil [Options] -ca.chain OutCACertChainFile [Index]

Retrieve the CA's certificate chain

OutCACertChainFile: output file

Index: CA certificate renewal index (defaults to most recent)

[-f] [-split] [-config Machine\CAName]

Return to [Menu](#)

-GetCRL

CertUtil [Options] -GetCRL OutFile [Index] [delta]

Get CRL

Index: CRL index or key index (defaults to CRL for newest key)

delta: delta CRL (default is base CRL)

[-f] [-split] [-config Machine\CAName]

Return to [Menu](#)

-CRL

CertUtil [Options] -CRL [dd:hh | republish] [delta]

Publish new CRLs [or delta CRLs only]

dd:hh -- new CRL validity period in days and hours

republish -- republish most recent CRLs

delta -- delta CRLs only (default is base and delta CRLs)

[-split] [-config Machine\CAName]

Return to [Menu](#)

-shutdown

CertUtil [Options] -shutdown

Shutdown Active Directory Certificate Services

[-config Machine\CAName]

Return to [Menu](#)

-installCert

CertUtil [Options] -installCert [CACertFile]

Install Certification Authority certificate

[-f] [-silent] [-config Machine\CAName]

Return to [Menu](#)

-renewCert

CertUtil [Options] -renewCert [ReuseKeys] [Machine\ParentCAName]

Renew Certification Authority certificate

Use -f to ignore an outstanding renewal request, and generate a new request.

[-f] [-silent] [-config Machine\CAName]

Return to [Menu](#)

-schema

CertUtil [Options] -schema [Ext | Attrib | CRL]

Dump Certificate Schema

Defaults to Request and Certificate table

Ext: Extension table

Attrib: Attribute table

CRL: CRL table

[-split] [-config Machine\CAName]

Return to [Menu](#)

-view

CertUtil [Options] -view [Queue | Log | LogFail | Revoked | Ext | Attrib | CRL] [csv]

Dump Certificate View

Queue: Request queue

Log: Issued or revoked certificates, plus failed requests

LogFail: Failed requests

Revoked: Revoked certificates

Ext: Extension table

Attrib: Attribute table

CRL: CRL table

csv: Output as Comma Separated Values

To display the StatusCode column for all entries: -out StatusCode

To display all columns for the last entry: -restrict "RequestId= \$"

To display RequestId and Disposition for three requests: -restrict "RequestId>=37,RequestId<40" -out "RequestId,Disposition"

To display Row Ids and CRL Numbers for all Base CRLs: -restrict "CRLMinBase=0" -out "CRLRowId,CRLNumber" CRL

To display Base CRL Number 3: -v -restrict "CRLMinBase=0,CRLNumber=3" -out "CRLRawCRL" CRL

To display the entire CRL table: CRL

Use "Date[+|-dd:hh]" for date restrictions

Use "now+dd:hh" for a date relative to the current time

[-silent] [-split] [-config Machine\CAName] [-restrict RestrictionList] [-out ColumnList]

Return to [Menu](#)

-db

CertUtil [Options] -db

Dump Raw Database

[-config Machine\CAName] [-restrict RestrictionList] [-out ColumnList]

Return to [Menu](#)

-deleterow

CertUtil [Options] -deleterow RowId | Date [Request | Cert | Ext | Attrib | CRL]

Delete server database row

Request: Failed and pending requests (submission date)

Cert: Expired and revoked certificates (expiration date)

Ext: Extension table

Attrib: Attribute table

CRL: CRL table (expiration date)

To delete failed and pending requests submitted by January 22, 2001: 1/22/2001 Request

To delete all certificates that expired by January 22, 2001: 1/22/2001 Cert

To delete the certificate row, attributes and extensions for RequestId 37: 37

To delete CRLs that expired by January 22, 2001: 1/22/2001 CRL

[-f] [-config Machine\CAName]

Return to [Menu](#)

-backup

CertUtil [Options] -backup BackupDirectory [Incremental] [KeepLog]

Backup Active Directory Certificate Services

BackupDirectory: directory to store backed up data

Incremental: perform incremental backup only (default is full backup)

KeepLog: preserve database log files (default is to truncate log files)

[-f] [-config Machine\CAName] [-p Password]

Return to [Menu](#)

-backupDB

CertUtil [Options] -backupDB BackupDirectory [Incremental] [KeepLog]

Backup Active Directory Certificate Services database

BackupDirectory: directory to store backed up database files

Incremental: perform incremental backup only (default is full backup)

KeepLog: preserve database log files (default is to truncate log files)

[-f] [-config Machine\CAName]

Return to [Menu](#)

-backupKey

CertUtil [Options] -backupKey BackupDirectory

Backup Active Directory Certificate Services certificate and private key

BackupDirectory: directory to store backed up PFX file

[-f] [-config Machine\CAName] [-p Password] [-t Timeout]

Return to [Menu](#)

-restore

CertUtil [Options] -restore BackupDirectory

Restore Active Directory Certificate Services

BackupDirectory: directory containing data to be restored

[-f] [-config Machine\CAName] [-p Password]

Return to [Menu](#)

-restoreDB

CertUtil [Options] -restoreDB BackupDirectory

Restore Active Directory Certificate Services database

BackupDirectory: directory containing database files to be restored

[-f] [-config Machine\CAName]

Return to [Menu](#)

-restoreKey

CertUtil [Options] -restoreKey BackupDirectory | PFXFile

Restore Active Directory Certificate Services certificate and private key

BackupDirectory: directory containing PFX file to be restored

PFXFile: PFX file to be restored

[-f] [-config Machine\CAName] [-p Password]

Return to [Menu](#)

-importPFX

CertUtil [Options] -importPFX [CertificateStoreName] PFXFile [Modifiers]

Import certificate and private key

CertificateStoreName: Certificate store name. See [-store](#).

PFXFile: PFX file to be imported

Modifiers: Comma separated list of one or more of the following:

1. AT_SIGNATURE: Change the KeySpec to Signature
2. AT_KEYEXCHANGE: Change the KeySpec to Key Exchange
3. NoExport: Make the private key non-exportable
4. NoCert: Do not import the certificate
5. NoChain: Do not import the certificate chain
6. NoRoot: Do not import the root certificate
7. Protect: Protect keys with password
8. NoProtect: Do not password protect keys

Defaults to personal machine store.

[-f] [-user] [-p Password] [-csp Provider]

Return to [Menu](#)

-dynamicfilelist

CertUtil [Options] -dynamicfilelist

Display dynamic file List

[-config Machine\CAName]

Return to [Menu](#)

-databaselocations

CertUtil [Options] -databaselocations

Display database locations

[-config Machine\CAName]

Return to [Menu](#)

-hashfile

CertUtil [Options] -hashfile InFile [HashAlgorithm]

Generate and display cryptographic hash over a file

Return to [Menu](#)

-store

CertUtil [Options] -store [CertificateStoreName [CertId [OutputFile]]]

Dump certificate store

CertificateStoreName: Certificate store name. Examples:

- "My", "CA" (default), "Root",
- "Ildap:///CN=Certification Authorities,CN=Public Key Services,CN=Services,CN=Configuration,DC=cpandl,DC=com?cACertificate?one?objectClass=certificationAuthority" (View Root Certificates)
- "Ildap:///CN=CAName,CN=Certification Authorities,CN=Public Key Services,CN=Services,CN=Configuration,DC=cpandl,DC=com?cACertificate?base?objectClass=certificationAuthority" (Modify Root Certificates)
- "Ildap:///CN=CAName,CN=MachineName,CN=CDP,CN=Public Key Services,CN=Services,CN=Configuration,DC=cpandl,DC=com?certificateRevocationList?base?objectClass=cRLDistributionPoint" (View CRLs)
- "Ildap:///CN=NTAuthCertificates,CN=Public Key Services,CN=Services,CN=Configuration,DC=cpandl,DC=com?cACertificate?base?objectClass=certificationAuthority" (Enterprise CA Certificates)
- ldap: (AD computer object certificates)
- -user ldap: (AD user object certificates)

CertId: Certificate or CRL match token. This can be a serial number, an SHA-1 certificate, CRL, CTL or public key hash, a numeric cert index (0, 1, and so on), a numeric CRL index (.0, .1, and so on), a numeric CTL index (..0, ..1, and so on), a public key, signature or extension ObjectId, a certificate subject Common Name, an e-mail address, UPN or DNS name, a key container name or CSP name, a template name or ObjectId, an EKU or Application Policies ObjectId, or a CRL issuer Common Name. Many of these may result in multiple matches.

OutputFile: file to save matching cert

Use -user to access a user store instead of a machine store.

Use -enterprise to access a machine enterprise store.

Use -service to access a machine service store.

Use -grouppolicy to access a machine group policy store.

Examples:

- -enterprise NTAUTH
- -enterprise Root 37
- -user My 26e0aaaf000000000004
- CA .11

[-f] [-enterprise] [-user] [-GroupPolicy] [-silent] [-split] [-dc DCName]

Return to [Menu](#)

-addstore

CertUtil [Options] -addstore CertificateStoreName InFile

Add certificate to store

CertificateStoreName: Certificate store name. See [-store](#).

InFile: Certificate or CRL file to add to store.

[-f] [-enterprise] [-user] [-GroupPolicy] [-dc DCName]

Return to [Menu](#)

-delstore

CertUtil [Options] -delstore CertificateStoreName CertId

Delete certificate from store

CertificateStoreName: Certificate store name. See [-store](#).

CertId: Certificate or CRL match token. See [-store](#).

[-enterprise] [-user] [-GroupPolicy] [-dc DCName]

Return to [Menu](#)

-verifystore

CertUtil [Options] -verifystore CertificateStoreName [CertId]

Verify certificate in store

CertificateStoreName: Certificate store name. See [-store](#).

CertId: Certificate or CRL match token. See [-store](#).

[-enterprise] [-user] [-GroupPolicy] [-silent] [-split] [-dc DCName] [-t Timeout]

Return to [Menu](#)

-repairstore

CertUtil [Options] -repairstore CertificateStoreName CertIdList [PropertyInFile | SDDLSecurityDescriptor]

Repair key association or update certificate properties or key security descriptor

CertificateStoreName: Certificate store name. See [-store](#).

CertIdList: comma separated list of Certificate or CRL match tokens. See [-store](#) CertId description.

PropertyInfFile -- INF file containing external properties:

```
[Properties]
19 = Empty ; Add archived property, OR:
19 =      ; Remove archived property

11 = "{text}Friendly Name" ; Add friendly name property

127 = "{hex}" ; Add custom hexadecimal property
    _continue_ = "00 01 02 03 04 05 06 07 08 09 0a 0b 0c 0d 0e 0f"
    _continue_ = "10 11 12 13 14 15 16 17 18 19 1a 1b 1c 1d 1e 1f"

2 = "{text}" ; Add Key Provider Information property
    _continue_ = "Container=Container Name&"
    _continue_ = "Provider=Microsoft Strong Cryptographic Provider&"
    _continue_ = "ProviderType=1&"
    _continue_ = "Flags=0&"
    _continue_ = "KeySpec=2"

9 = "{text}" ; Add Enhanced Key Usage property
    _continue_ = "1.3.6.1.5.5.7.3.2,"
    _continue_ = "1.3.6.1.5.5.7.3.1,"
```

[-f] [-enterprise] [-user] [-GroupPolicy] [-silent] [-split] [-csp Provider]

Return to [Menu](#)

-viewstore

CertUtil [Options] -viewstore [CertificateStoreName [CertId [OutputFile]]]

Dump certificate store

CertificateStoreName: Certificate store name. Examples:

- "My", "CA" (default), "Root",
- "Ildap:///CN=Certification Authorities,CN=Public Key Services,CN=Services,CN=Configuration,DC=cpandl,DC=com?cACertificate?one?objectClass=certificationAuthority" (View Root Certificates)
- "Ildap:///CN=CAName,CN=Certification Authorities,CN=Public Key Services,CN=Services,CN=Configuration,DC=cpandl,DC=com?cACertificate?base?objectClass=certificationAuthority" (Modify Root Certificates)
- "Ildap:///CN=CAName,CN=MachineName,CN=CDP,CN=Public Key Services,CN=Services,CN=Configuration,DC=cpandl,DC=com?certificateRevocationList?base?objectClass=cRLDistributionPoint" (View CRLs)
- "Ildap:///CN=NTAuthCertificates,CN=Public Key Services,CN=Services,CN=Configuration,DC=cpandl,DC=com?cACertificate?base?objectClass=certificationAuthority" (Enterprise CA Certificates)
- ldap: (AD machine object certificates)
- -user ldap: (AD user object certificates)

CertId: Certificate or CRL match token. This can be a serial number, an SHA-1 certificate, CRL, CTL or public key hash, a numeric cert index (0, 1, and so on), a numeric CRL index (.0, .1, and so on), a numeric CTL index (..0, ..1, and so on), a public key, signature or extension ObjectId, a certificate subject Common Name, an e-mail address, UPN or DNS name, a key container name or CSP name, a template name or ObjectId, an EKU or Application Policies ObjectId, or a CRL issuer Common Name. Many of these may result in multiple matches.

OutputFile: file to save matching cert

Use -user to access a user store instead of a machine store.

Use -enterprise to access a machine enterprise store.

Use -service to access a machine service store.

Use -grouppolicy to access a machine group policy store.

Examples:

1. -enterprise NTAUTH
2. -enterprise Root 37
3. -user My 26e0aaaf000000000004
4. CA .11

[-f] [-enterprise] [-user] [-GroupPolicy] [-dc DCName]

Return to [Menu](#)

-viewdelstore

CertUtil [Options] -viewdelstore [CertificateStoreName [CertId [OutputFile]]]

Delete certificate from store

CertificateStoreName: Certificate store name. Examples:

- "My", "CA" (default), "Root",
- "Ildap:///CN=Certification Authorities,CN=Public Key Services,CN=Services,CN=Configuration,DC=cpanidl,DC=com?cACertificate?one?objectClass=certificationAuthority" (View Root Certificates)
- "Ildap:///CN=CANAME,CN=Certification Authorities,CN=Public Key Services,CN=Services,CN=Configuration,DC=cpanidl,DC=com?cACertificate?base?objectClass=certificationAuthority" (Modify Root Certificates)
- "Ildap:///CN=CANAME,CN=MachineName,CN=CDP,CN=Public Key Services,CN=Services,CN=Configuration,DC=cpanidl,DC=com?certificateRevocationList?base?objectClass=cRLDistributionPoint" (View CRLs)
- "Ildap:///CN=NTAuthCertificates,CN=Public Key Services,CN=Services,CN=Configuration,DC=cpanidl,DC=com?cACertificate?base?objectClass=certificationAuthority" (Enterprise CA Certificates)
- ldap: (AD machine object certificates)
- -user ldap: (AD user object certificates)

CertId: Certificate or CRL match token. This can be a serial number, an SHA-1 certificate, CRL, CTL or public key hash, a numeric cert index (0, 1, and so on), a numeric CRL index (.0, .1, and so on), a numeric CTL index (..0, ..1, and so on), a public key, signature or extension ObjectId, a certificate subject Common Name, an e-mail address, UPN or DNS name, a key container name or CSP name, a template name or ObjectId, an EKU or Application Policies ObjectId, or a CRL issuer Common Name. Many of these may result in multiple matches.

OutputFile: file to save matching cert

Use -user to access a user store instead of a machine store.

Use -enterprise to access a machine enterprise store.

Use -service to access a machine service store.

Use -grouppolicy to access a machine group policy store.

Examples:

1. -enterprise NTAUTH
2. -enterprise Root 37
3. -user My 26e0aaaf000000000004
4. CA .11

[-f] [-enterprise] [-user] [-GroupPolicy] [-dc DCName]

Return to [Menu](#)

-dsPublish

CertUtil [Options] -dsPublish CertFile [NTAuthCA | RootCA | SubCA | CrossCA | KRA | User | Machine]

CertUtil [Options] -dsPublish CRLFile [DSCDPContainer [DSCDPCN]]

Publish certificate or CRL to Active Directory

CertFile: certificate file to publish

NTAuthCA: Publish cert to DS Enterprise store

RootCA: Publish cert to DS Trusted Root store

SubCA: Publish CA cert to DS CA object

CrossCA: Publish cross cert to DS CA object

KRA: Publish cert to DS Key Recovery Agent object

User: Publish cert to User DS object

Machine: Publish cert to Machine DS object

CRLFile: CRL file to publish

DSCDPContainer: DS CDP container CN, usually the CA machine name

DSCDPCN: DS CDP object CN, usually based on the sanitized CA short name and key index

Use -f to create DS object.

[-f] [-user] [-dc DCName]

Return to [Menu](#)

-ADTemplate

CertUtil [Options] -ADTemplate [Template]

Display AD templates

[-f] [-user] [-ut] [-mt] [-dc DCName]

-Template

CertUtil [Options] -Template [Template]

Display Enrollment Policy templates

[-f] [-user] [-silent] [-PolicyServer URLOrId] [-Anonymous] [-Kerberos] [-ClientCertificate ClientCertId] [-UserName UserName] [-p Password]

Return to [Menu](#)

-TemplateCAs

CertUtil [Options] -TemplateCAs Template

Display CAs for template

[-f] [-user] [-dc DCName]

Return to [Menu](#)

-CATemplates

CertUtil [Options] -CATemplates [Template]

Display templates for CA

[-f] [-user] [-ut] [-mt] [-config Machine\CAName] [-dc DCName]

Return to [Menu](#)

-SetCASites

CertUtil [Options] -SetCASites [set] [SiteName]

CertUtil [Options] -SetCASites verify [SiteName]

CertUtil [Options] -SetCASites delete

Set, Verify or Delete CA site names

- Use the -config option to target a single CA (Default is all CAs)
- *SiteName* is allowed only when targeting a single CA
- Use -f to override validation errors for the specified *SiteName*
- Use -f to delete all CA site names

[-f] [-config Machine\CAName] [-dc DCName]

NOTE

For more information on configuring CAs for Active Directory Domain Services (AD DS) site awareness, see [AD DS Site Awareness for AD CS and PKI clients](#).

Return to [Menu](#)

-enrollmentServerURL

CertUtil [Options] -enrollmentServerURL [URL AuthenticationType [Priority] [Modifiers]]

CertUtil [Options] -enrollmentServerURL URL delete

Display, add or delete enrollment server URLs associated with a CA

AuthenticationType: Specify one of the following client authentication methods while adding a URL

1. Kerberos: Use Kerberos SSL credentials
2. UserName: Use named account for SSL credentials
3. ClientCertificate: Use X.509 Certificate SSL credentials
4. Anonymous: Use anonymous SSL credentials

delete: deletes the specified URL associated with the CA

Priority: defaults to '1' if not specified when adding a URL

Modifiers -- Comma separated list of one or more of the following:

1. AllowRenewalsOnly: Only renewal requests can be submitted to this CA via this URL
2. AllowKeyBasedRenewal: Allows use of a certificate that has no associated account in the AD. This applies only with ClientCertificate and AllowRenewalsOnly Mode

[-config Machine\CAName] [-dc DCName]

Return to [Menu](#)

-ADCA

CertUtil [Options] -ADCA [CAName]

Display AD CAs

[-f] [-split] [-dc DCName]

Return to [Menu](#)

-CA

CertUtil [Options] -CA [CAName | TemplateName]

Display Enrollment Policy CAs

[-f] [-user] [-silent] [-split] [-PolicyServer URLOrId] [-Anonymous] [-Kerberos] [-ClientCertificate ClientCertId] [-UserName UserName] [-p Password]

Return to [Menu](#)

-Policy

Display Enrollment Policy

[-f] [-user] [-silent] [-split] [-PolicyServer URLOrId] [-Anonymous] [-Kerberos] [-ClientCertificate ClientCertId] [-UserName UserName] [-p Password]

Return to [Menu](#)

-PolicyCache

CertUtil [Options] -PolicyCache [delete]

Display or delete Enrollment Policy Cache entries

delete: delete Policy Server cache entries

-f: use -f to delete all cache entries

[-f] [-user] [-PolicyServer URLOrId]

Return to [Menu](#)

-CredStore

CertUtil [Options] -CredStore [URL]

CertUtil [Options] -CredStore URL add

CertUtil [Options] -CredStore URL delete

Display, add or delete Credential Store entries

URL: target URL. Use * to match all entries. Use https://machine* to match a URL prefix.

add: add a Credential Store entry. SSL credentials must also be specified.

delete: delete Credential Store entries

-f: use -f to overwrite an entry or to delete multiple entries.

[-f] [-user] [-silent] [-Anonymous] [-Kerberos] [-ClientCertificate ClientCertId] [-UserName UserName] [-p Password]

Return to [Menu](#)

-InstallDefaultTemplates

CertUtil [Options] -InstallDefaultTemplates

Install default certificate templates

[-dc DCName]

Return to [Menu](#)

-URLCache

CertUtil [Options] -URLCache [URL | CRL | * [delete]]

Display or delete URL cache entries

URL: cached URL

CRL: operate on all cached CRL URLs only

*: operate on all cached URLs

delete: delete relevant URLs from the current user's local cache

Use -f to force fetching a specific URL and updating the cache.

[-f] [-split]

Return to [Menu](#)

-pulse

CertUtil [Options] -pulse

Pulse autoenrollment events

[-user]

Return to [Menu](#)

-MachineInfo

CertUtil [Options] -MachineInfo DomainName\MachineName\$

Display Active Directory computer object information

Return to [Menu](#)

-DCInfo

CertUtil [Options] -DCInfo [Domain] [Verify | DeleteBad | DeleteAll]

Display domain controller information

Default is to display DC certs without verification

[-f] [-user] [-urlfetch] [-dc DCName] [-t Timeout]

TIP

The ability to specify an Active Directory Domain Services (AD DS) domain **[Domain]** and to specify a domain controller (**-dc**) was added in Windows Server 2012. To successfully run the command, you must use an account that is a member of **Domain Admins** or **Enterprise Admins**. The behavior modifications of this command are as follows:

- > 1. If a domain is not specified and a specific domain controller is not specified, this option returns a list of domain controllers to process from the default domain controller.
- > 2. If a domain is not specified, but a domain controller is specified, a report of the certificates on the specified domain controller is generated.
- > 3. If a domain is specified, but a domain controller is not specified, a list of domain controllers is generated along with reports on the certificates for each domain controller in the list.
- > 4. If the domain and domain controller are specified, a list of domain controllers is generated from the targeted domain controller. A report of the certificates for each domain controller in the list is also generated.

For example, assume there is a domain named CPANDL with a domain controller named CPANDL-DC1. You could run the following command to retrieve a list of domain controllers and their certificates that from CPANDL-DC1:

```
certutil -dc cpandl-dc1 -dcinfo cpandl
```

Return to [Menu](#)

-EntInfo

CertUtil [Options] -EntInfo DomainName\MachineName\$

[-f] [-user]

Return to [Menu](#)

-TCAInfo

CertUtil [Options] -TCAInfo [DomainDN | -]

Display CA information

[-f] [-enterprise] [-user] [-urlfetch] [-dc DCName] [-t Timeout]

Return to [Menu](#)

-SCInfo

CertUtil [Options] -SCInfo [ReaderName [CRYPT_DELETEKEYSET]]

Display smart card information

CRYPT_DELETEKEYSET: Delete all keys on the smart card

[-silent] [-split] [-urlfetch] [-t Timeout]

Return to [Menu](#)

-SCRoots

CertUtil [Options] -SCRoots update [+][InputRootFile] [ReaderName]

CertUtil [Options] -SCRoots save @OutputRootFile [ReaderName]

CertUtil [Options] -SCRoots view [InputRootFile | ReaderName]

CertUtil [Options] -SCRoots delete [ReaderName]

Manage smart card root certificates

[-f] [-split] [-p Password]

Return to [Menu](#)

-verifykeys

CertUtil [Options] -verifykeys [KeyContainerName CACertFile]

Verify public/private key set

KeyContainerName: key container name of the key to verify. Defaults to machine keys. Use -user for user keys.

CACertFile: signing or encryption certificate file

If no arguments are specified, each signing CA cert is verified against its private key.

This operation can only be performed against a local CA or local keys.

[-f] [-user] [-silent] [-config Machine\CAName]

Return to [Menu](#)

-verify

CertUtil [Options] -verify CertFile [ApplicationPolicyList | - [IssuancePolicyList]]

CertUtil [Options] -verify CertFile [CACertFile [CrossedCACertFile]]

CertUtil [Options] -verify CRLFile CACertFile [IssuedCertFile]

CertUtil [Options] -verify CRLFile CACertFile [DeltaCRLFile]

Verify certificate, CRL or chain

CertFile: Certificate to verify

ApplicationPolicyList: optional comma separated list of required Application Policy ObjectIds

IssuancePolicyList: optional comma separated list of required Issuance Policy ObjectIds

CACertFile: optional issuing CA certificate to verify against

CrossedCACertFile: optional certificate cross-certified by CertFile

CRLFile: CRL to verify

IssuedCertFile: optional issued certificate covered by CRLFile

DeltaCRLFile: optional delta CRL

If ApplicationPolicyList is specified, chain building is restricted to chains valid for the specified Application Policies.

If IssuancePolicyList is specified, chain building is restricted to chains valid for the specified Issuance Policies.

If CACertFile is specified, fields in CACertFile are verified against CertFile or CRLFile.

If CACertFile is not specified, CertFile is used to build and verify a full chain.

If CACertFile and CrossedCACertFile are both specified, fields in CACertFile and CrossedCACertFile are verified against CertFile.

If IssuedCertFile is specified, fields in IssuedCertFile are verified against CRLFile.

If DeltaCRLFile is specified, fields in DeltaCRLFile are verified against CRLFile.

[-f] [-enterprise] [-user] [-silent] [-split] [-urlfetch] [-t Timeout]

Return to [Menu](#)

-verifyCTL

CertUtil [Options] -verifyCTL CTLObject [CertDir] [CertFile]

Verify AuthRoot or Disallowed Certificates CTL

CTLObject: Identifies the CTL to verify:

- AuthRootWU: read AuthRoot CAB and matching certificates from the URL cache. Use -f to download from Windows Update instead.
- DisallowedWU: read Disallowed Certificates CAB and disallowed certificate store file from the URL cache. Use -f to download from Windows Update instead.
- AuthRoot: read registry cached AuthRoot CTL. Use with -f and a CertFile that is not already trusted to force updating the registry cached AuthRoot and Disallowed Certificate CTLs.
- Disallowed: read registry cached Disallowed Certificates CTL. -f has the same behavior as with AuthRoot.
- CTLFileName: file or http: path to CTL or CAB

CertDir: folder containing certificates matching CTL entries. An http: folder path must end with a path separator. If a folder is not specified with AuthRoot or Disallowed, multiple locations will be searched for matching certificates: local certificate stores, crypt32.dll resources and the local URL cache. Use -f to download from Windows Update when necessary. Otherwise defaults to the same folder or web site as the CTLObject.

CertFile: file containing certificate(s) to verify. Certificates will be matched against CTL entries, and match results displayed. Suppresses most of the default output.

[-f] [-user] [-split]

Return to [Menu](#)

-sign

CertUtil [Options] -sign InFileList[SerialNumber][CRL OutFileList [StartDate+dd:hh] [+SerialNumberList] -

SerialNumberList | -ObjectIdList | @ExtensionFile]

CertUtil [Options] -sign InFileList[SerialNumber]CRL OutFileList [#HashAlgorithm]
[+AlternateSignatureAlgorithm | -AlternateSignatureAlgorithm]

Re-sign CRL or certificate

InFileList: comma separated list of Certificate or CRL files to modify and re-sign

SerialNumber: Serial number of certificate to create. Validity period and other options must not be present.

CRL: Create an empty CRL. Validity period and other options must not be present.

OutFileList: comma separated list of modified Certificate or CRL output files. The number of files must match InFileList.

StartDate+dd:hh: new validity period: optional date plus; optional days and hours validity period; If both are specified, use a plus sign (+) separator. Use "now[+dd:hh]" to start at the current time. Use "never" to have no expiration date (for CRLs only).

SerialNumberList: comma separated serial number list to add or remove

ObjectIdList: comma separated extension ObjectId list to remove

@ExtensionFile: INF file containing extensions to update or remove:

```
[Extensions]
2.5.29.31 = ; Remove CRL Distribution Points extension
2.5.29.15 = "{hex}" ; Update Key Usage extension
_continue_="03 02 01 86"
```

HashAlgorithm: Name of the hash algorithm preceded by a # sign

AlternateSignatureAlgorithm: alternate Signature algorithm specifier

A minus sign causes serial numbers and extensions to be removed. A plus sign causes serial numbers to be added to a CRL. When removing items from a CRL, the list may contain both serial numbers and ObjectIds. A minus sign before AlternateSignatureAlgorithm causes the legacy signature format to be used. A plus sign before AlternateSignatureAlgorithm causes the alternate signature format to be used. If AlternateSignatureAlgorithm is not specified then the signature format in the certificate or CRL is used.

[-nullsign] [-f] [-silent] [-Cert CertId]

Return to [Menu](#)

-vroot

CertUtil [Options] -vroot [delete]

Create/delete web virtual roots and file shares

Return to [Menu](#)

-vocsproot

CertUtil [Options] -vocsproot [delete]

Create/delete web virtual roots for OCSP web proxy

Return to [Menu](#)

-addEnrollmentServer

CertUtil [Options] -addEnrollmentServer Kerberos | UserName | ClientCertificate [AllowRenewalsOnly]
[AllowKeyBasedRenewal]

Add an Enrollment Server application

Add an Enrollment Server application and application pool if necessary, for the specified CA. This command does not install binaries or packages. One of the following authentication methods with which the client connects to a Certificate Enrollment Server.

- Kerberos: Use Kerberos SSL credentials
- UserName: Use named account for SSL credentials
- ClientCertificate: Use X.509 Certificate SSL credentials
- AllowRenewalsOnly: Only renewal requests can be submitted to this CA via this URL
- AllowKeyBasedRenewal -- Allows use of a certificate that has no associated account in the AD. This applies only with ClientCertificate and AllowRenewalsOnly mode.

[-config Machine\CAName]

Return to [Menu](#)

-deleteEnrollmentServer

CertUtil [Options] -deleteEnrollmentServer Kerberos | UserName | ClientCertificate

Delete an Enrollment Server application

Delete an Enrollment Server application and application pool if necessary, for the specified CA. This command does not remove binaries or packages. One of the following authentication methods with which the client connects to a Certificate Enrollment Server.

1. Kerberos: Use Kerberos SSL credentials
2. UserName: Use named account for SSL credentials
3. ClientCertificate: Use X.509 Certificate SSL credentials

[-config Machine\CAName]

Return to [Menu](#)

-addPolicyServer

CertUtil [Options] -addPolicyServer Kerberos | UserName | ClientCertificate [KeyBasedRenewal]

Add a Policy Server application

Add a Policy Server application and application pool if necessary. This command does not install binaries or packages. One of the following authentication methods with which the client connects to a Certificate Policy Server:

- Kerberos: Use Kerberos SSL credentials
- UserName: Use named account for SSL credentials
- ClientCertificate: Use X.509 Certificate SSL credentials
- KeyBasedRenewal: Only policies that contain KeyBasedRenewal templates are returned to the client. This flag applies only for UserName and ClientCertificate authentication.

Return to [Menu](#)

-deletePolicyServer

CertUtil [Options] -deletePolicyServer Kerberos | UserName | ClientCertificate [KeyBasedRenewal]

Delete a Policy Server application

Delete a Policy Server application and application pool if necessary. This command does not remove binaries or packages. One of the following authentication methods with which the client connects to a Certificate Policy Server:

1. Kerberos: Use Kerberos SSL credentials
2. UserName: Use named account for SSL credentials
3. ClientCertificate: Use X.509 Certificate SSL credentials
4. KeyBasedRenewal: KeyBasedRenewal policy server

Return to [Menu](#)

-oid

CertUtil [Options] -oid ObjectId [DisplayName | delete [LanguageId [Type]]]

CertUtil [Options] -oid GroupId

CertUtil [Options] -oid AlgId | AlgorithmName [GroupId]

Display ObjectId or set display name

- ObjectId -- ObjectId to display or to add display name
- GroupId -- decimal GroupId number for ObjectIds to enumerate
- AlgId -- hexadecimal AlgId for ObjectId to look up
- AlgorithmName -- Algorithm Name for ObjectId to look up
- DisplayName -- Display Name to store in DS
- delete -- delete display name
- LanguageId -- Language Id (defaults to current: 1033)
- Type -- DS object type to create: 1 for Template (default), 2 for Issuance Policy, 3 for Application Policy
- Use -f to create DS object.

[-f]

Return to [Menu](#)

-error

CertUtil [Options] -error ErrorCode

Display error code message text

Return to [Menu](#)

-getreg

CertUtil [Options] -getreg [{ca|restore|policy|exit|template|enroll|chain|PolicyServers}][ProgId]
[RegistryValueName]

Display registry value

ca: Use CA's registry key

restore: Use CA's restore registry key

policy: Use policy module's registry key

exit: Use first exit module's registry key

template: Use template registry key (use -user for user templates)

enroll: Use enrollment registry key (use -user for user context)

chain: Use chain configuration registry key

PolicyServers: Use Policy Servers registry key

ProgId: Use policy or exit module's ProgId (registry subkey name)

RegistryValueName: registry value name (use "Name*" to prefix match)

Value: new numeric, string or date registry value or filename. If a numeric value starts with "+" or "-", the bits specified in the new value are set or cleared in the existing registry value.

If a string value starts with "+" or "-", and the existing value is a REG_MULTI_SZ value, the string is added to or removed from the existing registry value. To force creation of a REG_MULTI_SZ value, add a "\n" to the end of the string value.

If the value starts with "@", the rest of the value is the name of the file containing the hexadecimal text representation of a binary value. If it does not refer to a valid file, it is instead parsed as [Date][+|-][dd:hh] -- an optional date plus or minus optional days and hours. If both are specified, use a plus sign (+) or minus sign (-) separator. Use "now+dd:hh" for a date relative to the current time.

Use "chain\ChainCacheResyncFiletime @now" to effectively flush cached CRLs.

[-f] [-user] [-GroupPolicy] [-config Machine\CAName]

Return to [Menu](#)

-setreg

CertUtil [Options] -setreg [{ca|restore|policy|exit|template|enroll|chain|PolicyServers}[ProgId]]RegistryValueName Value

Set registry value

ca: Use CA's registry key

restore: Use CA's restore registry key

policy: Use policy module's registry key

exit: Use first exit module's registry key

template: Use template registry key (use -user for user templates)

enroll: Use enrollment registry key (use -user for user context)

chain: Use chain configuration registry key

PolicyServers: Use Policy Servers registry key

ProgId: Use policy or exit module's ProgId (registry subkey name)

RegistryValueName: registry value name (use "Name*" to prefix match)

Value: new numeric, string or date registry value or filename. If a numeric value starts with "+" or "-", the bits specified in the new value are set or cleared in the existing registry value.

If a string value starts with "+" or "-", and the existing value is a REG_MULTI_SZ value, the string is added to or removed from the existing registry value. To force creation of a REG_MULTI_SZ value, add a "\n" to the end of the string value.

If the value starts with "@", the rest of the value is the name of the file containing the hexadecimal text representation of a binary value. If it does not refer to a valid file, it is instead parsed as [Date][+|-][dd:hh] -- an optional date plus or minus optional days and hours. If both are specified, use a plus sign (+) or minus sign (-) separator. Use "now+dd:hh" for a date relative to the current time.

Use "chain\ChainCacheResyncFiletime @now" to effectively flush cached CRLs.

[-f] [-user] [-GroupPolicy] [-config Machine\CAName]

Return to [Menu](#)

-delreg

CertUtil [Options] -delreg [{ca|restore|policy|exit|template|enroll|chain|PolicyServers}[ProgId]]
[RegistryValueName]

Delete registry value

ca: Use CA's registry key

restore: Use CA's restore registry key

policy: Use policy module's registry key

exit: Use first exit module's registry key

template: Use template registry key (use -user for user templates)

enroll: Use enrollment registry key (use -user for user context)

chain: Use chain configuration registry key

PolicyServers: Use Policy Servers registry key

ProgId: Use policy or exit module's ProgId (registry subkey name)

RegistryValueName: registry value name (use "Name*" to prefix match)

Value: new numeric, string or date registry value or filename. If a numeric value starts with "+" or "-", the bits specified in the new value are set or cleared in the existing registry value.

If a string value starts with "+" or "-", and the existing value is a REG_MULTI_SZ value, the string is added to or removed from the existing registry value. To force creation of a REG_MULTI_SZ value, add a "\n" to the end of the string value.

If the value starts with "@", the rest of the value is the name of the file containing the hexadecimal text representation of a binary value. If it does not refer to a valid file, it is instead parsed as [Date][+|-][dd:hh] -- an optional date plus or minus optional days and hours. If both are specified, use a plus sign (+) or minus sign (-) separator. Use "now+dd:hh" for a date relative to the current time.

Use "chain\ChainCacheResyncFiletime @now" to effectively flush cached CRLs.

[-f] [-user] [-GroupPolicy] [-config Machine\CAName]

Return to [Menu](#)

-ImportKMS

CertUtil [Options] -ImportKMS UserKeyAndCertFile [CertId]

Import user keys and certificates into server database for key archival

UserKeyAndCertFile -- Data file containing user private keys and certificates to be archived. This can be any of the following:

- Exchange Key Management Server (KMS) export file
- PFX file

CertId: KMS export file decryption certificate match token. See [-store](#).

Use -f to import certificates not issued by the CA.

[-f] [-silent] [-split] [-config Machine\CAName] [-p Password] [-symkeyalg SymmetricKeyAlgorithm[,KeyLength]]

Return to [Menu](#)

-ImportCert

CertUtil [Options] -ImportCert Certfile [ExistingRow]

Import a certificate file into the database

Use ExistingRow to import the certificate in place of a pending request for the same key.

Use -f to import certificates not issued by the CA.

The CA may also need to be configured to support foreign certificate import: certutil -setreg ca\KRAFlags +KRAF_ENABLEFOREIGN

[-f] [-config Machine\CAName]

Return to [Menu](#)

-GetKey

CertUtil [Options] -GetKey SearchToken [RecoveryBlobOutFile]

CertUtil [Options] -GetKey SearchToken script OutputScriptFile

CertUtil [Options] -GetKey SearchToken retrieve | recover OutputFileName

Retrieve archived private key recovery blob, generate a recovery script, or recover archived keys

script: generate a script to retrieve and recover keys (default behavior if multiple matching recovery candidates are found, or if the output file is not specified).

retrieve: retrieve one or more Key Recovery Blobs (default behavior if exactly one matching recovery candidate is found, and if the output file is specified)

recover: retrieve and recover private keys in one step (requires Key Recovery Agent certificates and private keys)

SearchToken: Used to select the keys and certificates to be recovered.

Can be any of the following:

1. Certificate Common Name
2. Certificate Serial Number
3. Certificate SHA-1 hash (thumbprint)
4. Certificate KeyId SHA-1 hash (Subject Key Identifier)

5. Requester Name (domain\user)

6. UPN (user@domain)

RecoveryBlobOutFile: output file containing a certificate chain and an associated private key, still encrypted to one or more Key Recovery Agent certificates.

OutputScriptFile: output file containing a batch script to retrieve and recover private keys.

OutputFileBaseName: output file base name. For retrieve, any extension is truncated and a certificate-specific string and the .rec extension are appended for each key recovery blob. Each file contains a certificate chain and an associated private key, still encrypted to one or more Key Recovery Agent certificates. For recover, any extension is truncated and the .p12 extension is appended. Contains the recovered certificate chains and associated private keys, stored as a PFX file.

[-f] [-UnicodeText] [-silent] [-config Machine\CAName] [-p Password] [-ProtectTo SAMNameAndSIDList] [-csp Provider]

Return to [Menu](#)

-RecoverKey

CertUtil [Options] -RecoverKey RecoveryBlobInFile [PFXOutFile [RecipientIndex]]

Recover archived private key

[-f] [-user] [-silent] [-split] [-p Password] [-ProtectTo SAMNameAndSIDList] [-csp Provider] [-t Timeout]

Return to [Menu](#)

-MergePFX

CertUtil [Options] -MergePFX PFXInFileList PFXOutFile [ExtendedProperties]

PFXInFileList: Comma separated PFX input file list

PFXOutFile: PFX output file

ExtendedProperties: Include extended properties

The password specified on the command line is a comma separated password list. If more than one password is specified, the last password is used for the output file. If only one password is provided or if the last password is "", the user will be prompted for the output file password.

[-f] [-user] [-split] [-p Password] [-ProtectTo SAMNameAndSIDList] [-csp Provider]

Return to [Menu](#)

-ConvertEPF

CertUtil [Options] -ConvertEPF PFXInFileList EPFOutFile [cast | cast-] [V3CACertId][,Salt]

Convert PFX files to EPF file

PFXInFileList: Comma separated PFX input file list

EPF: EPF output file

cast: Use CAST 64 encryption

cast-: Use CAST 64 encryption (export)

V3CACertId: V3 CA Certificate match token. See [-store](#) CertId description.

Salt: EPF output file salt string

The password specified on the command line is a comma separated password list. If more than one password is specified, the last password is used for the output file. If only one password is provided or if the last password is "", the user will be prompted for the output file password.

[-f] [-silent] [-split] [-dc DCName] [-p Password] [-csp Provider]

Return to [Menu](#)

Options

This section defines the options that you can specify with the command.

| OPTIONS | DESCRIPTION |
|------------------------|---|
| -nullsign | Use hash of data as signature |
| -f | Force overwrite |
| -enterprise | Use local machine Enterprise registry certificate store |
| -user | Use HKEY_CURRENT_USER keys or certificate store |
| -GroupPolicy | Use Group Policy certificate store |
| -ut | Display user templates |
| -mt | Display machine templates |
| -Unicode | Write redirected output in Unicode |
| -UnicodeText | Write output file in Unicode |
| -gmt | Display times as GMT |
| -seconds | Display times with seconds and milliseconds |
| -silent | Use silent flag to acquire crypt context |
| -split | Split embedded ASN.1 elements, and save to files |
| -v | Verbose operation |
| -privatekey | Display password and private key data |
| -pin PIN | Smart Card PIN |
| -urlfetch | Retrieve and verify AIA Certs and CDP CRLs |
| -config Machine\CAName | CA and computer name string |

| OPTIONS | DESCRIPTION |
|--|--|
| -PolicyServer URLOrId | Policy Server URL or Id. For selection U/I, use -PolicyServer. For all Policy Servers, use -PolicyServer * |
| -Anonymous | Use anonymous SSL credentials |
| -Kerberos | Use Kerberos SSL credentials |
| -ClientCertificate ClientCertId | Use X.509 Certificate SSL credentials. For selection U/I, use -clientCertificate. |
| -UserName UserName | Use named account for SSL credentials. For selection U/I, use -UserName. |
| -Cert CertId | Signing certificate |
| -dc DCName | Target a specific Domain Controller |
| -restrict RestrictionList | Comma separated Restriction List. Each restriction consists of a column name, a relational operator and a constant integer, string or date. One column name may be preceded by a plus or minus sign to indicate the sort order. Examples: "RequestId = 47" "+RequesterName >= a, RequesterName < b" "-RequesterName > DOMAIN, Disposition = 21" |
| -out ColumnList | Comma separated Column List |
| -p Password | Password |
| -ProtectTo SAMNameAndSIDList | Comma separated SAM Name/SID List |
| -csp Provider | Provider |
| -t Timeout | URL fetch timeout in milliseconds |
| -symkeyalg SymmetricKeyAlgorithm[,KeyLength] | Name of Symmetric Key Algorithm with optional key length, example: AES,128 or 3DES |

Return to [Menu](#)

Additional certutil examples

For some examples of how to use this command, see

1. [Certutil Examples for Managing Active Directory Certificate Services \(AD CS\) from the Command Line](#)
2. [Certutil tasks for managing certificates](#)
3. [Binary Request Export Using the CertUtil.exe Command-Line Tool Walkthrough](#)
4. [Root CA certificate renewal](#)
5. [Certutil](#)

Return to [Menu](#)

change

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

changes Remote Desktop Session Host (rd Session Host) server settings for logons, COM port mappings, and install mode.

NOTE

In Windows Server 2008 R2, Terminal Services was renamed Remote Desktop Services. To find out what's new in the latest version, see [What's New in Remote Desktop Services in Windows Server 2012](#) in the Windows Server TechNet Library.

Syntax

```
change logon
change port
change user
```

Parameters

| PARAMETER | DESCRIPTION |
|------------------------------|---|
| change logon | Enables or disables logons from client sessions on an rd Session Host server, or displays current logon status. |
| change port | lists or changes the COM port mappings to be compatible with MS-DOS applications. |
| change user | changes the install mode for the rd Session Host server. |

additional references

[Command-Line Syntax Key Remote Desktop Services \(Terminal Services\) Command Reference](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

change logon

Enables or disables logons from client sessions, or displays current logon status. This utility is useful for system maintenance. for examples of how to use this command, see [Examples](#).

NOTE

In Windows Server 2008 R2, Terminal Services was renamed Remote Desktop Services. To find out what's new in the latest version, see [What's New in Remote Desktop Services in Windows Server 2012](#) in the Windows Server TechNet Library.

Syntax

```
change logon {/query | /enable | /disable | /drain | /drainuntilrestart}
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------------|--|
| /query | Displays the current logon status, whether enabled or disabled. |
| /enable | Enables logons from client sessions, but not from the console. |
| /disable | Disables subsequent logons from client sessions, but not from the console. Does not affect currently logged on users. |
| /drain | Disables logons from new client sessions, but allows reconnections to existing sessions. |
| /drainuntilrestart | Disables logons from new client sessions until the computer is restarted, but allows reconnections to existing sessions. |
| /? | Displays help at the command prompt. |

Remarks

- Only administrators can use the **change logon** command.
- Logons are re-enabled when you restart the system. If you are connected to the Remote Desktop Session Host (rd Session Host) server from a client session and disable logons, and then log off before re-enabling logons, you will not be able to reconnect to your session. To re-enable logons from client sessions, log on at the console. ## Examples
- To display the current logon status, type: `change logon /query`
- To enable logons from client sessions, type: `change logon /enable`
- To disable client logons, type: `change logon /disable` ##### additional references [Command-Line Syntax Key change Remote Desktop Services \(Terminal Services\) Command Reference](#)

change port

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

lists or changes the COM port mappings to be compatible with MS-DOS applications. for examples of how to use this command, see [Examples](#).

NOTE

In Windows Server 2008 R2, Terminal Services was renamed Remote Desktop Services. To find out what's new in the latest version, see [What's New in Remote Desktop Services in Windows Server 2012](#) in the Windows Server TechNet Library.

Syntax

```
change port [<PortX>=<PortY> | /d <PortX> | /query]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------------|
| = | Maps COM <PortX> to <PortY>. |
| /d | deletes the mapping for COM <PortX>. |
| /query | Displays the current port mappings. |
| /? | Displays help at the command prompt. |

Remarks

- Most MS-DOS applications support only COM1 through COM4 serial ports. The **change port** command maps a serial port to a different port number, allowing applications that do not support high-numbered COM ports to access the serial port. remapping works only for the current session and is not retained if you log off from a session and then log on again.
- Use **change port** without any parameters to display the available COM ports and their current mappings. ## Examples
- To map COM12 to COM1 for use by an MS-DOS-based application, type: `change port com12=com1`
- To display the current port mappings, type: `change port /query` ##### additional references [Command-Line Syntax Key change Remote Desktop Services \(Terminal Services\) Command Reference](#)

change user

7/11/2018 • 3 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

changes the install mode for the Remote Desktop Session Host (rd Session Host) server. for examples of how to use this command, see [Examples](#).

NOTE

In Windows Server 2008 R2, Terminal Services was renamed Remote Desktop Services. To find out what's new in the latest version, see [What's New in Remote Desktop Services in Windows Server 2012](#) in the Windows Server TechNet Library.

Syntax

```
change user {/execute | /install | /query}
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| /execute | Enables .ini file mapping to the home directory. This is the default setting. |
| /install | Disables .ini file mapping to the home directory. All .ini files are read and written to the system directory. You must disable .ini file mapping when installing applications on an rd Session Host server. |
| /query | Displays the current setting for .ini file mapping. |
| /? | Displays help at the command prompt. |

Remarks

- Use **change user /install** before installing an application to create .ini files for the application in the system directory. These files are used as the source when user-specific .ini files are created. After installing the application, use **change user /execute** to revert to standard .ini file mapping.
- The first time that you run the application, it searches the home directory for its .ini files. If the .ini files are not found in the home directory, but are found in the system directory, Remote Desktop Services copies the .ini files to the home directory, ensuring that each user has a unique copy of the application .ini files. Any new .ini files are created in the home directory.
- Each user should have a unique copy of the .ini files for an application. This prevents instances where different users might have incompatible application configurations (for example, different default directories or screen resolutions).
- When the system is in install mode (that is, **change user /install**), several things occur. All registry entries that are created are shadowed under **HKEY_LOCAL_MACHINE\SOFTWARE\Microsoft\Windows NT\Currentversion\Terminal Server\Install**, in either the **\SOFTWARE** subkey or the **\MACHINE** subkey. Subkeys added to **HKEY_CURRENT_USER** are copied under the **\SOFTWARE** subkey, and subkeys added to **HKEY_LOCAL_MACHINE** are copied under the **\MACHINE** subkey. If the application queries the Windows directory by using system calls, such as `GetWindowsDirectory`, the rd Session Host server returns the systemroot directory. If any .ini file entries are added by using system calls, such as `WritePrivateProfileString`, they are added to the .ini files under the

systemroot directory.

- When the system returns to execution mode (that is, **change user /execute**), and the application tries to read a registry entry under **HKEY_CURRENT_USER** that does not exist, Remote Desktop Services checks to see whether a copy of the key exists under the **\Terminal Server\Install** subkey. If it does, the subkeys are copied to the appropriate location under **HKEY_CURRENT_USER**. If the application tries to read from an .ini file that does not exist, Remote Desktop Services searches for that .ini file under the system root. If the .ini file is in the system root, it is copied to the \Windows subdirectory of the user's home directory. If the application queries the Windows directory, the rd Session Host server returns the \Windows subdirectory of the user's home directory.
- When you log on, Remote Desktop Services checks whether its system .ini files are newer than the .ini files on your computer. If the system version is newer, your .ini file is either replaced or merged with the newer version. This depends on whether or not the INISYNC bit, 0x40, is set for this .ini file. Your previous version of the .ini file is renamed as Inifile.ctx. If the system registry values under the **\Terminal Server\Install** subkey are newer than your version under **HKEY_CURRENT_USER**, your version of the subkeys is deleted and replaced with the new subkeys from **\Terminal Server\Install**. ## Examples
- To disable .ini file mapping in the home directory, type: `change user /install`
- To enable .ini file mapping in the home directory, type: `change user /execute`
- To display the current setting for .ini file mapping, type: `change user /query` ##### additional references [Command-Line Syntax Key change Remote Desktop Services \(Terminal Services\) Command Reference](#)

Changes the active console code page. If used without parameters, **chcp** displays the number of the active console code page.

For examples of how to use this command, see [Examples](#).

Syntax

```
chcp [<NNN>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------------|
| <NNN> | Specifies the code page. |
| /? | Displays help at the command prompt. |

The following table lists each supported code page and its country/region or language:

| CODE PAGE | COUNTRY/REGION OR LANGUAGE |
|-----------|----------------------------|
| 437 | United States |
| 850 | Multilingual (Latin I) |
| 852 | Slavic (Latin II) |
| 855 | Cyrillic (Russian) |
| 857 | Turkish |
| 860 | Portuguese |
| 861 | Icelandic |
| 863 | Canadian-French |
| 865 | Nordic |
| 866 | Russian |
| 869 | Modern Greek |
| 936 | Chinese |

Remarks

- Only the original equipment manufacturer (OEM) code page that is installed with Windows appears correctly in a Command Prompt window that uses Raster fonts. Other code pages appear correctly in full-screen mode or in Command Prompt windows that use TrueType fonts.
- You do not need to prepare code pages (as in MS-DOS).
- Programs that you start after you assign a new code page use the new code page. However, programs (except Cmd.exe) that you start before you assign the new code page use the original code page.

Examples

To view the active code page setting, type:

```
chcp
```

A message similar to the following appears:

```
Active code page: 437
```

To change the active code page to 850 (Multilingual), type:

```
chcp 850
```

If the specified code page is invalid, the following error message appears:

```
Invalid code page
```

Additional references

[Command-Line Syntax Key](#)

chdir

4/13/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

This command is the same as the **cd** command.
See [cd](#) for syntax and parameters.

additional references

- [Command-Line Syntax Key](#)

chglogon

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Enables or disables logons from client sessions on an rd Session Host server, or displays current logon status.

NOTE

In Windows Server 2008 R2, Terminal Services was renamed Remote Desktop Services. To find out what's new in the latest version, see [What's New in Remote Desktop Services in Windows Server 2012](#) in the Windows Server TechNet Library.

Remarks

This command is the same as the **change logon** command.

additional references

[change logon Command-Line Syntax Key Remote Desktop Services \(Terminal Services\) Command Reference](#)

chgport

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

lists or changes the COM port mappings to be compatible with MS-DOS applications.

NOTE

In Windows Server 2008 R2, Terminal Services was renamed Remote Desktop Services. To find out what's new in the latest version, see [What's New in Remote Desktop Services in Windows Server 2012](#) in the Windows Server TechNet Library.

Remarks

This command is the same as the **change port** command.

additional references

[change port Command-Line Syntax Key Remote Desktop Services \(Terminal Services\) Command Reference](#)

chgusr

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

changes the install mode for the Remote Desktop Session Host (rd Session Host) server.

NOTE

In Windows Server 2008 R2, Terminal Services was renamed Remote Desktop Services. To find out what's new in the latest version, see [What's New in Remote Desktop Services in Windows Server 2012](#) in the Windows Server TechNet Library.

Remarks

This command is the same as the **change user** command.

additional references

[change user](#)

[Command-Line Syntax Key](#)

[Remote Desktop Services \(Terminal Services\) Command Reference](#)

chkdsk

4/13/2018 • 6 minutes to read • [Edit Online](#)

Checks the file system and file system metadata of a volume for logical and physical errors. If used without parameters, **chkdsk** displays only the status of the volume and does not fix any errors. If used with the **/f**, **/r**, **/x**, or **/b** parameters, it fixes errors on the volume.

IMPORTANT

Membership in the local **Administrators** group, or equivalent, is the minimum required to run **chkdsk**. To open a command prompt window as an administrator, right-click **Command prompt** in the Start menu, and then click **Run as administrator**.

IMPORTANT

Interrupting **chkdsk** is not recommended. However, canceling or interrupting **chkdsk** should not leave the volume any more corrupt than it was before **chkdsk** was run. Rerunning **chkdsk** checks and repairs any remaining corruption on the volume.

For examples of how to use this command, see [Examples](#).

Syntax

```
chkdsk [<Volume>[<Path><FileName>]] [/f] [/v] [/r] [/x] [/i] [/c] [/l[:<Size>]] [/b]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| <Volume> | Specifies the drive letter (followed by a colon), mount point, or volume name. |
| [<Path>] | Use with file allocation table (FAT) and FAT32 only. Specifies the location and name of a file or set of files that you want chkdsk to check for fragmentation. You can use the ? and * wildcard characters to specify multiple files. |
| /f | Fixes errors on the disk. The disk must be locked. If chkdsk cannot lock the drive, a message appears that asks you if you want to check the drive the next time you restart the computer. |
| /v | Displays the name of each file in every directory as the disk is checked. |
| /r | Locates bad sectors and recovers readable information. The disk must be locked. /r includes the functionality of /f , with the additional analysis of physical disk errors. |

| PARAMETER | DESCRIPTION |
|-------------|--|
| /x | Forces the volume to dismount first, if necessary. All open handles to the drive are invalidated. /x also includes the functionality of /f . |
| /i | Use with NTFS only. Performs a less vigorous check of index entries, which reduces the amount of time required to run chkdsk . |
| /c | Use with NTFS only. Does not check cycles within the folder structure, which reduces the amount of time required to run chkdsk . |
| /l[:<Size>] | Use with NTFS only. Changes the log file size to the size you type. If you omit the size parameter, /l displays the current size. |
| /b | NTFS only: Clears the list of bad clusters on the volume and rescans all allocated and free clusters for errors. /b includes the functionality of /r . Use this parameter after imaging a volume to a new hard disk drive. |
| /? | Displays help at the command prompt. |

Remarks

- Skipping volume checks

The **/i** or **/c** switch reduces the amount of time required to run **chkdsk** by skipping certain volume checks.

- Checking a locked drive at restart

If you want **chkdsk** to correct disk errors, you cannot have open files on the drive. If files are open, the following error message appears:

```
Chkdsk cannot run because the volume is in use by another process. Would you like to schedule this volume to be checked the next time the system restarts? (Y/N)
```

If you choose to check the drive the next time you restart the computer, **chkdsk** checks the drive and corrects errors automatically when you restart the computer. If the drive partition is a boot partition, **chkdsk** automatically restarts the computer after it checks the drive.

You can also use the **chkntfs /c** command to schedule the volume to be checked the next time the computer is restarted. Use the **fsutil dirty set** command to set the volume's dirty bit (indicating corruption), so that Windows runs **chkdsk** when the computer is restarted.

- Reporting disk errors

You should use **chkdsk** occasionally on FAT and NTFS file systems to check for disk errors. **Chkdsk** examines disk space and disk use and provides a status report specific to each file system. The status report shows errors found in the file system. If you run **chkdsk** without the **/f** parameter on an active partition, it might report spurious errors because it cannot lock the drive.

- Fixing logical disk errors

Chkdsk corrects logical disk errors only if you specify the **/f** parameter. **Chkdsk** must be able to lock the

drive to correct errors.

Because repairs on FAT file systems usually change a disk's file allocation table and sometimes cause a loss of data, **chkdsk** might display a confirmation message similar to the following:

```
10 lost allocation units found in 3 chains.  
Convert lost chains to files?
```

If you press **Y**, Windows saves each lost chain in the root directory as a file with a name in the format File<nnnn>.chk. When **chkdsk** finishes, you can check these files to see if they contain any data you need. If you press **N**, Windows fixes the disk, but it does not save the contents of the lost allocation units.

If you do not use the **/f** parameter, **chkdsk** displays a message that the file needs to be fixed, but it does not fix any errors.

If you use **chkdsk /f** on a very large disk or a disk with a very large number of files (for example, millions of files), **chkdsk /f** might take a long time to complete.

- Finding physical disk errors

Use the **/r** parameter to find physical disk errors in the file system and attempt to recover data from any affected disk sectors.

- Using **chkdsk** with open files

If you specify the **/f** parameter, **chkdsk** displays an error message if there are open files on the disk. If you do not specify the **/f** parameter and open files exist, **chkdsk** might report lost allocation units on the disk. This could happen if open files have not yet been recorded in the file allocation table. If **chkdsk** reports the loss of a large number of allocation units, consider repairing the disk.

- Using **chkdsk** with Shadow Copies for Shared Folders

Because the Shadow Copies for Shared Folders source volume cannot be locked while Shadow Copies for Shared Folders is enabled, running **chkdsk** against the source volume might report false errors or cause **chkdsk** to unexpectedly quit. You can, however, check shadow copies for errors by running **chkdsk** in Read-only mode (without parameters) to check the Shadow Copies for Shared Folders storage volume.

- Understanding exit codes

The following table lists the exit codes that **chkdsk** reports after it has finished.

| EXIT CODE | DESCRIPTION |
|-----------|---|
| 0 | No errors were found. |
| 1 | Errors were found and fixed. |
| 2 | Performed disk cleanup (such as garbage collection) or did not perform cleanup because /f was not specified. |
| 3 | Could not check the disk, errors could not be fixed, or errors were not fixed because /f was not specified. |

- The **chkdsk** command, with different parameters, is available from the Recovery Console.
- On servers that are infrequently restarted, you may want to use the **chkntfs** or the **fsutil dirty query** commands to determine whether the volume's dirty bit is already set before running **chkdsk**.

Examples

If you want to check the disk in drive D and have Windows fix errors, type:

```
chkdsk d: /f
```

If it encounters errors, **chkdsk** pauses and displays messages. **Chkdsk** finishes by displaying a report that lists the status of the disk. You cannot open any files on the specified drive until **chkdsk** finishes.

To check all files on a FAT disk in the current directory for noncontiguous blocks, type:

```
chkdsk *.*
```

Chkdsk displays a status report, and then lists the files that match the file specifications that have noncontiguous blocks.

Additional references

[Command-Line Syntax Key](#)

chkntfs

4/13/2018 • 2 minutes to read • [Edit Online](#)

Displays or modifies automatic disk checking when the computer is started. If used without options, **chkntfs** displays the file system of the specified volume. If automatic file checking is scheduled to run, **chkntfs** displays whether the specified volume is dirty or is scheduled to be checked the next time the computer is started.

NOTE

To run **chkntfs**, you must be a member of the Administrators group.

For examples of how to use this command, see [Examples](#).

Syntax

```
chkntfs <Volume> [...]  
chkntfs [/d]  
chkntfs [/t[:<Time>]]  
chkntfs [/x <Volume> [...]]  
chkntfs [/c <Volume> [...]]
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------------|--|
| <Volume> [...] | Specifies one or more volumes to check when the computer starts. Valid volumes include drive letters (followed by a colon), mount points, or volume names. |
| /d | Restores all chkntfs default settings, except the countdown time for automatic file checking. By default, all volumes are checked when the computer is started, and chkdsk runs on those that are dirty. |
| /t [:<Time>] | Changes the Autochk.exe initiation countdown time to the amount of time specified in seconds. If you do not enter a time, /t displays the current countdown time. |
| /x <Volume> [...] | Specifies one or more volumes to exclude from checking when the computer is started, even if the volume is marked as requiring chkdsk . |
| /c <Volume> [...] | Schedules one or more volumes to be checked when the computer is started, and runs chkdsk on those that are dirty. |
| /? | Displays help at the command prompt. |

Examples

To display the type of file system for drive C, type:

```
chkntfs c:
```

The following output indicates an NTFS file system:

```
The type of the file system is NTFS.
```

NOTE

If automatic file checking is scheduled to run, additional output will display, indicating whether the drive is dirty or has been manually scheduled to be checked the next time the computer is started.

To display the Autochk.exe initiation countdown time, type:

```
chkntfs /t
```

For example, if the countdown time is set to 10 seconds, the following output displays:

```
The AUTOCHK initiation countdown time is set to 10 second(s).
```

To change the Autochk.exe initiation countdown time to 30 seconds, type:

```
chkntfs /t:30
```

NOTE

Although you can set the Autochk.exe initiation countdown time to zero, doing so will prevent you from canceling a potentially time-consuming automatic file check.

The **/x** command-line option is not accumulative. If you type it more than once, the most recent entry overrides the previous entry. To exclude multiple volumes from being checked, you must list each of them in a single command. For example, to exclude both the D and E volumes, type:

```
chkntfs /x d: e:
```

The **/c** command-line option is accumulative. If you type **/c** more than once, each entry remains. To ensure that only a particular volume is checked, reset the defaults to clear all previous commands, exclude all volumes from being checked, and then schedule automatic file checking on the desired volume.

For example, to schedule automatic file checking on the D volume but not the C or E volumes, type the following commands in order:

```
chkntfs /d  
chkntfs /x c: d: e:  
chkntfs /c d:
```

Additional references

[Command-Line Syntax Key](#)

choice

4/13/2018 • 2 minutes to read • [Edit Online](#)

Prompts the user to select one item from a list of single-character choices in a batch program, and then returns the index of the selected choice. If used without parameters, **choice** displays the default choices **Y** and **N**.

For examples of how to use this command, see [Examples](#).

Syntax

```
choice [/c [<Choice1><Choice2><...>]] [/n] [/cs] [/t <Timeout> /d <Choice>] [/m <"Text">]
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------------|--|
| /c <Choice1> <...> | Specifies the list of choices to be created. Valid choices include a-z, A-Z, 0-9, and extended ASCII characters (128-254). The default list is "YN", which is displayed as <code>[Y,N]?</code> . |
| /n | Hides the list of choices, although the choices are still enabled and the message text (if specified by /m) is still displayed. |
| /cs | Specifies that the choices are case-sensitive. By default, the choices are not case-sensitive. |
| /t <Timeout> | Specifies the number of seconds to pause before using the default choice specified by /d . Acceptable values are from 0 to 9999 . If /t is set to 0 , choice does not pause before returning the default choice. |
| /d <Choice> | Specifies the default choice to use after waiting the number of seconds specified by /t . The default choice must be in the list of choices specified by /c . |
| /m <"Text"> | Specifies a message to display before the list of choices. If /m is not specified, only the choice prompt is displayed. |
| /? | Displays help at the command prompt. |

Remarks

- The ERRORLEVEL environment variable is set to the index of the key that the user selects from the list of choices. The first choice in the list returns a value of 1, the second a value of 2, and so on. If the user presses a key that is not a valid choice, **choice** sounds a warning beep. If **choice** detects an error condition, it returns an ERRORLEVEL value of 255. If the user presses CTRL+BREAK or CTRL+C, **choice** returns an ERRORLEVEL value of 0.

NOTE

When you use ERRORLEVEL values in a batch program, list them in decreasing order.

Examples

To present the choices Y, N, and C, type the following line in a batch file:

```
choice /c ync
```

The following prompt appears when the batch file runs the **choice** command:

```
[Y,N,C]?
```

To hide the choices Y, N, and C, but display the text "Yes, No, or Continue", type the following line in a batch file:

```
choice /c ync /n /m "Yes, No, or Continue?"
```

The following prompt appears when the batch file runs the **choice** command:

```
Yes, No, or Continue?
```

NOTE

If you use the **/n** parameter, but do not use **/m**, the user is not prompted when **choice** is waiting for input.

To show both the text and the options used in the previous examples, type the following line in a batch file:

```
choice /c ync /m "Yes, No, or Continue"
```

The following prompt appears when the batch file runs the **choice** command:

```
Yes, No, or Continue [Y,N,C]?
```

To set a time limit of five seconds and specify **N** as the default value, type the following line in a batch file:

```
choice /c ync /t 5 /d n
```

The following prompt appears when the batch file runs the **choice** command:

```
[Y,N,C]?
```

NOTE

In this example, if the user does not press a key within five seconds, **choice** selects **N** by default and returns an error value of 2. Otherwise, **choice** returns the value corresponding to the user's choice.

Additional references

[Command-Line Syntax Key](#)

cipher

4/13/2018 • 3 minutes to read • [Edit Online](#)

Displays or alters the encryption of directories and files on NTFS volumes. If used without parameters, **cipher** displays the encryption state of the current directory and any files it contains.

For examples of how to use this command, see [Examples](#).

Syntax

```
cipher [/e | /d | /c] [/s:<Directory>] [/b] [/h] [PathName [...]]
cipher /k
cipher /r:<FileName> [/smartcard]
cipher /u [/n]
cipher /w:<Directory>
cipher /x[:efsfile] [FileName]
cipher /y
cipher /adduser [/certhash:<Hash> | /certfile:<FileName>] [/s:<Directory>] [/b] [/h] [PathName [...]]
cipher /removeuser /certhash:<Hash> [/s:<Directory>] [/b] [/h] [<PathName> [...]]
cipher /rekey [PathName [...]]
```

Parameters

| PARAMETERS | DESCRIPTION |
|----------------------------|--|
| /b | Aborts if an error is encountered. By default, cipher continues to run even if errors are encountered. |
| /c | Displays information on the encrypted file. |
| /d | Decrypts the specified files or directories. |
| /e | Encrypts the specified files or directories. Directories are marked so that files that are added afterward will be encrypted. |
| /h | Displays files with hidden or system attributes. By default, these files are not encrypted or decrypted. |
| /k | Creates a new certificate and key for use with Encrypting File System (EFS) files. If the /k parameter is specified, all other parameters are ignored. |
| /r:<FileName> [/smartcard] | Generates an EFS recovery agent key and certificate, then writes them to a .pfx file (containing certificate and private key) and a .cer file (containing only the certificate). If /smartcard is specified, it writes the recovery key and certificate to a smart card, and no .pfx file is generated. |
| /s:<Directory> | Performs the specified operation on all subdirectories in the specified <i>Directory</i> . |

| PARAMETERS | DESCRIPTION |
|------------------------------|--|
| /u [/n] | Finds all encrypted files on the local drive(s). If used with the /n parameter, no updates are made. If used without /n , /u compares the user's file encryption key or the recovery agent's key to the current ones, and updates them if they have changed. This parameter works only with /n . |
| /w:<Directory> | Removes data from available unused disk space on the entire volume. If you use the /w parameter, all other parameters are ignored. The directory specified can be located anywhere in a local volume. If it is a mount point or points to a directory in another volume, the data on that volume is removed. |
| /x[:efsfile] [<FileName>] | Backs up the EFS certificate and keys to the specified file name. If used with :efsfile , /x backs up the user's certificate(s) that were used to encrypt the file. Otherwise, the user's current EFS certificate and keys are backed up. |
| /y | Displays your current EFS certificate thumbnail on the local computer. |
| /adduser [/certhash:<Hash>] | /certfile:] |
| /rekey | Updates the specified encrypted file(s) to use the currently configured EFS key. |
| /removeuser /certhash:<Hash> | Removes a user from the specified file(s). The <i>Hash</i> provided for /certhash must be the SHA1 hash of the certificate to remove. |
| /? | Displays help at the command prompt. |

Remarks

- If the parent directory is not encrypted, an encrypted file could become decrypted when it is modified. Therefore, when you encrypt a file, you should also encrypt the parent directory.
- An administrator can add the contents of a .cer file to the EFS recovery policy to create the recovery agent for users, and then import the .pfx file to recover individual files.
- You can use multiple directory names and wildcards.
- You must put spaces between multiple parameters.

Examples

To display the encryption status of each of the files and subdirectories in the current directory, type:

```
cipher
```

Encrypted files and directories are marked with an **E**. Unencrypted files and directories are marked with a **U**. For example, the following output indicates that the current directory and all its contents are currently unencrypted:

```
Listing C:\Users\MainUser\Documents\  
New files added to this directory will not be encrypted.  
U Private  
U hello.doc  
U hello.txt
```

To enable encryption on the Private directory used in the previous example, type:

```
cipher /e private
```

The following output displays:

```
Encrypting files in C:\Users\MainUser\Documents\  
Private [OK]  
1 file(s) [or directorie(s)] within 1 directorie(s) were encrypted.
```

The **cipher** command displays the following output:

```
Listing C:\Users\MainUser\Documents\  
New files added to this directory will not be encrypted.  
E Private  
U hello.doc  
U hello.txt
```

Note that the Private directory is marked as encrypted.

Additional References

[Command-Line Syntax Key](#)

clip

4/13/2018 • 2 minutes to read • [Edit Online](#)

Redirects command output from the command line to the Windows clipboard. You can then paste this text output into other programs.

For examples of how to use this command, see [Examples](#).

Syntax

```
<Command> | clip  
clip < <FileName>
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|---|
| <Command> | Specifies a command whose output you want to send to the Windows Clipboard. |
| <FileName> | Specifies a file whose contents you want to send to the Windows Clipboard. |
| /? | Displays help at the command prompt. |

Remarks

You can use the **clip** command to copy data directly into any application that can receive text from the Clipboard.

Examples

To copy the current directory listing to the Windows clipboard, type:

```
dir | clip
```

To copy the output of a program called Generic.awk to the Windows Clipboard, type:

```
awk -f generic.awk input.txt | clip
```

To copy the contents of a file called Readme.txt to the Windows Clipboard, type:

```
clip < readme.txt
```

Additional references

[Command-Line Syntax Key](#)

cls

4/13/2018 • 2 minutes to read • [Edit Online](#)

Clears the Command Prompt window.

For examples of how to use this command, see [Examples](#).

Syntax

```
cls
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------------|
| /? | Displays help at the command prompt. |

Examples

To clear all information that appears in the Command Prompt window and return to a blank window, type:

```
cls
```

Additional references

[Command-Line Syntax Key](#)

Cmd

4/13/2018 • 5 minutes to read • [Edit Online](#)

Starts a new instance of the command interpreter, Cmd.exe. If used without parameters, **cmd** displays the version and copyright information of the operating system.

Syntax

```
cmd [/c|/k] [/s] [/q] [/d] [/a|/u] [/t:{<B><F>|<F>}] [/e:{on|off}] [/f:{on|off}] [/v:{on|off}] [<String>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------|--|
| /c | Carries out the command specified by <i>String</i> and then stops. |
| /k | Carries out the command specified by <i>String</i> and continues. |
| /s | Modifies the treatment of <i>String</i> after /c or /k . |
| /q | Turns the echo off. |
| /d | Disables execution of AutoRun commands. |
| /a | Formats internal command output to a pipe or a file as American National Standards Institute (ANSI). |
| /u | Formats internal command output to a pipe or a file as Unicode. |
| /t:{<F> <F>} | Sets the background (<i>B</i>) and foreground (<i>F</i>) colors. |
| /e:on | Enables command extensions. |
| /e:off | Disables commands extensions. |
| /f:on | Enables file and directory name completion. |
| /f:off | Disables file and directory name completion. |
| /v:on | Enables delayed environment variable expansion. |
| /v:off | Disables delayed environment variable expansion. |
| <String> | Specifies the command you want to carry out. |
| /? | Displays help at the command prompt. |

The following table lists valid hexadecimal digits that you can use as the values for and <F>

| VALUE | COLOR |
|-------|--------------|
| 0 | Black |
| 1 | Blue |
| 2 | Green |
| 3 | Aqua |
| 4 | Red |
| 5 | Purple |
| 6 | Yellow |
| 7 | White |
| 8 | Gray |
| 9 | Light blue |
| a | Light green |
| b | Light aqua |
| c | Light red |
| d | Light purple |
| e | Light yellow |
| f | Bright white |

Remarks

- Using multiple commands

To use multiple commands for <String>, separate them by the command separator **&&** and enclose them in quotation marks. For example:

```
"<Command>&&<Command>&&<Command>"
```

- Processing quotation marks

If you specify **/c** or **/k, cmd** processes the remainder of *String*, and quotation marks are preserved only if all of the following conditions are met:

- You do not use **/s**.
- You use exactly one set of quotation marks.
- You do not use any special characters within the quotation marks (for example: & < > () @ ^ |).

- You use one or more white-space characters within the quotation marks.
- The *String* within quotation marks is the name of an executable file.

If the previous conditions are not met, *String* is processed by examining the first character to verify whether it is an opening quotation mark. If the first character is an opening quotation mark, it is stripped along with the closing quotation mark. Any text following the closing quotation marks is preserved.

- Executing registry subkeys

If you do not specify **/d** in *String*, Cmd.exe looks for the following registry subkeys:

HKEY_LOCAL_MACHINE\Software\Microsoft\Command Processor\AutoRun\REG_SZ

HKEY_CURRENT_USER\Software\Microsoft\Command Processor\AutoRun\REG_EXPAND_SZ

If one or both registry subkeys are present, they are executed before all other variables.

Caution

Incorrectly editing the registry may severely damage your system. Before making changes to the registry, you should back up any valued data on the computer.

- Enabling and disabling command extensions

Command extensions are enabled by default in Windows XP. You can disable them for a particular process by using ***/e:off***. You can enable or disable extensions for all ***/cmd*** command-line options on a computer or user session by setting the following ***/REG_DWORD*** values:

/HKEY_LOCAL_MACHINE\Software\Microsoft\Command Processor\EnableExtensions\REG_DWORD

/HKEY_CURRENT_USER\Software\Microsoft\Command Processor\EnableExtensions\REG_DWORD

Set the ***/REG_DWORD*** value to either ***/0x1*** (enabled) or ***/0x0*** (disabled) in the registry by using Regedit.exe. User-specified settings take precedence over computer settings, and command-line options take precedence over registry settings.

[!CAUTION]

Incorrectly editing the registry may severely damage your system. Before making changes to the registry, you should back up any valued data on the computer.

When you enable command extensions, the following commands are affected:

- ***/assoc***
- ***/call***
- ***/chdir (cd)***
- ***/color***
- ***/del (erase)***
- ***/endlocal***
- ***/for***
- ***/ftype***
- ***/goto***
- ***/if***
- ***/mkdir (md)***
- ***/popd***
- ***/prompt***
- ***/pushd***
- ***/set***
- ***/setlocal***
- ***/shift***
- ***/start*** (also includes changes to external command processes)

- Enabling delayed environment variable expansion

If you enable delayed environment variable expansion, you can use the exclamation point character to substitute the value of an environment variable at run time.

- Enabling file and directory name completion

File and directory name completion is not enabled by default. You can enable or disable file name completion for a particular process of the **cmd** command with **/f:{on|off}**. You can enable or disable file and directory name completion for all processes of the **cmd** command on a computer or for a user logon session by setting the following **REG_DWORD** values:

HKEY_LOCAL_MACHINE\Software\Microsoft\Command Processor\CompletionChar\REG_DWORD

HKEY_LOCAL_MACHINE\Software\Microsoft\Command Processor\PathCompletionChar\REG_DWORD

HKEY_CURRENT_USER\Software\Microsoft\Command Processor\CompletionChar\REG_DWORD

HKEY_CURRENT_USER\Software\Microsoft\Command Processor\PathCompletionChar\REG_DWORD

To set the **REG_DWORD** value, run Regedit.exe and use the hexadecimal value of a control character for a particular function (for example, **0x9** is TAB and **0x08** is BACKSPACE). User-specified settings take precedence over computer settings, and command-line options take precedence over registry settings.

Caution

Incorrectly editing the registry may severely damage your system. Before making changes to the registry, you should back up any valued data on the computer.

If you enable file and directory name completion by using **/f:on**, use CTRL+D for directory name completion and CTRL+F for file name completion. To disable a particular completion character in the registry, use the value for white space [**0x20**] because it is not a valid control character.

When you press CTRL+D or CTRL+F, **cmd** processes file and directory name completion. These key combination functions append a wildcard character to *String* (if one is not present), build a list of paths that match, and then display the first matching path. If none of the paths match, the file and directory name completion function beeps and does not change the display. To move through the list of matching paths, press CTRL+D or CTRL+F repeatedly. To move through the list backwards, press the SHIFT key and CTRL+D or CTRL+F simultaneously. To discard the saved list of matching paths and generate a new list, edit *String* and press CTRL+D or CTRL+F. If you switch between CTRL+D and CTRL+F, the saved list of matching paths is discarded and a new list is generated. The only difference between the key combinations CTRL+D and CTRL+F is that CTRL+D only matches directory names and CTRL+F matches both file and directory names. If you use file and directory name completion on any of the built-in directory commands (that is, **CD**, **MD**, or **RD**), directory completion is assumed.

File and directory name completion correctly processes file names that contain white space or special characters if you place quotation marks around the matching path.

The following special characters require quotation marks: & < > [] { } ^ = ; ! ' + , ` ~ [white space].

If the information that you supply contains spaces, use quotation marks around the text (for example, "Computer Name").

If you process file and directory name completion from within *String*, any part of the *Path* to the right of the cursor is discarded (at the point in *String* where the completion was processed).

Additional references

[Command-Line Syntax Key](#)

cmdkey

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

creates, lists, and deletes stored user names and passwords or credentials.

Syntax

```
cmdkey [{/add:<TargetName>|/generic:<TargetName>}] [/smartcard|/user:<UserName> [/pass:<Password>]] [/delete{:  
<TargetName>|/ras}] /list:<TargetName>
```

Parameters

| PARAMETERS | DESCRIPTION |
|------------------|--|
| /add: | adds a user name and password to the list. Requires the parameter of which identifies the computer or domain name that this entry will be associated with. |
| /generic: | adds generic credentials to the list. Requires the parameter of which identifies the computer or domain name that this entry will be associated with. |
| /smartcard | Retrieves the credential from a smart card. |
| /user: | Specifies the user or account name to store with this entry. If <i>UserName</i> is not supplied, it will be requested. |
| /pass: | Specifies the password to store with this entry. If <i>Password</i> is not supplied, it will be requested. |
| /delete: /ras} | deletes a user name and password from the list. If <i>TargetName</i> is specified, that entry will be deleted. If /ras is specified, the stored remote access entry will be deleted. |
| /list: | Displays the list of stored user names and credentials. If <i>TargetName</i> is not specified, all stored user names and credentials will be listed. |
| /? | Displays help at the command prompt. |

Remarks

- if more than one smart card is found on the system when the /smartcard command-line option is used, **cmdkey** will display information about all available smart cards and then prompt the user to specify which one to use.

- Passwords will not be displayed once they are stored. ## Examples To display a list of all user names and credentials that are stored, type: `cmdkey /list` To add a user name and password for user Mikedan to access computer Server01 with the password Kleo, type: `cmdkey /add:server01 /user:mikedan /pass:Kleo` To add a user name and password for user Mikedan to access computer Server01 and prompt for the password whenever Server01 is accessed, type: `cmdkey /add:server01 /user:mikedan` To delete the credential that remote access has stored, type: `cmdkey /delete /ras` To delete the credential that is stored for Server01, type: `cmdkey /delete:Server01` ## additional references [Command-Line Syntax Key](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Installs or removes a Connection Manager service profile. Used without optional parameters, **cmstp** installs a service profile with default settings appropriate to the operating system and to the user's permissions.

Syntax

Syntax 1:

```
ServiceProfileFileName .exe /q:a /c:"cmstp.exe ServiceProfileFileName .inf [/nf] [/ni] [/ns] [/s] [/su] [/u]"
```

Syntax 2:

```
cmstp.exe [/nf] [/ni] [/ns] [/s] [/su] [/u] [Drive:][path]ServiceProfileFileName.inf"
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------------------------|--|
| < ServiceProfileFileName >.exe | <p>Specifies, by name, the installation package that contains the profile that you want to install.</p> <p>Required for Syntax 1 but not valid for Syntax 2.</p> |
| /q:a | <p>Specifies that the profile should be installed without prompting the user. The verification message that the installation has succeeded will still appear.</p> <p>Required for Syntax 1 but not valid for Syntax 2.</p> |
| [Drive:][path] .inf | <p>Required. Specifies, by name, the configuration file that determines how the profile should be installed.</p> <p>The [Drive:][path] parameter is not valid for Syntax 1.</p> |
| /nf | <p>Specifies that the support files should not be installed.</p> |
| /ni | <p>Specifies that a desktop icon should not be created. This parameter is only valid for computers running Windows 95, Windows 98, Windows NT 4.0, or Windows Millennium edition.</p> |
| /ns | <p>Specifies that a desktop shortcut should not be created. This parameter is only valid for computers running a member of the Windows Server 2003 family, Windows 2000, or Windows XP.</p> |

| PARAMETER | DESCRIPTION |
|-----------|--|
| /s | Specifies that the service profile should be installed or uninstalled silently (without prompting for user response or displaying verification message). |
| /su | Specifies that the service profile should be installed for a single user rather than for all users. This parameter is only valid for computers running a Windows Server 2003, Windows 2000, or Windows XP. |
| /au | Specifies that the service profile should be installed for all users. This parameter is only valid for computers running Windows Server 2003, Windows 2000, or Windows XP. |
| /u | Specifies that the service profile should be uninstalled. |
| /? | Displays help at the command prompt. |

Remarks

/s is the only parameter that you can use in combination with **/u**. Syntax 1 is the typical syntax used in a custom installation application. To use this syntax, you must run **cmstp** from the directory that contains the .exe file.

Examples

To install the Fiction service profile without any support files, type:

```
fiction.exe /c:"cmstp.exe fiction.inf /nf"
```

To silently install the Fiction service profile for a single user, type:

```
fiction.exe /c:"cmstp.exe fiction.inf /s /su"
```

To silently uninstall the Fiction service profile, type:

```
fiction.exe /c:"cmstp.exe fiction.inf /s /u"
```

additional references

- [Command-Line Syntax Key](#)

color

4/13/2018 • 2 minutes to read • [Edit Online](#)

Changes the foreground and background colors in the Command Prompt window for the current session. If used without parameters, **color** restores the default Command Prompt window foreground and background colors.

For examples of how to use this command, see [Examples](#).

Syntax

```
color [[<B>]<F>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------------|
| | Specifies the background color. |
| <F> | Specifies the foreground color. |
| /? | Displays help at the command prompt. |

Remarks

- The following table lists valid hexadecimal digits that you can use as the values for *B* and *F*.
|Value|Color| |-----|-----| |0|Black| |1|Blue| |2|Green| |3|Aqua| |4|Red| |5|Purple| |6|Yellow| |7|White| |8|Gray| |9|Light blue| |A|Light green| |B|Light aqua| |C|Light red| |D|Light purple| |E|Light yellow| |F|Bright white|
- Do not use space characters between *B* and *F*.
- If you specify only one hexadecimal digit, the corresponding color is used as the foreground color and the background color is set to the default color.
- To set the default Command Prompt window color, click the upper-left corner of the Command Prompt window, click **Defaults**, click the **Colors** tab, and then click the colors that you want to use for the **Screen Text** and **Screen Background**.
- If *B* and *F* are the same, the **color** command sets ERRORLEVEL to 1, and no change is made to either the foreground or the background color.

Examples

To change the Command Prompt window background color to gray and the foreground color to red, type:

```
color 84
```

To change the Command Prompt window foreground color to light yellow, type:

```
color e
```

NOTE

In this example, the background is set to the default color because only one hexadecimal digit is specified.

Additional references

[Command-Line Syntax Key](#)

comp

4/13/2018 • 3 minutes to read • [Edit Online](#)

Compares the contents of two files or sets of files byte-by-byte. If used without parameters, **comp** prompts you to enter the files to compare.

For examples of how to use this command, see [Examples](#).

Syntax

```
comp [<Data1>] [<Data2>] [/d] [/a] [/l] [/n=<Number>] [/c]
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------|--|
| <Data1> | Specifies the location and name of the first file or set of files that you want to compare. You can use wildcard characters (*) and ?) to specify multiple files. |
| <Data2> | Specifies the location and name of the second file or set of files that you want to compare. You can use wildcard characters (*) and ?) to specify multiple files. |
| /d | Displays differences in decimal format. (The default format is hexadecimal.) |
| /a | Displays differences as characters. |
| /l | Displays the number of the line where a difference occurs, instead of displaying the byte offset. |
| /n= <Number> | Compares only the number of lines that are specified for each file, even if the files are different sizes. |
| /c | Performs a comparison that is not case-sensitive. |
| /off[line] | Processes files with the offline attribute set. |
| /? | Displays Help at the command prompt. |

Remarks

- How the **comp** command identifies mismatching information

During the comparison, **comp** displays messages that identify the locations of unequal information between the files. Each message indicates the offset memory address of the unequal bytes and the contents of the bytes (in hexadecimal notation unless the **/a** or **/d** command-line parameter is specified). Messages appear in the following format:

```
Compare error at OFFSET xxxxxxxx
```

```
file1 = xx
```

```
file2 = xx
```

After ten unequal comparisons, **comp** stops comparing the files and displays the following message:

```
10 Mismatches - ending compare
```

- Handling special cases for *Data1* and *Data2*
 - If you omit necessary components of either *Data1* or *Data2* or if you omit *Data2*, **comp** prompts you for the missing information.
 - If *Data1* contains only a drive letter or a directory name with no file name, **comp** compares all of the files in the specified directory to the file specified in *Data1*.
 - If *Data2* contains only a drive letter or a directory name, the default file name for *Data2* is the same as that in *Data1*.
 - If **comp** cannot find the file(s) you specify, it prompts you with a message to determine whether you want to compare more files.
- Comparing files in different locations

Comp can compare files on the same drive or on different drives, and in the same directory or in different directories. When **comp** compares the files, it displays their locations and file names.

- Comparing files with the same names

The files that you compare can have the same file name, provided they are in different directories or on different drives. If you do not specify a file name for *Data2*, the default file name for *Data2* is the same as the file name in *Data1*. You can use wildcard characters (***** and **?**) to specify file names.

- Comparing files of different sizes

You must specify **/n** to compare files of different sizes. If the file sizes are different and **/n** is not specified, **comp** displays the following message:

```
Files are different sizes
```

```
Compare more files (Y/N)?
```

To compare these files, press N to stop the **comp** command. Then, rerun the **comp** command with the **/n** option to compare only the first portion of each file.

- Comparing files sequentially

If you use wildcard characters (***** and **?**) to specify multiple files, **comp** finds the first file that matches *Data1* and compares it with the corresponding file in *Data2*, if it exists. The **comp** command reports the results of the comparison for each file matching *Data1*. When finished, **comp** displays the following message:

```
Compare more files (Y/N)?
```

To compare more files, press Y. The **comp** command prompts you for the locations and names of the new files. To stop the comparisons, press N. When you press Y, **comp** prompts you for command-line options to use. If you do not specify any command-line options, **comp** uses the ones you specified before.

Examples

To compare the contents of the directory C:\Reports with the backup directory \\Sales\Backup\April, type:

```
comp c:\reports \\sales\backup\april
```

To compare the first ten lines of the text files in the \Invoice directory and display the result in decimal format, type:

```
comp \invoice\*.txt \invoice\backup\*.txt /n=10 /d
```

Additional references

[Command-Line Syntax Key](#)

compact

4/13/2018 • 2 minutes to read • [Edit Online](#)

Displays or alters the compression of files or directories on NTFS partitions. If used without parameters, **compact** displays the compression state of the current directory and the files it contains.

For examples of how to use this command, see [Examples](#).

Syntax

```
compact [/c | /u] [/s[:<Dir>]] [/a] [/i] [/f] [/q] [<FileName>[...]]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|--|
| /c | Compresses the specified directory or file. |
| /u | Uncompresses the specified directory or file. |
| /s[:<Dir>] | Applies the compact command to all subdirectories of the specified directory (or of the current directory if none is specified). |
| /a | Displays hidden or system files. |
| /i | Ignores errors. |
| /f | Forces compression or uncompression of the specified directory or file. /f is used in the case of a file that was partly compressed when the operation was interrupted by a system crash. To force the file to be compressed in its entirety, use the /c and /f parameters and specify the partially compressed file. |
| /q | Reports only the most essential information. |
| <FileName> | Specifies the file or directory. You can use multiple file names, and the * and ? wildcard characters. |
| /? | Displays help at the command prompt. |

Remarks

- The **compact** command is the command-line version of the NTFS file system compression feature. The compression state of a directory indicates whether files are automatically compressed when they are added to the directory. Setting the compression state of a directory does not necessarily change the compression state of files that are already in the directory.
- You cannot use **compact** to read, write, or mount volumes that have been compressed using DriveSpace or DoubleSpace.

- You cannot use **compact** to compress file allocation table (FAT) or FAT32 partitions.

Examples

To set the compression state of the current directory, its subdirectories, and existing files, type:

```
compact /c /s
```

To set the compression state of files and subdirectories within the current directory, without altering the compression state of the current directory itself, type:

```
compact /c /s *.*
```

To compress a volume, from the root directory of the volume, type:

```
compact /c /i /s:\
```

NOTE

This example sets the compression state of all directories (including the root directory on the volume) and compresses every file on the volume. The **/i** parameter prevents error messages from interrupting the compression process.

To compress all files with the .bmp file name extension in the \Tmp directory and all subdirectories of \Tmp, without modifying the compressed attribute of the directories, type:

```
compact /c /s:\tmp *.bmp
```

To force complete compression of the file Zebra.bmp, which was partially compressed during a system crash, type:

```
compact /c /f zebra.bmp
```

To remove the compressed attribute from the directory C:\Tmp, without changing the compression state of any files in that directory, type:

```
compact /u c:\tmp
```

Additional references

[Command-Line Syntax Key](#)

convert

4/13/2018 • 2 minutes to read • [Edit Online](#)

Converts file allocation table (FAT) and FAT32 volumes to the NTFS file system, leaving existing files and directories intact. Volumes converted to the NTFS file system cannot be converted back to FAT or FAT32.

For examples of how to use this command, see [Examples](#).

Syntax

```
convert [<Volume>] /fs:ntfs [/v] [/cvtarea:<FileName>] [/nosecurity] [/x]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------------|--|
| <Volume> | Specifies the drive letter (followed by a colon), mount point, or volume name to convert to NTFS. |
| /fs:ntfs | Required. Converts the volume to NTFS. |
| /v | Runs convert in verbose mode, which displays all messages during the conversion process. |
| /cvtarea:<FileName> | Specifies that the Master File Table (MFT) and other NTFS metadata files are written to an existing, contiguous placeholder file. This file must be in the root directory of the file system to be converted. Use of the /cvtarea parameter can result in a less fragmented file system after conversion. For best results, the size of this file should be 1 KB multiplied by the number of files and directories in the file system, although the convert utility accepts files of any size. Important: You must create the placeholder file by using the fsutil file createnew command prior to running convert . Convert does not create this file for you. Convert overwrites this file with NTFS metadata. After conversion, any unused space in this file is freed. |
| /nosecurity | Specifies that the security settings on the converted files and directories allow access by all users. |
| /x | Dismounts the volume, if necessary, before it is converted. Any open handles to the volume will no longer be valid. |
| /? | Displays help at the command prompt. |

Remarks

- If **convert** cannot lock the drive (for example, the drive is the system volume or the current drive), you are given the option to convert the drive the next time you restart the computer. If you cannot restart the computer immediately to complete the conversion, plan a time to restart the computer and allow extra time for the

conversion process to complete.

- For volumes converted from FAT or FAT32 to NTFS:

Due to existing disk usage, the MFT is created in a different location than on a volume originally formatted with NTFS, so volume performance might not be as good as on volumes originally formatted with NTFS. For optimal performance, consider recreating these volumes and formatting them with the NTFS file system.

Volume conversion from FAT or FAT32 to NTFS leaves the files intact, but the volume might lack some performance benefits compared to volumes initially formatted with NTFS. For example, the MFT might become fragmented on converted volumes. In addition, on converted boot volumes, **convert** applies the same default security that is applied during Windows Setup.

Examples

To convert the volume on drive E to NTFS and display all messages during the conversion process, type:

```
convert e: /fs:ntfs /v
```

Additional references

[Command-Line Syntax Key](#)

copy

4/13/2018 • 6 minutes to read • [Edit Online](#)

Copies one or more files from one location to another.

For examples of how to use this command, see [Examples](#).

Syntax

```
copy [/d] [/v] [/n] [/y | /-y] [/z] [/a | /b] <Source> [/a | /b] [+<Source> [/a | /b] [+ ...]] [<Destination> [/a | /b]]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------|--|
| /d | Allows the encrypted files being copied to be saved as decrypted files at the destination. |
| /v | Verifies that new files are written correctly. |
| /n | Uses a short file name, if available, when copying a file with a name longer than eight characters, or with a file name extension longer than three characters. |
| /y | Suppresses prompting to confirm that you want to overwrite an existing destination file. |
| /-y | Prompts you to confirm that you want to overwrite an existing destination file. |
| /z | Copies networked files in restartable mode. |
| /a | Indicates an ASCII text file. |
| /b | Indicates a binary file. |
| <Source> | Required. Specifies the location from which you want to copy a file or set of files. <i>Source</i> can consist of a drive letter and colon, a directory name, a file name, or a combination of these. |
| <Destination> | Required. Specifies the location to which you want to copy a file or set of files. <i>Destination</i> can consist of a drive letter and colon, a directory name, a file name, or a combination of these. |
| /? | Displays help at the command prompt. |

Remarks

- You can copy an ASCII text file that uses an end-of-file character (CTRL+Z) to indicate the end of the file.
- Using **/a**

When **/a** precedes or follows a list of files on the command line, it applies to all files listed until **copy** encounters **/b**. In this case, **/b** applies to the file preceding **/b**.

The effect of **/a** depends on its position in the command-line string. When **/a** follows *Source*, **copy** treats the file as an ASCII file and copies data that precedes the first end-of-file character (CTRL+Z).

When **/a** follows *Destination*, **copy** adds an end-of-file character (CTRL+Z) as the last character of the file.

- Using **/b**

/b directs the command interpreter to read the number of bytes specified by the file size in the directory. **/b** is the default value for **copy**, unless **copy** combines files.

When **/b** precedes or follows a list of files on the command line, it applies to all listed files until **copy** encounters **/a**. In this case, **/a** applies to the file preceding **/a**.

The effect of **/b** depends on its position in the command-line string. When **/b** follows *Source*, **copy** copies the entire file, including any end-of-file character (CTRL+Z).

When **/b** follows *Destination*, **copy** does not add an end-of-file character (CTRL+Z).

- Using **/v**

If a write operation cannot be verified an error message appears. Although recording errors rarely occur with **copy**, you can use **/v** to verify that critical data has been correctly recorded. The **/v** command-line option also slows down the **copy** command, because each sector recorded on the disk must be checked.

- Using **/y** and **/-y**

If **/y** is preset in the COPYCMD environment variable, you can override this setting by using **/-y** at the command line. By default, you are prompted when you replace this setting, unless the **copy** command is executed in a batch script.

- Appending files

To append files, specify a single file for *Destination*, but multiple files for *Source* (use wildcard characters or *File1+File2+File3* format).

- Using **/z**

If the connection is lost during the copy phase (for example, if the server going offline breaks the connection), **copy /z** resumes after the connection is re-established. **/z** also displays the percentage of the copy operation that is completed for each file.

- Copying to and from devices

You can substitute a device name for one or more occurrences of *Source* or *Destination*.

- Using or omitting **/b** when copying to a device

When *Destination* is a device (for example, Com1 or Lpt1), **/b** copies data to the device in binary mode. In binary mode, **copy /b** copies all characters (including special characters such as CTRL+C, CTRL+S, CTRL+Z, and ENTER) to the device as data. However, if you omit **/b**, data is copied to the device in ASCII mode. In ASCII mode, special characters might cause files to combine during the copying process.

- Using the default destination file

If you do not specify a destination file, a copy is created with the same name, modified date, and modified

time as the original file. The new copy is stored in the current directory on the current drive. If the source file is on the current drive and in the current directory and you do not specify a different drive or directory for the destination file, the **copy** command stops and displays the following error message:

```
File cannot be copied onto itself
```

```
0 File(s) copied
```

- Combining files

If you specify more than one file in *Source*, **copy** combines them all into a single file using the file name specified in *Destination*. **Copy** assumes the combined files are ASCII files unless you use the **/b** option.

- Copying zero-length files

Copy does not copy files that are 0 bytes long. Use **xcopy** to copy these files.

- Changing the time and date of a file

If you want to assign the current time and date to a file without modifying the file, use the following syntax:

```
copy /b <Source> +,,
```

The commas indicate the omission of the *Destination* parameter.

- Copying files in subdirectories

To copy all of a directory's files and subdirectories, use the **xcopy** command.

- The **copy** command, with different parameters, is available from the Recovery Console.

Examples

To copy a file called Memo.doc to Letter.doc in the current drive and ensure that an end-of-file character (CTRL+Z) is at the end of the copied file, type:

```
copy memo.doc letter.doc /a
```

To copy a file named Robin.typ from the current drive and directory to an existing directory named Birds that is located on drive C, type:

```
copy robin.typ c:\birds
```

If the Birds directory does not exist, the file Robin.typ is copied into a file named Birds that is located in the root directory on the disk in drive C.

To combine Mar89.rpt, Apr89.rpt, and May89.rpt, which are located in the current directory, and place them in a file named Report (also in the current directory), type:

```
copy mar89.rpt + apr89.rpt + may89.rpt Report
```

When you combine files, **copy** marks the destination file with the current date and time. If you omit *Destination*, the files are combined and stored under the name of the first file in the list. For example, to combine all files in Report when a file named Report already exists, type:

```
copy report + mar89.rpt + apr89.rpt + may89.rpt
```

To combine all files in the current directory that have the .txt file name extension into a single file named Combined.doc, type:

```
copy *.txt Combined.doc
```

If you want to combine several binary files into one file by using wildcard characters, include **/b**. This prevents Windows from treating CTRL+Z as an end-of-file character. For example, type:

```
copy /b *.exe Combined.exe
```

Caution

If you combine binary files, the resulting file might be unusable due to internal formatting.

In the following example, **copy** combines each file that has a .txt extension with its corresponding .ref file. The result is a file with the same file name but with a .doc extension. **Copy** combines File1.txt with File1.ref to form File1.doc, and then **copy** combines File2.txt with File2.ref to form File2.doc, and so on. For example, type:

```
copy *.txt + *.ref *.doc
```

To combine all files with the .txt extension, and then combine all files with the .ref extension into one file named Combined.doc, type:

```
copy *.txt + *.ref Combined.doc
```

Additional references

[Command-Line Syntax Key](#)

cprofile

4/13/2018 • 2 minutes to read • [Edit Online](#)

Cprofile - Cprofile is deprecated, and is not guaranteed to be supported in future releases of Windows.

Cprofile.exe: Clean profile. This tool is included in all Windows Server 2003 operating systems except Windows Server 2003, Web edition. For more information see [Terminal Services Tools and Settings](#).

#

#

cscript

10/18/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

starts a script so that it runs in a command-line environment.

Syntax

```
cscript <Scriptname.extension> [/B] [/D] [/E:<Engine>] [{/H:cscript|/H:wscript}] [/I] [/Job:<Identifier>]
[{/Logo|/NoLogo}] [/S] [/T:<Seconds>] [/X] [/U] [/?] [<ScriptArguments>]
```

Parameters

| PARAMETER | DESCRIPTION |
|----------------------|--|
| Scriptname.extension | Specifies the path and file name of the script file with optional file name extension. |
| /B | Specifies batch mode, which does not display alerts, scripting errors, or input prompts. |
| /D | Starts the debugger. |
| /E: | Specifies the engine that is used to run the script. |
| /H:cscript | Registers cscript.exe as the default script host for running scripts. |
| /H:wscript | Registers wscript.exe as the default script host for running scripts. This is the default. |
| /I | Specifies interactive mode, which displays alerts, scripting errors, and input prompts. This is the default and the opposite of /B . |
| /Job: | Runs the job identified by <i>Identifier</i> in a .wsf script file. |
| /Logo | Specifies that the Windows Script Host banner is displayed in the console before the script runs. This is the default and the opposite of /NoLogo . |
| /NoLogo | Specifies that the Windows Script Host banner is not displayed before the script runs. |
| /S | Saves the current command-prompt options for the current user. |

| PARAMETER | DESCRIPTION |
|-----------------|---|
| /T: | Specifies the maximum time the script can run (in seconds). You can specify up to 32,767 seconds. The default is no time limit. |
| /U | Specifies Unicode for input and output that is redirected from the console. |
| /X | starts the script in the debugger. |
| /? | Displays available command parameters and provides help for using them. This is the same as typing cscript.exe with no parameters and no script. |
| ScriptArguments | Specifies the arguments passed to the script. Each script argument must be preceded by a slash (/). |

Remarks

- Performing this task does not require you to have administrative credentials. Therefore, as a security best practice, consider performing this task as a user without administrative credentials.
- To open a command prompt, on the **start** screen, type **cmd**, and then click **command prompt**.
- Each parameter is optional; however, you cannot specify script arguments without specifying a script. If you do not specify a script or any script arguments, cscript.exe displays the cscript.exe syntax and the valid host options.
- The **/T** parameter prevents excessive running of scripts by setting a timer. When the run time exceeds the specified value, cscript interrupts the script engine and ends the process.
- Windows script files usually have one of the following file name extensions: .wsf, .vbs, .js.
- You can set properties for individual scripts. See Related Topics for more information.
- Windows Script Host can use .wsf script files. Each .wsf file can use multiple scripting engines and perform multiple jobs.
- if you double-click a script file with an extension that has no association, the **Open With** dialog box appears. select wscript or cscript, and then select **Always use this program to open this file type**. This registers wscript.exe or cscript as the default script host for files of this file type.
- You can set properties for individual scripts. See [additional References](#) for more information.
- Windows Script Host can use .wsf script files. Each .wsf file can use multiple scripting engines and perform multiple jobs.

Additional references

[Command-Line Syntax Key](#)

date

4/13/2018 • 2 minutes to read • [Edit Online](#)

Displays or sets the system date. If used without parameters, **date** displays the current system date setting and prompts you to enter a new date.

For examples of how to use this command, see [Examples](#).

Syntax

```
date [/t | <Month-Day-Year>]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------------|--|
| <Month-Day-Year> | Sets the date specified, where <i>Month</i> is the month (one or two digits), <i>Day</i> is the day (one or two digits), and <i>Year</i> is the year (two or four digits). |
| /t | Displays the current date without prompting you for a new date. |
| /? | Displays help at the command prompt. |

Remarks

- To change the current date, you must have administrative credentials.
- You must separate values for *Month*, *Day*, and *Year* with periods (.), hyphens (-), or slash marks (/).
- Valid *Month* values are 1 through 12.
- Valid *Day* values are 1 through 31.
- Valid *Year* values are either 00 through 99, or 1980 through 2099. If you use two digits, the values 80 through 99 correspond to the years 1980 through 1999.

Examples

If command extensions are enabled, to display the current system date, type:

```
date /t
```

To change the current system date to August 3, 2007, you can type any of the following:

```
date 08.03.2007
date 08-03-07
date 8/3/07
```

To display the current system date, followed by a prompt to enter a new date, type:

```
The current date is: Mon 04/02/2007
Enter the new date: (mm-dd-yy)
```

To keep the current date and return to the command prompt, press ENTER. To change the current date, type the new date and then press ENTER.

Additional references

[Command-Line Syntax Key](#)

dcgpofix

4/13/2018 • 2 minutes to read • [Edit Online](#)

Recreates the default Group Policy Objects (GPOs) for a domain. For examples of how this command can be used, see [Examples](#).

Syntax

```
DCGPOfix [/ignoreschema] [/target: {Domain | DC | Both}] [/?]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------|---|
| /ignoreschema | Ignores the version of the Active Directory® schema mc when you run this command. Otherwise, the command only works on the same schema version as the Windows version in which the command was shipped. |
| /target {Domain | DC |
| /? | Displays Help at the command prompt. |

Remarks

- The **dcgpofix** command is available in Windows Server 2008 R2 and Windows Server 2008, except on Server Core installations.
- Although the Group Policy Management Console (GPMC) is distributed with Windows Server 2008 R2 and Windows Server 2008, you must install Group Policy Management as a feature through Server Manager.

Examples

Restore the Default Domain Policy GPO to its original state. You will lose any changes that you have made to this GPO. As a best practice, you should configure the Default Domain Policy GPO only to manage the default Account Policies settings, Password Policy, Account Lockout Policy, and Kerberos Policy. In this example, you ignore the version of the Active Directory schema so that the **dcgpofix** command is not limited to same schema as the Windows version in which the command was shipped.

```
dcgpofix /ignoreschema /target:Domain
```

Restore the Default Domain Controllers Policy GPO to its original state. You will lose any changes that you have made to this GPO. As a best practice, you should configure the Default Domain Controllers Policy GPO only to set user rights and audit policies. In this example, you ignore the version of the Active Directory schema so that the **dcgpofix** command is not limited to same schema as the Windows version in which the command was shipped.

```
dcgpofix /ignoreschema /target:DC
```

Additional references

- [Group Policy TechCenter](#)
- [Command-Line Syntax Key](#)

defrag

3/21/2018 • 4 minutes to read • [Edit Online](#)

Applies To: Windows 10, Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Locates and consolidates fragmented files on local volumes to improve system performance. Membership in the local **Administrators** group, or equivalent, is the minimum required to run this command.

Syntax

```
defrag <volumes> | /C | /E <volumes> [/H] [/M [n]] [/U] [/V]]
defrag <volumes> | /C | /E <volumes> /A [/H] [/M [n]] [/U] [/V]]
defrag <volumes> | /C | /E <volumes> /X [/H] [/M [n]] [/U] [/V]]
defrag <volume> [/<Parameter>]*
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| <volume> | Specifies the drive letter or mount point path of the volume to be defragmented or analyzed. |
| A | Perform analysis on the specified volumes. |
| C | Perform the operation on all volumes. |
| D | Perform traditional defrag (this is the default). On a tiered volume though, traditional defrag is performed only on the Capacity tier. |
| E | Perform the operation on all volumes except those specified. |
| G | Optimize the storage tiers on the specified volumes. |
| H | Run the operation at normal priority (default is low). |
| I n | Tier optimization would run for at most n seconds on each volume. |
| K | Perform slab consolidation on the specified volumes. |
| L | Perform retrim on the specified volumes. |
| M [n] | Run the operation on each volume in parallel in the background. At most n threads optimize the storage tiers in parallel. |
| O | Perform the proper optimization for each media type. |

| PARAMETER | DESCRIPTION |
|-----------|---|
| T | Track an operation already in progress on the specified volume. |
| U | print the progress of the operation on the screen. |
| V | print verbose output containing the fragmentation statistics. |
| X | Perform free space consolidation on the specified volumes. |
| ? | Displays this help information. |

Remarks

- You cannot defragment specific types of file system volumes or drives:
 - You cannot defragment volumes that the file system has locked.
 - You cannot defragment volumes that the file system has marked as dirty, which indicates possible corruption. You must run **chkdsk** on a dirty volume before you can defragment it. You can determine if a volume is dirty by using the **fsutil** dirty query command. For more information about **chkdsk** and **fsutil** dirty, see [additional references](#).
 - You cannot defragment network drives.
 - You cannot defragment cdROMs.
 - You cannot defragment file system volumes that are not **NTFS**, **ReFS**, **Fat** or **Fat32**.
- With Windows Server 2008 R2 , Windows Server 2008 , and, Windows Vista, you can schedule to defragment a volume. However, you cannot schedule to defragment a Solid State Drive (SSD) or a volume on a Virtual Hard Disk (VHD) that resides on an SSD.
- To perform this procedure, you must be a member of the Administrators group on the local computer, or you must have been delegated the appropriate authority. If the computer is joined to a domain, members of the Domain Admins group might be able to perform this procedure. As a security best practice, consider using **Run As** to perform this procedure.
- A volume must have at least 15% free space for **defrag** to completely and adequately defragment it. **defrag** uses this space as a sorting area for file fragments. If a volume has less than 15% free space, **defrag** will only partially defragment it. To increase the free space on a volume, delete unneeded files or move them to another disk.
- While **defrag** is analyzing and defragmenting a volume, it displays a blinking cursor. When **defrag** is finished analyzing and defragmenting the volume, it displays the analysis report, the defragmentation report, or both reports, and then exits to the command prompt.
- By default, **defrag** displays a summary of both the analysis and defragmentation reports if you do not specify the **/a** or **/v** parameters.
- You can send the reports to a text file by typing `>FileName.txt`, where *FileName.txt* is a file name you specify. For example: `defrag volume /v > FileName.txt`
- To interrupt the defragmentation process, at the command line, press **CTRL+C**.
- Running the **defrag** command and Disk defragmenter are mutually exclusive. If you are using Disk defragmenter to defragment a volume and you run the **defrag** command at a command-line, the **defrag** command fails. Conversely, if you run the **defrag** command and open Disk defragmenter, the defragmentation options in Disk defragmenter are unavailable.

Examples

To defragment the volume on drive C while providing progress and verbose output, type:

```
defrag C: /U /V
```

To defragment the volumes on drives C and D in parallel in the background, type:

```
defrag C: D: /M
```

To perform a fragmentation analysis of a volume mounted on drive C and provide progress, type:

```
defrag C: mountpoint /A /U
```

To defragment all volumes with normal priority and provide verbose output, type:

```
defrag /C /H /V
```

Scheduled task

Defrag's scheduled task runs as a maintenance task and is usually scheduled to run every week. Administrator can change the frequency using **Optimize Drives** application.

- When run from the scheduled task, **defrag** has below policy for SSDs:
 - **Traditional defrag** (i.e. moving files to make them reasonably contiguous) and **retrim** is run only once every month.
 - If both **traditional defrag** and **retrim** are skipped, **analysis** is not run.
 - If user ran **traditional defrag** manually on an SSD, say 3 weeks after the last scheduled task run, then the next scheduled task run will perform **analysis** and **retrim** but skip **traditional defrag** on that SSD.
 - If **analysis** is skipped, the **Last run** time in **Optimize Drives** will not be updated. So for SSDs the **Last run** time in **Optimize Drives** can be a month old.
- This maintenance task might not defrag all the volumes, at times because this task does the following:
 - Doesn't wake the computer in order to run defrag
 - Starts only if the computer is on AC power, and stops if the computer switches to battery power
 - Stops if the computer ceases to be idle

Additional references

- [chkdsk](#)
- [fsutil](#)
- [fsutil dirty](#)
- [Command-Line Syntax Key](#)

del

4/13/2018 • 2 minutes to read • [Edit Online](#)

Deletes one or more files. This command is the same as the **erase** command.

For examples of how to use this command, see [Examples](#).

Syntax

```
del [/p] [/f] [/s] [/q] [/a[:]<Attributes>] <Names>
erase [/p] [/f] [/s] [/q] [/a[:]<Attributes>] <Names>
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------------|--|
| <Names> | Specifies a list of one or more files or directories. Wildcards may be used to delete multiple files. If a directory is specified, all files within the directory will be deleted. |
| /p | Prompts for confirmation before deleting the specified file. |
| /f | Forces deletion of read-only files. |
| /s | Deletes specified files from the current directory and all subdirectories. Displays the names of the files as they are being deleted. |
| /q | Specifies quiet mode. You are not prompted for delete confirmation. |
| /a[:]<Attributes> | Deletes files based on the following file attributes: r Read-only files h Hidden files i Not content indexed files s System files a Files ready for archiving I Reparse points - Prefix meaning 'not' |
| /? | Displays help at the command prompt. |

Remarks

Caution

If you use **del** to delete a file from your disk, you cannot retrieve it.

- If you use **/p**, **del** displays the name of a file and sends the following message:

```
`FileName, Delete (Y/N)?`
```

To confirm the deletion, press Y. To cancel the deletion and display the next file name (that is, if you specified a group of files), press N. To stop the **del** command, press CTRL+C.

- If you disable command extensions, **/s** displays the names of any files that were not found instead of displaying the names of files that are being deleted (that is, the behavior is reversed).
- If you specify a folder in *Names*, all of the files in the folder are deleted. For example, the following command deletes all of the files in the \Work folder:

```
del \work
```

- You can use wildcards (***** and **?**) to delete more than one file at a time. However, to avoid deleting files unintentionally, you should use wildcards cautiously with the **del** command. For example, if you type the following command:

```
del *.*
```

The **del** command displays the following prompt:

```
Are you sure (Y/N)?
```

To delete all of the files in the current directory, press Y and then press ENTER. To cancel the deletion, press N and then press ENTER.

NOTE

Before you use wildcard characters with the **del** command, use the same wildcard characters with the **dir** command to list all the files that will be deleted.

- The **del** command, with different parameters, is available from the Recovery Console.

Examples

To delete all the files in a folder named Test on drive C, type either of the following:

```
del c:\test
del c:\test\*.*
```

To delete all files with the .bat file name extension from the current directory, type:

```
del *.bak
```

To delete all read-only files in the current directory, type:

```
del /a:r *.*
```

Additional references

[Command-Line Syntax Key](#)

dfsrmig

7/11/2018 • 8 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

The `dfsrmig` command migrates SYSvol replication from File Replication Service (FRS) to Distributed File System (DFS) Replication, provides information about the progress of the migration, and modifies active directory Domain Services (AD DS) objects to support the migration. For examples of how to use this command, see the [Examples](#) section later in this document.

Syntax

```
dfsrmig [/SetGlobalState <state> | /GetGlobalState | /GetMigrationState | /createGlobalObjects |  
/deleteRoNtfrsMember [<read_only_domain_controller_name>] | /deleteRoDfsrMember  
[<read_only_domain_controller_name>] | /?]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------|---|
| /SetGlobalState | <p>Sets the desired global migration state for the domain to the state that corresponds to the value specified by <i>state</i>.</p> <p>To proceed through the migration or the rollback processes, use this command to cycle through the valid states. This option enables you to initiate and control the migration process by setting the global migration state in AD DS on the PDC emulator. If the PDC emulator is not available, this command fails.</p> <p>You can only set the global migration state to a stable state. The valid values for <i>state</i>, therefore, are 0 for the start state, 1 for the Prepared state, 2 for the Redirected state, and 3 for the Eliminated state.</p> <p>Migration to the Eliminated state is irreversible and rollback from that state is not possible, so use a value of 3 for <i>state</i> only when you are fully committed to using DFS Replication for SYSvol replication.</p> |

| PARAMETER | DESCRIPTION |
|--------------------|---|
| /GetGlobalState | <p>Retrieves the current global migration state for the domain from the local copy of the AD DS database, when run on the PDC emulator.</p> <p>Use this option to confirm that you set the correct global migration state. Only stable migration states can be global migration states, so the results that the dfsrmig command reports with the /GetGlobalState option correspond to the states you can set with the /SetGlobalState option.</p> <p>You should run the dfsrmig command with the /GetGlobalState option only on the PDC emulator. Active directory replication replicates the global state to the other domain controllers in the domain, but replication latencies can cause inconsistencies if you run the dfsrmig command with the /GetGlobalState option on a domain controller other than the PDC emulator. To check the local migration status of a domain controller other than the PDC emulator, use the /GetMigrationState option instead.</p> |
| /GetMigrationState | <p>Retrieves the current local migration state for all domain controllers in the domain, and determines whether those local states match the current global migration state.</p> <p>Use this option to determine if all domain controllers have reached the global migration state. The output of the dfsrmig command when you use the /GetMigrationState option indicates whether or not migration to the current global state is complete, and it lists the local migration state for any domain controllers that have not reached the current global migration state. Local migration state for domain controllers can include transition states for domain controllers that have not reached the current global migration state.</p> |

| PARAMETER | DESCRIPTION |
|--|---|
| /createGlobalObjects | <p>creates the global objects and settings in AD DS that DFS Replication uses.</p> <p>You should not need to use this option during a normal migration process, because the DFS Replication service automatically creates these AD DS objects and settings during the migration from the start state to the Prepared state. Use this option to manually create these objects and settings in the following situations:</p> <ul style="list-style-type: none"> - A new read-only domain controller is promoted during migration. The DFS Replication service automatically creates the AD DS objects and settings for DFS Replication during the migration from the start state to the Prepared state. If a new read-only domain controller is promoted in the domain after this transition, but before migration to the Eliminated state, then the objects that correspond to the newly activated read-only domain controller are not created in AD DS causing replication and migration to fail. - In this case, you can run the dfsrmig command with the /createGlobalObjects option to manually create the objects on any read-only domain controllers that do not already have them. Running this command does not affect the domain controllers that already have the objects and settings for the DFS Replication service. - The global settings for the DFS Replication service are missing or were deleted. If these settings are missing for a particular domain controller, migration from the start state to the Prepared state stalls at the Preparing transition state for the domain controller. In this case, you can use the dfsrmig command with the /createGlobalObjects option to manually create the settings. Note: Because the global AD DS settings for the DFS Replication service for a read-only domain controller are created on the PDC emulator, these settings need to replicate to the read-only domain controller from the PDC emulator before the DFS Replication service on the read-only domain controller can use these settings. Because of active irectory replication latencies, this replication can take some time to occur. |
| /deleteRoNtfrsMember [<read_only_domain_controller_name>] | <p>deletes the global AD DS settings for FRS replication that correspond to the specified read-only domain controller, or deletes the global AD DS settings for FRS replication for all read-only domain controllers if no value is specified for <i>read_only_domain_controller_name</i>.</p> <p>You should not need to use this option during a normal migration process, because the DFS Replication service automatically deletes these AD DS settings during the migration from the Redirected state to the Eliminated state. Because read-only domain controllers cannot delete these settings from AD DS, the PDC emulator performs this operation, and the changes eventually replicate to the read-only domain controllers after the applicable latencies for active directory replication.</p> <p>You use this option to manually delete the AD DS settings only when the automatic deletion fails on a read-only domain controller and stalls the read-only domain controller for a long ime during the migration from the Redirected state to the Eliminated state.</p> |

| PARAMETER | DESCRIPTION |
|--|--|
| /deleteRoDfsrMember [<read_only_domain_controller_name>] | <p>deletes the global AD DS settings for DFS Replication that correspond to the specified read-only domain controller, or deletes the global AD DS settings for DFS Replication for all read-only domain controllers if no value is specified for <i>read_only_domain_controller_name</i>.</p> <p>Use this option to manually delete the AD DS settings only when the automatic deletion fails on a read-only domain controller and stalls the read-only domain controller for a long time when rolling back the migration from the Prepared state to the start state.</p> |
| /? | Displays help at the command prompt. Equivalent to running dfsrmig without any options. |

Remarks

- dfsrmig.exe, the migration tool for the DFS Replication service, is installed with the DFS Replication service. for a new Windows Server 2008 server, Dcpromo.exe installs and starts the DFS Replication service when you promote the computer to a domain controller. When you upgrade a server from Windows Server 2003 to Windows Server 2008 , the upgrade process installs and starts the DFS Replication service. You do not need to install the DFS Replication role service to have the DFS Replication service installed and started.
- The **dfsrmig** tool is supported only on domain controllers that run at the Windows Server 2008 domain functional level, because SYSvol migration from FRS to DFS Replication is only possible on domain controllers that operate at the Windows Server 2008 domain functional level.
- You can run the **dfsrmig** command on any domain controller, but operations that create or manipulate AD DS objects are only allowed on read-write capable domain controllers (not on read-only domain controllers).
- Running **dfsrmig** without any options displays help at the command prompt. ## Examples To set the global migration state to prepared (1) and initiate migration to or rollback from the Prepared state, type:
`dfsrmig /SetGlobalState 1` To set the global migration state to start (0) and initiate rollback to the start state, type: `dfsrmig /SetGlobalState 0` To display the global migration state, type: `dfsrmig /GetGlobalState` This example shows typical output from the **dfsrmig /GetGlobalState** command.

`Current DFSR global state: Prepared Succeeded.` To display the information about whether the local migration states on all of the domain controllers match the global migration state and the local migration states for any domain controllers where the local state does not match the global state, type: `dfsrmig /GetMigrationState` This example shows typical output from the **dfsrmig /GetMigrationState** command when the local migration states on all of the domain controllers match the global migration state.

`All Domain Controllers have migrated successfully to Global state (Prepared). Migration has reached a consistent state on all Domain Controllers. Succeeded.`

This example shows typical output from the **dfsrmig /GetMigrationState** command when the local migration states on some domain controllers do not match the global migration state. `` The following Domain Controllers are not in sync with Global state (Prepared):

Domain Controller (Local Migration State) DC type

CONTOSO-DC2 (start) ReadOnly DC CONTOSO-DC3 (Preparing) Writable DC Migration has not yet reached a consistent state on all domain controllers State information might be stale due to AD latency.

To create the global objects and settings that DFS Replication uses in AD DS on domain controllers where those settings were not created automatically during migration or where those settings are missing, type:

dfsrmig /createGlobalObjects

To delete the global AD DS settings for FRS replication for a read-only domain controller named contoso-dc2 if those settings were not deleted automatically by the migration process, type:

dfsrmig /deleteRoNtfrsMember contoso-dc2

To delete the global AD DS settings for FRS replication for all read-only domain controllers if those settings were not deleted automatically by the migration process, type:

dfsrmig /deleteRoNtfrsMember

To delete the global AD DS settings for DFS Replication for a read-only domain controller named contoso-dc2 if those settings were not deleted automatically by the migration process, type:

dfsrmig /deleteRoDfsrMember contoso-dc2

To delete the global AD DS settings for DFS Replication for all read-only domain controllers if those settings were not deleted automatically by the migration process, type:

dfsrmig /deleteRoDfsrMember

additional references

[Command-Line Syntax Key](<https://go.microsoft.com/fwlink/?LinkId=122056>)

[SYSvol Migration Series: Part 2 dfsrmig.exe: The SYSvol Migration Tool](<https://go.microsoft.com/fwlink/?LinkId=121757>)

diantz

4/13/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

This command is the same as the **makecab** command. See [makecab](#) for syntax and parameters.

additional references

- [Command-Line Syntax Key](#)

dir

4/13/2018 • 6 minutes to read • [Edit Online](#)

Displays a list of a directory's files and subdirectories. If used without parameters, **dir** displays the disk's volume label and serial number, followed by a list of directories and files on the disk (including their names and the date and time each was last modified). For files, **dir** displays the name extension and the size in bytes. **Dir** also displays the total number of files and directories listed, their cumulative size, and the free space (in bytes) remaining on the disk.

For examples of how to use this command, see [Examples](#).

Syntax

```
dir [<Drive>:][<Path>][<FileName>] [...] [/p] [/q] [/w] [/d] [/a[:<Attributes>]] [/o[:<SortOrder>]] [/t[:<TimeField>]] [/s] [/b] [/l] [/n] [/x] [/c] [/4]
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------|---|
| [<Drive>:] | Specifies the drive and directory for which you want to see a listing. |
| [<FileName>] | Specifies a particular file or group of files for which you want to see a listing. |
| /p | Displays one screen of the listing at a time. To see the next screen, press any key on the keyboard. |
| /q | Displays file ownership information. |
| /w | Displays the listing in wide format, with as many as five file names or directory names on each line. |
| /d | Displays the listing in the same format as /w , but the files are sorted by column. |

| PARAMETER | DESCRIPTION |
|---------------------|--|
| /a[[:]<Attributes>] | <p>Displays only the names of those directories and files with the attributes that you specify. If you omit /a, dir displays the names of all files except hidden and system files. If you use /a without specifying <i>Attributes</i>, dir displays the names of all files, including hidden and system files.</p> <p>The following list describes each of the values that you can use for <i>Attributes</i>. Using a colon (:) is optional. Use any combination of these values, and do not separate the values with spaces.</p> <ul style="list-style-type: none"> d Directories h Hidden files s System files l Reparse points r Read-only files a Files ready for archiving i Not content indexed files - Prefix meaning "not" |
| /o[[:]<SortOrder>] | <p>Sorts the output according to <i>SortOrder</i>, which can be any combination of the following values:</p> <ul style="list-style-type: none"> n By name (alphabetical) e By extension (alphabetical) g Group directories first s By size (smallest first) d By date/time (oldest first) - Prefix to reverse order <p>Note: Using a colon is optional. Multiple values are processed in the order in which you list them. Do not separate multiple values with spaces.</p> <p>If <i>SortOrder</i> is not specified, dir /o lists the directories in alphabetic order, followed by the files, which are also sorted in alphabetic order.</p> |
| /t[[:]<TimeField>] | <p>Specifies which time field to display or use for sorting. The following list describes each of the values you can use for <i>TimeField</i>:</p> <ul style="list-style-type: none"> c Creation a Last access w Last written |
| /s | Lists every occurrence of the specified file name within the specified directory and all subdirectories. |
| /b | Displays a bare list of directories and files, with no additional information. /b overrides /w . |
| /l | Displays unsorted directory names and file names in lowercase. |
| /n | Displays a long list format with file names on the far right of the screen. |
| /x | Displays the short names generated for non-8dot3 file names. The display is the same as the display for /n , but the short name is inserted before the long name. |
| /c | Displays the thousand separator in file sizes. This is the default behavior. Use /-c to hide separators. |

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------------|
| /4 | Displays years in four-digit format. |
| /? | Displays help at the command prompt. |

Remarks

- To use multiple *FileName* parameters, separate each file name with a space, comma, or semicolon.
- You can use wildcard characters (* or ?), to represent one or more characters of a file name and to display a subset of files or subdirectories.

Asterisk (*): Use the asterisk as a substitute for any string of characters, for example:

- **dir *.txt** lists all files in the current directory with extensions that begin with .txt, such as .txt, .txt1, .txt_old.
- **dir read*.txt** lists all files in the current directory that begin with "read" and with extensions that begin with .txt, such as .txt, .txt1, or .txt_old.
- **dir read*.*** lists all files in the current directory that begin with "read" with any extension.

The asterisk wildcard always uses short file name mapping, so you might get unexpected results. For example, the following directory contains two files (t.txt2 and t97.txt):

```
C:\test>dir /x
Volume in drive C has no label.
Volume Serial Number is B86A-EF32

Directory of C:\test

11/30/2004  01:40 PM <DIR>  .
11/30/2004  01:40 PM <DIR>  ..
11/30/2004  11:05 AM 0 T97B4~1.TXT t.txt2
11/30/2004  01:16 PM 0 t97.txt
```

You might expect that typing **dir t97*** would return the file t97.txt. However, typing **dir t97*** returns both files, because the asterisk wildcard matches the file t.txt2 to t97.txt by using its short name map T97B4~1.TXT. Similarly, typing **del t97*** would delete both files.

Question mark (?): Use the question mark as a substitute for a single character in a name. For example, typing **dir read???.txt** lists any files in the current directory with the .txt extension that begin with "read" and are followed by up to three characters. This includes Read.txt, Read1.txt, Read12.txt, Read123.txt, and Readme1.txt, but not Readme12.txt.

- Specifying file display attributes

If you use **/a** with more than one value in *Attributes*, **dir** displays the names of only those files with all the specified attributes. For example, if you use **/a** with **r** and **-h** as attributes (by using either **/a:r-h** or **/a-rh**), **dir** will only display the names of the read-only files that are not hidden.

- Specifying file name sorting

If you specify more than one *SortOrder* value, **dir** sorts the file names by the first criterion, then by the second criterion, and so on. For example, if you use **/o** with the **e** and **-s** values for *SortOrder* (by using either **/o:e-s** or **/oe-s**), **dir** sorts the names of directories and files by extension, with the largest first, and then displays the final result. The alphabetic sorting by extension causes file names with no extensions to appear first, then directory names, and then file names with extensions.

- Using redirection symbols and pipes

When you use the redirection symbol (>) to send **dir** output to a file or a pipe (|) to send **dir** output to another command, use **/a:-d** and **/b** to list the file names only. You can use *FileName* with **/b** and **/s** to specify that **dir** is to search the current directory and its subdirectories for all file names that match *FileName*. **Dir** lists only the drive letter, directory name, file name, and file name extension (one path per line), for each file name it finds. Before you use a pipe to send **dir** output to another command, you should set the TEMP environment variable in your Autoexec.nt file.

- The **dir** command, with different parameters, is available from the Recovery Console.

Examples

To display all directories one after the other, in alphabetical order, in wide format, and pausing after each screen, make sure that the root directory is the current directory, and then type:

```
dir /s/w/o/p
```

Dir lists the root directory, the subdirectories, and the files in the root directory, including extensions. Then, **dir** lists the subdirectory names and file names in each subdirectory in the tree.

To alter the preceding example so that **dir** displays the file names and extensions, but omits the directory names, type:

```
dir /s/w/o/p/a:-d
```

To print a directory listing, type:

```
dir > prn
```

When you specify **prn**, the directory list is sent to the printer that is attached to the LPT1 port. If your printer is attached to a different port, you must replace **prn** with the name of the correct port.

You can also redirect output of the **dir** command to a file by replacing **prn** with a file name. You can also type a path. For example, to direct **dir** output to the file dir.doc in the Records directory, type:

```
dir > \records\dir.doc
```

If dir.doc does not exist, **dir** creates it, unless the Records directory does not exist. In that case, the following message appears:

```
File creation error
```

To display a list of all the file names with the .txt extension in all directories on drive C, type:

```
dir c:\*.txt /w/o/s/p
```

Dir displays, in wide format, an alphabetized list of the matching file names in each directory, and it pauses each time the screen fills until you press any key to continue.

Additional references

[Command-Line Syntax Key](#)

diskcomp

4/13/2018 • 3 minutes to read • [Edit Online](#)

Compares the contents of two floppy disks. If used without parameters, **diskcomp** uses the current drive to compare both disks. For examples of how to use this command, see [Examples](#).

Syntax

```
diskcomp [<Drive1>: [<Drive2>:]]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| <Drive1> | Specifies the drive containing one of the floppy disks. |
| <Drive2> | Specifies the drive containing the other floppy disk. |
| /? | Displays help at the command prompt. |

Remarks

- Using disks

The **diskcomp** command works only with floppy disks. You cannot use **diskcomp** with a hard disk. If you specify a hard disk drive for *Drive1* or *Drive2*, **diskcomp** displays the following error message:

```
Invalid drive specification
Specified drive does not exist
or is nonremovable
```

- Comparing disks

If all tracks on the two disks being compared are the same, **diskcomp** displays the following message:

```
Compare OK
```

If the tracks are not the same, **diskcomp** displays a message similar to the following:

```
Compare error on
side 1, track 2
```

When **diskcomp** completes the comparison, it displays the following message:

```
Compare another diskette (Y/N)?
```

If you press Y, **diskcomp** prompts you to insert the disk for the next comparison. If you press N, **diskcomp**

stops the comparison.

When **diskcomp** makes the comparison, it ignores a disk's volume number.

- Omitting drive parameters

If you omit the *Drive2* parameter, **diskcomp** uses the current drive for *Drive2*. If you omit both drive parameters, **diskcomp** uses the current drive for both. If the current drive is the same as *Drive1*, **diskcomp** prompts you to swap disks as necessary.

- Using one drive

If you specify the same floppy disk drive for *Drive1* and *Drive2*, **diskcomp** compares them by using one drive and prompts you to insert the disks as necessary. You might have to swap the disks more than once, depending on the capacity of the disks and the amount of available memory.

- Comparing different types of disks

Diskcomp cannot compare a single-sided disk with a double-sided disk, nor a high-density disk with a double-density disk. If the disk in *Drive1* is not of the same type as the disk in *Drive2*, **diskcomp** displays the following message:

```
Drive types or diskette types not compatible
```

- Using **diskcomp** with networks and redirected drives

Diskcomp does not work on a network drive or on a drive created by the **subst** command. If you attempt to use **diskcomp** with a drive of any of these types, **diskcomp** displays the following error message:

```
Invalid drive specification
```

- Comparing an original disk with a copy

When you use **diskcomp** with a disk that you made by using **copy**, **diskcomp** might display a message similar to the following:

```
Compare error on  
side 0, track 0
```

This type of error can occur even if the files on the disks are identical. Although **copy** duplicates information, it does not necessarily place it in the same location on the destination disk.

- Understanding **diskcomp** exit codes

The following table explains each exit code.

| EXIT CODE | DESCRIPTION |
|-----------|-------------------------------|
| 0 | Disks are the same |
| 1 | Differences were found |
| 3 | Hard error occurred |
| 4 | Initialization error occurred |

To process exit codes that are returned by **diskcomp**, you can use the ERRORLEVEL environment variable on the **if** command line in a batch program.

Examples

If your computer has only one floppy disk drive (for example, drive A), and you want to compare two disks, type:

```
diskcomp a: a:
```

Diskcomp prompts you to insert each disk, as needed.

The following example illustrates how to process a **diskcomp** exit code in a batch program that uses the ERRORLEVEL environment variable on the **if** command line:

```
rem Checkout.bat compares the disks in drive A and B
echo off
diskcomp a: b:
if errorlevel 4 goto ini_error
if errorlevel 3 goto hard_error
if errorlevel 1 goto no_compare
if errorlevel 0 goto compare_ok
:ini_error
echo ERROR: Insufficient memory or command invalid
goto exit
:hard_error
echo ERROR: An irrecoverable error occurred
goto exit
:break
echo "You just pressed CTRL+C" to stop the comparison
goto exit
:no_compare
echo Disks are not the same
goto exit
:compare_ok
echo The comparison was successful; the disks are the same
goto exit
:exit
```

Additional references

[Command-Line Syntax Key](#)

diskcopy

5/7/2018 • 3 minutes to read • [Edit Online](#)

Copies the contents of the floppy disk in the source drive to a formatted or unformatted floppy disk in the destination drive. If used without parameters, **diskcopy** uses the current drive for the source disk and the destination disk.

For examples of how to use this command, see [Examples](#).

NOTE

This command isn't included in Windows 10.

Syntax

```
diskcopy [<Drive1>: [<Drive2>:]] [/v]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| <Drive1> | Specifies the drive that contains the source disk. |
| <Drive2> | Specifies the drive that contains the destination disk. |
| /v | Verifies that the information is copied correctly. This option slows down the copying process. |
| /? | Displays help at the command prompt. |

Remarks

- Using disks

Diskcopy works only with removable disks such as floppy disks, which must be the same type. You cannot use **diskcopy** with a hard disk. If you specify a hard disk drive for *Drive1* or *Drive2*, **diskcopy** displays the following error message:

```
Invalid drive specification
Specified drive does not exist or is nonremovable
```

The **diskcopy** command prompts you to insert the source and destination disks and waits for you to press any key on the keyboard before continuing.

After it copies the disk, **diskcopy** displays the following message:

```
Copy another diskette (Y/N)?
```

If you press Y, **diskcopy** prompts you to insert source and destination disks for the next copy operation. To stop the **diskcopy** process, press N.

If you are copying to an unformatted floppy disk in *Drive2*, **diskcopy** formats the disk with the same number of sides and sectors per track as are on the disk in *Drive1*. **Diskcopy** displays the following message while it formats the disk and copies the files:

```
Formatting while copying
```

- Disk serial numbers

If the source disk has a volume serial number, **diskcopy** creates a new volume serial number for the destination disk and displays the number when the copy operation is complete.

- Omitting drive parameters

If you omit the *Drive2* parameter, **diskcopy** uses the current drive as the destination drive. If you omit both drive parameters, **diskcopy** uses the current drive for both. If the current drive is the same as *Drive1*, **diskcopy** prompts you to swap disks as necessary.

- Using one drive for copying

Run **diskcopy** from a drive other than the floppy disk drive, for example the C drive. If floppy disk *Drive1* and floppy disk *Drive2* are the same, **diskcopy** prompts you to switch disks. If the disks contain more information than the available memory can hold, **diskcopy** cannot read all of the information at once. **Diskcopy** reads from the source disk, writes to the destination disk, and prompts you to insert the source disk again. This process continues until you have copied the entire disk.

- Avoiding disk fragmentation

Fragmentation is the presence of small areas of unused disk space between existing files on a disk. A fragmented source disk can slow down the process of finding, reading, or writing files.

Because **diskcopy** makes an exact copy of the source disk on the destination disk, any fragmentation on the source disk is transferred to the destination disk. To avoid transferring fragmentation from one disk to another, use **copy** or **xcopy** to copy your disk. Because **copy** and **xcopy** copy files sequentially, the new disk is not fragmented.

NOTE

You cannot use **xcopy** to copy a startup disk.

- Understanding **diskcopy** exit codes

The following table explains each exit code.

| Exit code | Description |
|-----------|------------------------------------|
| 0 | Copy operation was successful |
| 1 | Nonfatal Read/Write error occurred |
| 3 | Fatal hard error occurred |
| 4 | Initialization error occurred |

To process the exit codes that are returned by **diskcopy**, you can use the `*ERRORLEVEL*` environment variable on the **if** command line in a batch program.

Examples

To copy the disk in drive B to the disk in drive A, type:

```
diskcopy b: a:
```

To use floppy disk drive A to copy one floppy disk to another, first switch to the C drive and then type:

```
diskcopy a: a:
```

Additional references

[Command-Line Syntax Key](#)

DiskPart commands

11/14/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows 10, Windows 8.1, Windows 8, Windows 7, Windows Server 2019, Windows Server 2016, Windows Server 2012 R2, Windows Server 2012, and Windows Server 2008 R2, Windows Server 2008

DiskPart commands help you to manage your PC's drives (disks, partitions, volumes, or virtual hard disks). Before you can use DiskPart commands, you must first list, and then select an object to give it focus. When an object has focus, any DiskPart commands that you type will act on that object.

You can list the available objects and determine an object's number or drive letter by using the **list disk**, **list volume**, **list partition**, and **list vdisk** commands. The **list disk**, **list vdisk** and **list volume** commands display all disks and volumes on the computer. However, the **list partition** command only displays partitions on the disk that has focus. When you use the **list** commands, an asterisk (*) appears next to the object with focus.

When you select an object, the focus remains on that object until you select a different object. For example, if the focus is set on disk 0 and you select volume 8 on disk 2, the focus shifts from disk 0 to disk 2, volume 8. Some commands automatically change the focus. For example, when you create a new partition, the focus automatically switches to the new partition.

You can only give focus to a partition on the selected disk. When a partition has focus, the related volume (if any) also has focus. When a volume has focus, the related disk and partition also have focus if the volume maps to a single specific partition. If this is not the case, focus on the disk and partition is lost.

DiskPart commands

To start the DiskPart command interpreter, at the command prompt type:

```
diskpart
```

IMPORTANT

Membership in the local **Administrators** group, or equivalent, is the minimum required to run DiskPart.

You can run the following commands in the Diskpart command interpreter:

- [Active](#)
- [Add](#)
- [Assign](#)
- [Attach vdisk](#)
- [Attributes](#)
- [Automount](#)
- [Break](#)
- [Clean](#)
- [Compact vdisk](#)
- [Convert](#)

- [Create](#)
- [Delete](#)
- [Detach vdisk](#)
- [Detail](#)
- [Exit](#)
- [Expand vdisk](#)
- [Extend](#)
- [Filesystems](#)
- [Format](#)
- [GPT](#)
- [Help](#)
- [Import](#)
- [Inactive](#)
- [List](#)
- [Merge vdisk](#)
- [Offline](#)
- [Online](#)
- [Recover](#)
- [Rem](#)
- [Remove](#)
- [Repair](#)
- [Rescan](#)
- [Retain](#)
- [San](#)
- [Select](#)
- [Set id](#)
- [Shrink](#)
- [Uniqueid](#)

Additional references

[Command-Line Syntax Key](#)

[Storage Cmdlets in Windows PowerShell](#)

diskperf

4/13/2018 • 2 minutes to read • [Edit Online](#)

In Windows 2000, physical and logical disk performance counters are not enabled by default.

Diskperf is included in Windows XP, Windows Server 2003, Windows Server 2008, Windows Vista, Windows Server 2008 R2, and Windows 7 so that it can be used to remotely enable or disable physical or logical disk performance counters on computers running Windows 2000.

Syntax

```
diskperf [-Y[D|V] | -N[D|V]] [\\computername]
```

Options

| OPTION | DESCRIPTION |
|------------------|---|
| -? | Displays context sensitive help. |
| -Y | Start all disk performance counters when the computer restarts. |
| -YD | Enable disk performance counters for physical drives when the computer restarts. |
| -YV | Enable disk performance counters for logical drives or storage volumes when the computer restarts. |
| -N | Disable all disk performance counters when the computer restarts. |
| -ND | Disable disk performance counters for physical drives when the computer restarts. |
| -NV | Disable disk performance counters for logical drives or storage volumes when the computer restarts. |
| \\<computername> | Specify the name of the computer where you want to enable or disable disk performance counters. |

diskraid

4/13/2018 • 26 minutes to read • [Edit Online](#)

DiskRAID is a command-line tool that enables you to configure and manage redundant array of independent (or inexpensive) disks (RAID) storage subsystems.

RAID is a method used to standardize and categorize fault-tolerant disk systems. RAID levels provide various mixes of performance, reliability, and cost. RAID is usually used on servers. Some servers provide three of the RAID levels: Level 0 (striping), Level 1 (mirroring), and Level 5 (striping with parity).

A hardware RAID subsystem distinguishes physically addressable storage units from one another by using a Logical Unit Number (LUN). A LUN object must have at least one plex, and can have any number of additional plexes. Each plex contains a copy of the data on the LUN object. Plexes can be added to and removed from a LUN object.

Most DiskRAID commands operate on a specific host bus adapter (HBA) port, initiator adapter, initiator portal, provider, subsystem, controller, port, drive, LUN, target portal, target, or target portal group. You use the SELECT command to select an object. The selected object is said to have focus. Focus simplifies common configuration tasks, such as creating multiple LUNs within the same subsystem.

NOTE

The DiskRAID command-line tool works only with storage subsystems that support Virtual Disk Service (VDS).

DiskRAID commands

To view the command syntax, click a command:

- [add](#)
- [associate](#)
- [automagic](#)
- [break](#)
- [chap](#)
- [create](#)
- [delete](#)
- [detail](#)
- [dissociate](#)
- [exit](#)
- [extend](#)
- [flushcache](#)
- [help](#)
- [importtarget](#)
- [initiator](#)
- [invalidatecache](#)
- [lbpolicy](#)
- [list](#)
- [login](#)
- [logout](#)

- [maintenance](#)
- [name](#)
- [offline](#)
- [online](#)
- [recover](#)
- [reenumerate](#)
- [refresh](#)
- [rem](#)
- [remove](#)
- [replace](#)
- [reset](#)
- [select](#)
- [setflag](#)
- [shrink](#)
- [standby](#)
- [unmask](#)

add

Adds an existing LUN to the currently selected LUN, or adds an iSCSI target portal to the currently selected iSCSI target portal group.

Syntax

```
add plex lun=n [noerr]
add tpgroup tportal=n [noerr]
```

Parameters

plex lun=*n*

Specifies the LUN number to add as a plex to the currently selected LUN.

Caution

All data on the LUN being added as a plex will be deleted.

tpgroup tportal=*n*

Specifies the iSCSI target portal number to add to the currently selected iSCSI target portal group.

noerr

Specifies that any failures that occur while performing this operation will be ignored. This is useful in script mode.

associate

Sets the specified list of controller ports as active for the currently selected LUN (other controller ports are made inactive), or adds the specified controller ports to the list of existing active controller ports for the currently selected LUN, or associates the specified iSCSI target for the currently selected LUN.

Syntax

```
associate controllers [add] <n>[,<n> [...]]
associate ports [add] <n-m>[,<n-m> [...]]
associate targets [add] <n>[,<n> [...]]
```

Parameters

controllers

For use with VDS 1.0 providers only. Adds to or replaces the list of controllers that are associated with the currently selected LUN.

ports

For use with VDS 1.1 providers only. Adds to or replaces the list of controller ports that are associated with the currently selected LUN.

targets

For use with VDS 1.1 providers only. Adds to or replaces the list of iSCSI targets that are associated with the currently selected LUN.

add

For VDS 1.0 providers, adds the specified controllers to the existing list of controllers associated with the LUN. If this parameter is not specified, the list of controllers replaces the existing list of controllers associated with this LUN.

For VDS 1.1 providers, adds the specified controller ports to the existing list of controller ports associated with the LUN. If this parameter is not specified, the list of controller ports replaces the existing list of controller ports associated with this LUN.

```
<n>[,<n> [, ...]]
```

For use with the **controllers** or **targets** parameter. Specifies the numbers of the controllers or iSCSI targets to set to active or associate.

```
<n-m>[,<n-m>[,...]]
```

For use with the **ports** parameter. Specifies the controller ports to set active using a controller number (*n*) and port number (*m*) pair.

Example

The following example shows how to associate and add ports to a LUN that uses a VDS 1.1 provider:

```
DISKRAID> SEL LUN 5
LUN 5 is now the selected LUN.

DISKRAID> ASSOCIATE PORTS 0-0,0-1
Controller port associations changed.
(Controller ports active after this command: Ctlr 0 Port 0, Ctlr 0 Port 1)

DISKRAID> ASSOCIATE PORTS ADD 1-1
Controller port associations changed.
(Controller ports active after this command: Ctlr 0 Port 0, Ctlr 0 Port 1, Ctlr 1 Port 1)
```

automagic

Sets or clears flags that give hints to providers on how to configure a LUN. Used with no parameters, the **automagic** operation displays a list of flags.

Syntax

```
automagic {set | clear | apply} all <flag=value> [<flag=value> [...]]
```

Parameters

set

Sets the specified flags to the specified values.

clear

Clears the specified flags. The **all** keyword clears all the automagic flags.

apply

Applies the current flags to the selected LUN.

<flag>

Flags are identified by three-letter acronyms.

| FLAG | DESCRIPTION |
|------|--------------------------------|
| FCR | Fast Crash Recovery Required |
| FTL | Fault Tolerant |
| MSR | Mostly Reads |
| MXD | Maximum Drives |
| MXS | Maximum Size Expected |
| ORA | Optimal Read Alignment |
| ORS | Optimal Read Size |
| OSR | Optimize For Sequential Reads |
| OSW | Optimize For Sequential Writes |
| OWA | Optimal Write Alignment |
| OWS | Optimal Write Size |
| RBP | Rebuild Priority |
| RBV | Read Back Verify Enabled |
| RMP | Remap Enabled |
| STS | Stripe Size |
| WTC | Write-Through Caching Enabled |
| YNK | Removable |

break

Removes the plex from the currently selected LUN. The plex and the data it contained are not retained, and the drive extents may be reclaimed.

Syntax

```
break plex=<plex_number> [noerr]
```

Parameters

plex

Specifies the number of the plex to remove. The plex and the data it contained will not be retained, and the resources used by this plex will be reclaimed. The data contained on the LUN is not guaranteed to be consistent. If you want to retain this plex, use the Volume Shadow Copy Service (VSS).

noerr

Specifies that any failures that occur while performing this operation will be ignored. This is useful in script mode.

Remarks

NOTE

You must first select a mirrored LUN before using the **break** command.

Caution

All data on the plex will be deleted.

Caution

All data contained on the original LUN is not guaranteed to be consistent.

chap

Sets the Challenge Handshake Authentication Protocol (CHAP) shared secret so that iSCSI initiators and iSCSI targets can communicate with one another.

Syntax

```
chap initiator set secret=[<secret>] [target=<target>]
chap initiator remember secret=[<secret>] target=<target>
chap target set secret=[<secret>] [initiator=<initiatorname>]
chap target remember secret=[<secret>] initiator=<initiatorname>
```

Parameters

initiator set

Sets the shared secret in the local iSCSI initiator service used for mutual CHAP authentication when the initiator authenticates the target.

initiator remember

Communicates the CHAP secret of an iSCSI target to the local iSCSI initiator service so that the initiator service can use the secret in order to authenticate itself to the target during CHAP authentication.

target set

Sets the shared secret in the currently selected iSCSI target used for CHAP authentication when the target authenticates the initiator.

target remember

Communicates the CHAP secret of an iSCSI initiator to the current in-focus iSCSI target so that the target can use the secret in order to authenticate itself to the initiator during mutual CHAP authentication.

secret

Specifies the secret to use. If empty the secret will be cleared.

target

Specifies a target in the currently selected subsystem to associate with the secret. This is optional when setting a secret on the initiator and leaving it out indicates that the secret will be used for all targets that do not already have an associated secret.

initiatorname

Specifies an initiator iSCSI name to associate with the secret. This is optional when setting a secret on a target and leaving it out indicates that the secret will be used for all initiators that do not already have an associated secret.

create

Creates a new LUN or iSCSI target on the currently selected subsystem, or creates a target portal group on the currently selected target. You can view the actual binding using the **DiskRAID list** command.

Syntax

```
create lun simple [size=<n>] [drives=<n>] [noerr]
create lun stripe [size=<n>] [drives=<n, n> [...]] [stripesize=<n>] [noerr]
create lun raid [size=<n>] [drives=<n, n> [...]] [stripesize=<n>] [noerr]
create lun mirror [size=<n>] [drives=<n, n> [...]] [stripesize=<n>] [noerr]
create lun automagic size=<n> [noerr]
create target name=<name> [iscsiname=<iscsiname>] [noerr]
create tpgroup [noerr]
```

Parameter

simple

Creates a simple LUN.

stripe

Creates a striped LUN.

RAID

Creates a striped LUN with parity.

mirror

Creates a mirrored LUN.

automagic

Creates a LUN using the *automagic* hints currently in effect. See the **automagic** sub-command for more information.

size=

Specifies the total LUN size in megabytes. If the **size=** parameter is not specified, the LUN created will be the largest possible size allowed by all the specified drives.

A provider typically creates a LUN at least as big as the requested size, but the provider may have to round up to the next largest size in some cases. For example, if size is specified as .99 GB and the provider can only allocate GB disk extents, the resulting LUN would be 1 GB.

To specify the size using other units, use one of the following recognized suffixes immediately after the size:

- **B** for byte.
- **KB** for kilobyte.
- **MB** for megabyte.

- **GB** for gigabyte.
- **TB** for terabyte.
- **PB** for petabyte.

drives=

Specifies the *drive_number* for the drives to use to create a LUN. If the **size=** parameter is not specified, the LUN created is the largest possible size allowed by all the specified drives. If the **size=** parameter is specified, providers will select drives from the specified drive list to create the LUN. Providers will attempt to use the drives in the order specified when possible.

stripesize=

Specifies the size in megabytes for a *stripe* or *RAID* LUN. The stripesize cannot be changed after the LUN is created.

To specify the size using other units, use one of the following recognized suffixes immediately after the size:

- **B** for byte.
- **KB** for kilobyte.
- **MB** for megabyte.
- **GB** for gigabyte.
- **TB** for terabyte.
- **PB** for petabyte.

target

Creates a new iSCSI target on the currently selected subsystem.

name

Supplies the friendly name for the target.

iscsiname

Supplies the iSCSI name for the target and can be omitted to have the provider generate a name.

tpgroup

Creates a new iSCSI target portal group on the currently selected target.

noerr

Specifies that any failures that occur while performing this operation will be ignored. This is useful in script mode.

Remarks

- Either the **size=** or the **drives=** parameter must be specified. They can also be used together.
- The stripe size for a LUN cannot be changed after creation.

delete

Deletes the currently selected LUN, iSCSI target (as long as there are not any LUNs associated with the iSCSI target) or iSCSI target portal group.

Syntax

```
delete lun [uninstall] [noerr]
delete target [noerr]
delete tpgroup [noerr]
```

Parameters

lun

Deletes the currently selected LUN and all data on it.

uninstall

Specifies that the disk on the local system associated with the LUN will be cleaned up before the LUN is deleted.

target

Deletes the currently selected iSCSI target if no LUNs are associated with the target.

tpgroup

Deletes the currently selected iSCSI target portal group.

noerr

Specifies that any failures that occur while performing this operation will be ignored. This is useful in script mode.

detail

Displays detailed information about the currently selected object of the specified type.

Syntax

```
Detail {hbaport | iadapter | iportal | provider | subsystem | controller | port | drive | lun | tportal |  
target | tpgroup} [verbose]
```

Parameters

hbaport

Lists detailed information about the currently selected host bus adapter (HBA) port.

iadapter

Lists detailed information about the currently selected iSCSI initiator adapter.

iportal

Lists detailed information about the currently selected iSCSI initiator portal.

provider

Lists detailed information about the currently selected provider.

subsystem

Lists detailed information about the currently selected subsystem.

controller

Lists detailed information about the currently selected controller.

port

Lists detailed information about the currently selected controller port.

drive

Lists detailed information about the currently selected drive, including the occupying LUNs.

lun

Lists detailed information about the currently selected LUN, including the contributing drives. The output differs

slightly depending on whether the LUN is part of a Fibre Channel or iSCSI subsystem. If the Unmasked Hosts list contains only an asterisk, this means that the LUN is unmasked to all hosts.

tpportal

Lists detailed information about the currently selected iSCSI target portal.

target

Lists detailed information about the currently selected iSCSI target.

tpgroup

Lists detailed information about the currently selected iSCSI target portal group.

verbose

For use only with the LUN parameter. Lists additional information, including its plexes.

dissociate

Sets specified list of controller ports as inactive for the currently selected LUN (other controller ports are not affected), or dissociates the specified list of iSCSI targets for the currently selected LUN.

Syntax

```
dissociate controllers <n> [, <n> [...]]  
dissociate ports <n-m>[, <n-m> [...]]  
dissociate targets <n> [, <n> [...]]
```

Parameter

controllers

For use with VDS 1.0 providers only. Removes controllers from the list of controllers that are associated with the currently selected LUN.

ports

For use with VDS 1.1 providers only. Removes controller ports from the list of controller ports that are associated with the currently selected LUN.

targets

For use with VDS 1.1 providers only. Removes targets from the list of iSCSI targets that are associated with the currently selected LUN.

```
<n> [, <n> [...]]
```

For use with the **controllers** or **targets** parameter. Specifies the numbers of the controllers or iSCSI targets to set as inactive or dissociate.

```
<n-m>[, <n-m> [...]]
```

For use with the **ports** parameter. Specifies the controller ports to set as inactive by using a controller number (*n*) and port number (*m*) pair.

Example

```
DISKRAID> SEL LUN 5
LUN 5 is now the selected LUN.

DISKRAID> ASSOCIATE PORTS 0-0,0-1
Controller port associations changed.
(Controller ports active after this command: Ctlr 0 Port 0, Ctlr 0 Port 1)

DISKRAID> ASSOCIATE PORTS ADD 1-1
Controller port associations changed.
(Controller ports active after this command: Ctlr 0 Port 0, Ctlr 0 Port 1, Ctlr 1 Port 1)

DISKRAID> DISSOCIATE PORTS 0-0,1-1
Controller port associations changed.
(Controller ports active after this command: Ctlr 0 Port 1)
```

exit

Exits DiskRAID.

Syntax

```
exit
```

extend

Extends the currently selected LUN by adding sectors to the end of the LUN. Not all providers support extending LUNs. Does not extend any volumes or file systems contained on the LUN. After you extend the LUN, you should extend the associated on-disk structures using the **DiskPart extend** command.

Syntax

```
extend lun [size=<LUN_size>] [drives=<drive_number>, [<drive_number>, ...]] [noerr]
```

Parameters

size=

Specifies the size in megabytes to extend the LUN. If the **size=** parameter is not specified, the LUN is extended by the largest possible size allowed by all the specified drives. If the **size=** parameter is specified, providers select drives from the list specified by the **drives=** parameter to create the LUN.

To specify the size using other units, use one of the following recognized suffixes immediately after the size:

- **B** for byte.
- **KB** for kilobyte.
- **MB** for megabyte.
- **GB** for gigabyte.
- **TB** for terabyte
- **PB** for petabyte

drives=

Specifies the <drive_number> for the drives to use when creating a LUN. If the **size=** parameter is not specified, the LUN created is the largest possible size allowed by all the specified drives. Providers use the drives in the order specified when possible.

noerr

Specifies that any failures that occur while performing this operation should be ignored. This is useful in script mode.

Remarks

Either the *size* or the <drive> parameter must be specified. They can also be used together.

flushcache

Clears the cache on the currently selected controller.

Syntax

```
flushcache controller
```

help

Displays a list of all DiskRAID commands.

Syntax

```
help
```

importtarget

Retrieves or sets the current Volume Shadow Copy Service (VSS) import target that is set for the currently selected subsystem.

Syntax

```
importtarget subsystem [set target]
```

Parameter

set target

If specified, sets the currently selected target to the VSS import target for the currently selected subsystem. If not specified, the command retrieves the current VSS import target that is set for the currently selected subsystem.

initiator

Retrieves information about the local iSCSI initiator.

Syntax

```
initiator
```

invalidatecache

Invalidates the cache on the currently selected controller.

Syntax

```
invalidatecache controller
```

lbpolicy

Sets the load balance policy on the currently selected LUN.

Syntax

```
lbpolicy set lun type=<type> [paths=<path>-{primary | <weight>}[,<path>-{primary | <weight>}[,...]]  
lbpolicy set lun paths=<path>-{primary | <weight>}[,<path>-{primary | <weight>}[,...]]
```

Parameters

type

Specifies the load balance policy. If the type is not specified, then the **path** parameter must be specified. Type can be one of the following:

FAILOVER: Uses one primary path with other paths being backup paths.

ROUNDROBIN: Uses all paths in round-robin fashion, which tries each path sequentially.

SUBSETROUNDROBIN: Uses all primary paths in round-robin fashion; backup paths are used only if all primary paths fail.

DYNLQD: Uses the path with the least number of active requests.

WEIGHTED: Uses the path with the least weight (each path must be assigned a weight).

LEASTBLOCKS: Uses the path with the least blocks.

VENDORSPECIFIC: Uses a vendor-specific policy.

paths

Specifies whether a path is **primary** or has a particular <weight>. Any paths not specified are implicitly set as backup. Any paths listed must be one of the currently selected LUN's paths.

list

Displays a list of objects of the specified type.

Syntax

```
List {hbaports | iadapters | iportals | providers | subsystems | controllers | ports | drives | LUNs | tportals  
| targets | tpgroups}
```

Parameters

hbaports

Lists summary information about all HBA ports known to VDS. The currently selected HBA port is marked by an asterisk (*).

iadapters

Lists summary information about all iSCSI initiator adapters known to VDS. The currently selected initiator adapter is marked by an asterisk (*).

iportals

Lists summary information about all iSCSI initiator portals in the currently selected initiator adapter. The currently selected initiator portal is marked by an asterisk (*).

providers

Lists summary information about each provider known to VDS. The currently selected provider is marked by an asterisk (*).

subsystems

Lists summary information about each subsystem in the system. The currently selected subsystem is marked by an asterisk (*).

controllers

Lists summary information about each controller in the currently selected subsystem. The currently selected controller is marked by an asterisk (*).

ports

Lists summary information about each controller port in the currently selected controller. The currently selected port is marked by an asterisk (*).

drives

Lists summary information about each drive in the currently selected subsystem. The currently selected drive is marked by an asterisk (*).

luns

Lists summary information about each LUN in the currently selected subsystem. The currently selected LUN is marked by an asterisk (*).

tportals

Lists summary information about all iSCSI target portals in the currently selected subsystem. The currently selected target portal is marked by an asterisk (*).

targets

Lists summary information about all iSCSI targets in the currently selected subsystem. The currently selected target is marked by an asterisk (*).

tpgroups

Lists summary information about all iSCSI target portal groups in the currently selected target. The currently selected portal group is marked by an asterisk (*).

login

Logs the specified iSCSI initiator adapter into the currently selected iSCSI target.

Syntax

```
login target iadapter=<iadapter> [type={manual | persistent | boot}] [chap={none | oneway | mutual}] [iportal=
<iportal>] [tportal=<tportal>] [<flag> [<flag> [...]]]
```

Parameters

type

Specifies the type of login to perform: **manual**, **persistent**, or **boot**. If unspecified, a manual login will be performed.

manual - Login manually.

persistent - Automatically use the same login when the computer is restarted.

boot - (This option is for future development and is not currently used.)

chap

Specifies the type of CHAP authentication to use: **none**, **oneway** CHAP, or **mutual** CHAP; if unspecified, no authentication will be used.

tportal

Specifies an optional target portal in the currently selected subsystem to use for the log in.

iportal

Specifies an optional initiator portal in the specified initiator adapter to use for the log in.

<flag>

Identified by three letter acronyms:

IPS: Require IPsec

EMP: Enable multipath

EHD: Enable header digest

EDD: Enable data digest

logout

Logs the specified iSCSI initiator adapter out of the currently selected iSCSI target.

Syntax

```
logout target iadapter= <iadapter>
```

Parameters

iadapter

Specifies the initiator adapter with a login session to logout from.

maintenance

Performs maintenance operations on the currently selected object of the specified type.

Syntax

```
maintenance <object operation> [count=<iteration>]
```

Parameters

<object>

Specifies the type of object on which to perform the operation. The *object* type can be a **subsystem**, **controller**, **port**, **drive** or **LUN**.

<operation>

Specifies the maintenance operation to perform. The *operation* type can be **spinup**, **spindown**, **blink**, **beep** or **ping**. An *operation* must be specified.

count=

Specifies the number of times to repeat the *operation*. This is typically used with **blink**, **beep**, or **ping**.

name

Sets the friendly name of the currently selected subsystem, LUN, or iSCSI target to the specified name.

Syntax

```
name {subsystem | lun | target} [<name>]
```

Parameter

<name>

Specifies a name for the subsystem, LUN, or target. The name must be less than 64 characters in length. If no name is supplied, the existing name, if any, is deleted.

offline

Sets the state of the currently selected object of the specified type to **offline**.

Syntax

```
offline <object>
```

Parameter

<object>

Specifies the type of object on which to perform this operation. The <object> type can be **subsystem**, **controller**, **drive**, **LUN**, or **tportal**.

online

Sets the state of the selected object of the specified type to **online**. If object is **hbaport**, changes the status of the paths to the currently selected HBA port to **online**.

Syntax

```
online <object>
```

Parameter

<object>

Specifies the type of object on which to perform this operation. The <object> type can be **hbaport**, **subsystem**, **controller**, **drive**, **LUN**, or **tportal**.

recover

Performs operations necessary, such as resynchronization or hot sparing, to repair the currently selected fault-tolerant LUN. For example, RECOVER might cause a hot spare to be bound to a RAID set that has a failed disk or other disk extent reallocation.

Syntax

```
recover <lun>
```

reenumerate

Reenumerates objects of the specified type. If you use the extend LUN command, you must use the refresh command to update the disk size before using the reenumerate command.

Syntax

```
reenumerate {subsystems | drives}
```

Parameters

subsystems

Queries the provider to discover any new subsystems that were added in the currently selected provider.

drives

Queries the internal I/O buses to discover any new drives that were added in the currently selected subsystem.

refresh

Refreshes internal data for the currently selected provider.

Syntax

```
refresh provider
```

rem

Used to comment scripts.

Syntax

```
Rem <comment>
```

remove

Removes the specified iSCSI target portal from the currently selected target portal group.

Syntax

```
remove tpgroup tportal=<tportal> [noerr]
```

Parameter

tpgroup tportal= <tportal>

Specifies the iSCSI target portal to remove.

noerr

Specifies that any failures that occur while performing this operation should be ignored. This is useful in script mode.

replace

Replaces the specified drive with the currently selected drive.

Syntax

```
replace drive=<drive_number>
```

Parameter

drive=

Specifies the <drive_number> for the drive to be replaced.

Remarks

- The specified drive may not be the currently selected drive.

reset

Resets the currently selected controller or port.

Syntax

```
Reset {controller | port}
```

Parameters

controller

Resets the controller.

port

Resets the port.

select

Displays or changes the currently selected object.

Syntax

```
Select {hbaport | iadapter | iportal | provider | subsystem | controller | port | drive | lun | tportal |  
target | tpgroup } [<n>]
```

Parameters

object

Specifies the type of object to select. The <object> type can be **provider**, **subsystem**, **controller**, **drive**, or **LUN**.

hbaport [<n>]

Sets the focus to the specified local HBA port. If no HBA port is specified, the command displays the currently selected HBA port (if any). Specifying an invalid HBA port index results in no in-focus HBA port. Selecting an HBA port deselects any selected initiator adapters and initiator portals.

iaadapter [<n>]

Sets the focus to the specified local iSCSI initiator adapter. If no initiator adapter is specified, the command displays the currently selected initiator adapter (if any). Specifying an invalid initiator adapter index results in no in-focus initiator adapter. Selecting an initiator adapter deselects any selected HBA ports and initiator portals.

iportal [<n>]

Sets the focus to the specified local iSCSI initiator portal within the selected iSCSI initiator adapter. If no initiator portal is specified, the command displays the currently selected initiator portal (if any). Specifying an invalid initiator portal index results in no selected initiator portal.

provider [<n>]

Sets the focus to the specified provider. If no provider is specified, the command displays the currently selected provider (if any). Specifying an invalid provider index results in no in-focus provider.

subsystem [<n>]

Sets the focus to the specified subsystem. If no subsystem is specified, the command displays the subsystem with focus (if any). Specifying an invalid subsystem index results in no in-focus subsystem. Selecting a subsystem implicitly selects its associated provider.

controller [<n>]

Sets the focus to the specified controller within the currently selected subsystem. If no controller is specified, the command displays the currently selected controller (if any). Specifying an invalid controller index results in no in-focus controller. Selecting a controller deselects any selected controller ports, drives, LUNs, target portals, targets, and target portal groups.

port [<n>]

Sets the focus to the specified controller port within the currently selected controller. If no port is specified, the command displays the currently selected port (if any). Specifying an invalid port index results in no selected port.

drive [<n>]

Sets the focus to the specified drive, or physical spindle, within the currently selected subsystem. If no drive is specified, the command displays the currently selected drive (if any). Specifying an invalid drive index results in no in-focus drive. Selecting a drive deselects any selected controllers, controller ports, LUNs, target portals, targets, and target portal groups.

lun [<n>]

Sets the focus to the specified LUN within the currently selected subsystem. If no LUN is specified, the command displays the currently selected LUN (if any). Specifying an invalid LUN index results in no selected LUN. Selecting a LUN deselects any selected controllers, controller ports, drives, target portals, targets, and target portal groups.

tportal [<n>]

Sets the focus to the specified iSCSI target portal within the currently selected subsystem. If no target portal is specified, the command displays the currently selected target portal (if any). Specifying an invalid target portal index results in no selected target portal. Selecting a target portal deselects any controllers, controller ports, drives, LUNs, targets, and target portal groups.

target [<n>]

Sets the focus to the specified iSCSI target within the currently selected subsystem. If no target is specified, the command displays the currently selected target (if any). Specifying an invalid target index results in no selected target. Selecting a target deselects any controllers, controller ports, drives, LUNs, target portals, and target portal groups.

tpgroup [<n>]

Sets the focus to the specified iSCSI target portal group within the currently selected iSCSI target. If no target portal group is specified, the command displays the currently selected target portal group (if any). Specifying an invalid target portal group index results in no in-focus target portal group.

[<n>]

Specifies the <object number> to select. If the specified is not valid, any existing selections for objects of the specified type are cleared. If no is specified, the current object is displayed.

setflag

Sets the currently selected drive as a hot spare.

Syntax

```
setflag drive hotspare={true | false}
```

Parameters

true

Selects the currently selected drive as a hot spare.

false

Unselects the currently selected drive as a hot spare.

Remarks

Hot spares cannot be used for ordinary LUN binding operations. They are reserved for fault handling only. The drive must not be currently bound to any existing LUN.

shrink

Reduces the size of the selected LUN.

Syntax

```
shrink lun size=<n> [noerr]
```

Parameters

size=

Specifies the desired amount of space in megabytes (MB) to reduce the size of the LUN by. To specify the size using other units, use one of the recognized suffixes (B, KB, MB, GB, TB and PB) immediately after the size.

noerr

Specifies that any failures that occur while performing this operation will be ignored. This is useful in script mode.

standby

Changes the status of the paths to the currently selected host bus adapter (HBA) port to STANDBY.

Syntax

```
standby hbaport
```

Parameters**hbaport**

Changes the status of the paths to the currently selected host bus adapter (HBA) port to STANDBY.

unmask

Makes the currently selected LUNs accessible from the specified hosts.

Syntax

```
unmask LUN {all | none | [add] wwn=<hexadecimal_number> [;<hexadecimal_number> [;...]] | [add] initiator=
<initiator>[;<initiator>[;...]]} [uninstall]
```

Parameters**all**

Specifies that the LUN should be made accessible from all hosts. However, you cannot unmask the LUN to all targets in an iSCSI subsystem.

IMPORTANT

You must logout of the target before you run the UNMASK ALL command.

none

Specifies that the LUN should not be accessible to any host.

IMPORTANT

You must logout of the target before you run the UNMASK LUN NONE command.

add

Specifies that the hosts specified must be added to the existing list of hosts that this LUN is accessible from. If this parameter is not specified, the list of hosts supplied replaces the existing list of hosts that this LUN is accessible from.

WWN =

Specifies a list of hexadecimal numbers representing world-wide names from which the LUN or hosts should

be made accessible. To mask/unmask to a specific set of hosts in a Fibre Channel subsystem, you can type a semicolon-separated list of WWN's for the ports on the host machines of interest.

initiator=

Specifies a list of iSCSI initiators to which the currently selected LUN should be made accessible. To mask/unmask to a specific set of hosts in an iSCSI subsystem, you can type a semicolon-separated list of iSCSI initiator names for the initiators on the host computers of interest.

uninstall

If specified, uninstalls the disk associated with the LUN on the local system before the LUN is masked.

Scripting DiskRAID

DiskRAID can be scripted on any computer running Windows Server 2008 or Windows Server 2003 with an associated VDS hardware provider. To invoke a DiskRAID script, at the command prompt type:

```
diskraid /s <script.txt>
```

By default, DiskRAID stops processing commands and returns an error code if there is a problem in the script. To continue running the script and ignore errors, include the NOERR parameter on the command. This permits such useful practices as using a single script to delete all the LUNs in a subsystem regardless of the total number of LUNs. Not all commands support the NOERR parameter. Errors are always returned on command-syntax errors, regardless of whether you included the NOERR parameter,

DiskRAID error codes

| ERROR CODE | ERROR DESCRIPTION |
|------------|--|
| 0 | No error occurred. The entire script ran without failure. |
| 1 | A fatal exception occurred. |
| 2 | The arguments specified on a DiskRAID command line were incorrect. |
| 3 | DiskRAID was unable to open the specified script or output file. |
| 4 | One of the services DiskRAID uses returned a failure. |
| 5 | A command syntax error occurred. The script failed because an object was improperly selected or was invalid for use with that command. |

Example: Interactively View Status of Subsystem

If you want to view the status of subsystem 0 on your computer, type the following at the command line:

```
diskraid
```

Press ENTER. The following is displayed:

```
Microsoft Diskraid version 5.2.xxxx
Copyright (©) 2003 Microsoft Corporation
On computer: COMPUTER_NAME
```

To select subsystem 0, type the following at the DiskRAID prompt:

```
select subsystem 0
```

Press ENTER. Output similar to the following is displayed:

```
Subsystem 0 is now the selected subsystem.
```

```
DISKRAID> list drives
```

| Drive ### | Status | Health | Size | Free | Bus | Slot | Flags |
|-----------|-----------|---------|--------|--------|-----|------|-------|
| ----- | ----- | ----- | ----- | ----- | --- | ---- | ----- |
| Drive 0 | Online | Healthy | 107 GB | 107 GB | 0 | 1 | |
| Drive 1 | Offline | Healthy | 29 GB | 29 GB | 1 | 0 | |
| Drive 2 | Online | Healthy | 107 GB | 107 GB | 0 | 2 | |
| Drive 3 | Not Ready | Healthy | 19 GB | 19 GB | 1 | 1 | |

To exit DiskRAID, type the following at the DiskRAID prompt:

```
exit
```

diskshadow

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

diskshadow.exe is a tool that exposes the functionality offered by the volume shadow copy Service (VSS). By default, diskshadow uses an interactive command interpreter similar to that of diskraid or DiskPart. diskshadow also includes a scriptable mode.

NOTE

Membership in the local Administrators group, or equivalent, is the minimum required to run diskshadow.

for examples of how to use diskshadow commands, see [Examples](#).

Syntax

for interactive mode, type the following at the command prompt to start the diskshadow command interpreter:

```
diskshadow
```

for script mode, type the following, where *script.txt* is a script file containing diskshadow commands:

```
diskshadow -s script.txt
```

diskshadow commands

You can run the following commands in the diskshadow command interpreter or through a script file:

| PARAMETER | DESCRIPTION |
|----------------------------------|---|
| set_2 | Sets the context, options, verbose mode, and metadata file for creating shadow copies. |
| Simulate restore | Tests writer involvement in restore sessions on the computer without issuing PreRestore or PostRestore events to writers. |
| Load metadata | Loads a metadata .cab file prior to importing a transportable shadow copy or loads the writer metadata in the case of a restore. |
| writer | verifies that a writer or component is included or excludes a writer or component from the backup or restore procedure. |
| add_1 | adds volumes to the set of volumes that are to be shadow copied, or adds aliases to the alias environment. |

| PARAMETER | DESCRIPTION |
|-----------------------------|---|
| <code>create_1</code> | starts the shadow copy creation process, using the current context and option settings. |
| <code>exec</code> | executes a file on the local computer. |
| <code>Begin backup</code> | starts a full backup session. |
| <code>End backup</code> | Ends a full backup session and issues a Backupcomplete event with the appropriate writer state, if needed. |
| <code>Begin restore</code> | starts a restore session and issues a PreRestore event to involved writers. |
| <code>End restore</code> | Ends a restore session and issues a PostRestore event to involved writers. |
| <code>reset</code> | resets diskshadow to the default state. |
| <code>list</code> | lists writers, shadow copies, or currently registered shadow copy providers that are on the system. |
| <code>delete shadows</code> | deletes shadow copies. |
| <code>import</code> | imports a transportable shadow copy from a loaded metadata file into the system. |
| <code>mask</code> | removes hardware shadow copies that were imported by using the import command. |
| <code>expose</code> | exposes a persistent shadow copy as a drive letter, share, or mount point. |
| <code>unexpose</code> | unexposes a shadow copy that was exposed by using the expose command. |
| <code>break_2</code> | Disassociates a shadow copy volume from VSS. |
| <code>revert</code> | reverts a volume back to a specified shadow copy. |
| <code>exit_1</code> | exits diskshadow. |

Remarks

- at a minimum, only **add** and **create** are necessary to create a shadow copy. However, this will forfeit the context and option settings, will be a copy backup, and will only create a shadow copy with no backup execution script.

Examples

This is a sample sequence of commands that will create a shadow copy for backup. It can be saved to file as `script.dsh`, and executed with `diskshadow /s script.dsh`

Assume the following:

- You have an existing directory called c:\diskshadowdata.
- Your system volume is C: and your data volume is D:.
- You have a backupscript.cmd file in c:\diskshadowdata.
- Your backupscript.cmd file will perform the copy of shadow data p: and q: to your backup drive.

You can enter these commands manually or script them:

```
#diskshadow script file
set context persistent nowriters
set metadata c:\diskshadowdata\example.cab
set verbose on
begin backup
add volume c: alias Systemvolumeshadow
add volume d: alias Datavolumeshadow

create

expose %Systemvolumeshadow% p:
expose %Datavolumeshadow% q:
exec c:\diskshadowdata\backupscript.cmd
end backup
#End of script
```

additional references

[Command-Line Syntax Key](#)

dispdiag

4/13/2018 • 2 minutes to read • [Edit Online](#)

Logs display information to a file.

Syntax

```
dispdiag [-testacpi] [-d] [-delay <Seconds>] [-out <FilePath>]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------------|--|
| - testacpi | Runs hotkey diagnostics test. Displays the key name, code and scan code for any key pressed during the test. |
| -d | Generates a dump file with test results. |
| -delay <Seconds> | Delays the collection of data by specified time in <i>seconds</i> . |
| -out <FilePath> | Specifies path and filename to save collected data. This must be the last parameter. |
| -? | Displays available command parameters and provides help for using them. |

Dnscmd

7/11/2018 • 40 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

A command-line interface for managing DNS servers. This utility is useful in scripting batch files to help automate routine DNS management tasks, or to perform simple unattended setup and configuration of new DNS servers on your network.

Syntax

```
dnscmd <ServerName> <command> [<command parameters>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| | The IP address or host name of a remote or local DNS server. |

Commands

| COMMAND | DESCRIPTION |
|--|--|
| dnscmd /ageallrecords | Sets the current time on all time stamps in a zone or node. |
| dnscmd /clearcache | Clears the DNS server cache. |
| dnscmd /config | resets the DNS server or zone configuration. |
| dnscmd /createbuiltindirectorypartitions | creates the built-in DNS application directory partitions. |
| dnscmd /createdirectorypartition | creates a DNS application directory partition. |
| dnscmd /deletedirectorypartition | deletes a DNS application directory partition. |
| dnscmd /directorypartitioninfo | lists information about a DNS application directory partition. |
| dnscmd /enlistdirectorypartition | adds a DNS server to the replication set of a DNS application directory partition. |
| dnscmd /enumdirectorypartitions | lists the DNS application directory partitions for a server. |
| dnscmd /enumrecords | lists the resource records in a zone. |
| dnscmd /enumzones | lists the zones hosted by the specified server. |

| COMMAND | DESCRIPTION |
|---|---|
| <code>dnscmd /exportsettings</code> | Writes server configuration information to a text file. |
| <code>dnscmd /info</code> | Gets server information. |
| <code>dnscmd /ipvalidate</code> | Validates remote DNS servers. |
| <code>dnscmd /nodedelete</code> | deletes all records for a node in a zone. |
| <code>dnscmd /recordadd</code> | adds a resource record to a zone. |
| <code>dnscmd /recorddelete</code> | removes a resource record from a zone. |
| <code>dnscmd /resetforwarders</code> | Sets DNS servers to forward recursive queries. |
| <code>dnscmd /resetlistenaddresses</code> | Sets server IP addresses to serve DNS requests. |
| <code>dnscmd /startscavenging</code> | Initiates server scavenging. |
| <code>dnscmd /statistics</code> | Queries or clears server statistics data. |
| <code>dnscmd /unenlistdirectorypartition</code> | removes a DNS server from the replication set of a DNS application directory partition. |
| <code>dnscmd /writebackfiles</code> | Saves all zone or root-hint data to a file. |
| <code>dnscmd /zoneadd</code> | creates a new zone on the DNS server. |
| <code>dnscmd /zonechangedirectorypartition</code> | changes the directory partition on which a zone resides. |
| <code>dnscmd /zonedelelete</code> | deletes a zone from the DNS server. |
| <code>dnscmd /zoneexport</code> | Writes the resource records of a zone to a text file. |
| <code>dnscmd /zoneinfo</code> | Displays zone information. |
| <code>dnscmd /zonepause</code> | pauses a zone. |
| <code>dnscmd /zoneprint</code> | Displays all records in the zone. |
| <code>dnscmd /zonerefresh</code> | forces a refresh of the secondary zone from the master zone. |
| <code>dnscmd /zonereload</code> | Reloads a zone from its database. |
| <code>dnscmd /zoneresetmasters</code> | changes the master servers that provide zone transfer information to a secondary zone. |
| <code>dnscmd /zoneresetscavengeservers</code> | changes the servers that can scavenge a zone. |
| <code>dnscmd /zoneresetsecondaries</code> | resets secondary information for a zone. |

| COMMAND | DESCRIPTION |
|--|--|
| dnscmd /zoneresettype | changes the zone type. |
| dnscmd /zonerestart | Resumes a zone. |
| dnscmd /zoneupdatefromds | Updates an active directory integrated zone with data from active directory Domain Services (AD DS). |
| dnscmd /zonewriteback | Saves zone data to a file. |

dnscmd /ageallrecords

Sets the current time on a time stamp on resource records at a specified zone or node on a DNS server.

Syntax

```
dnscmd [<ServerName>] /ageallrecords <ZoneName>[<NodeName>] | [/tree][ /f]
```

Parameters

Specifies the DNS server that the administrator plans to manage, represented by IP address, fully qualified domain name (FQDN), or Host name. If this parameter is omitted, the local server is used.

Specifies the FQDN of the zone.

Specifies a specific node or subtree in the zone. *NodeName* specifies the node or subtree in the zone using the following:

- @ for root zone or FQDN
- The FQDN of a node (the name with a period (.) at the end)
- A single label for the name relative to the zone root
 - /tree
Specifies that all child nodes also receive the time stamp.
 - /f
Runs the command without asking for confirmation.
 - #### Remarks
- The **ageallrecords** command is for backward compatibility between the current version of DNS and previous releases of DNS in which aging and scavenging were not supported. It adds a time stamp with the current time to resource records that do not have a time stamp, and it sets the current time on resource records that do have a time stamp.
- Record scavenging does not occur unless the records are time stamped. Name server (NS) resource records, start of authority (SOA) resource records, and Windows Internet Name Service (WINS) resource records are not included in the scavenging process, and they are not time stamped even when the **ageallrecords** command runs.
- This command fails unless scavenging is enabled for the DNS server and the zone. For information about how to enable scavenging for the zone, see the **aging** parameter under Zone-Level Syntax in the [config](#) command.
- The addition of a time stamp to DNS resource records makes them incompatible with DNS servers that run on operating systems other than Windows 2000, Windows XP, or Windows Server 2003. A time stamp that you add by using the **ageallrecords** command cannot be reversed.
- if none of the optional parameters are specified, the command returns all resource records at the specified node. If a value is specified for at least one of the optional parameters, **dnscmd** enumerates only the resource records that correspond to the value or values that are specified in the optional parameter or parameters.

Example

See [Example 1: Set the current time on a time stamp to resource records.](#)

dnscmd /clearcache

Clears the DNS cache memory of resource records on the specified DNS server.

Syntax

```
dnscmd [<ServerName>] /clearcache
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

Sample usage

```
dnscmd dnssvr1.contoso.com /clearcache
```

dnscmd /config

changes values in the registry for the DNS server and individual zones. Accepts server-level settings and zone-level settings.

> [!CAUTION]

> Do not edit the registry directly unless you have no alternative. The registry editor bypasses standard safeguards, allowing settings that can degrade performance, damage your system, or even require you to reinstall Windows. You can safely alter most registry settings by using the programs in Control Panel or Microsoft Management Console (mmc). If you must edit the registry directly, back it up first. Read the registry editor help for more information.

Server-level syntax

```
dnscmd [<ServerName>] /config <Parameter>
```

dnscmd /config

Modifies the configuration of the specified server.

Parameters

Specifies the DNS server that you are planning to manage, represented by local computer syntax, IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

Specify a setting and, as an option, a value. Parameter values use this syntax: *Parameter* [Value]

The following parameter values are described in the remainder of this section:

- **/addressanswerlimit**
- **/bindsecondaries**
- **/bootmethod**
- **/defaulttagingstate**
- **/defaultnorefreshinterval**
- **/defaultrefreshinterval**
- **/disableautoreversezones**
- **/disablensrecordsautocreation**
- **/dspollinginterval**
- **/dstombstoneinterval**
- **/ednscachetimeout**
- **/enablednsprobes**
- **/enablednssec**
- **/enableglobalnamesupport**
- **/enableglobalqueryblocklist**
- **/eventloglevel**
- **/forwarddelegations**
- **/forwardingtimeout**

- **/globalnamesqueryorder**
- **/globalqueryblocklist**
- **/isslave**
- **/localnetpriority**
- **/logfilemaxsize**
- **/logfilepath**
- **/logipfilterlist**
- **/loglevel**
- **/maxcachesize**
- **/maxcachettl**
- **/namecheckflag**
- **/notcp**
- **/norecursion**
- **/recursionretry**
- **/recursiontimeout**
- **/roundrobin**
- **/rpcprotocol**
- **/scavenginginterval**
- **/secureresponses**
- **/sendport**
- **/strictfileparsing**
- **/updateoptions**
- **/writeauthorityns**
- **/xfrconnecttimeout**

/addressanswerlimit [0|5-28]

Specifies the maximum number of host records that a DNS server can send in response to a query. The value can be zero (0), or it can be in the range of 5 through 28 records. The default value is zero (0).

/bindsecondaries[0|1]

changes the format of the zone transfer so that it can achieve maximum compression and efficiency. However, this format is not compatible with earlier versions of Berkeley Internet Name Domain (BIND).

0

Uses maximum compression. This format is compatible with BIND versions 4.9.4 and later only.

1

Sends only one resource record per message to non-Microsoft DNS servers. This format is compatible with BIND versions earlier than 4.9.4. This is the default setting.

/bootmethod[0|1|2|3]

Determines the source from which the DNS server gets its configuration information.

0

Clears the source of configuration information.

1

Loads from the BIND file that is located in the DNS directory, which is %systemroot%\System32\DNS by default.

2

Loads from the registry.

3

Loads from AD DS and the registry. This is the default setting.

/defaulttagingstate[0|1]

Determines whether the DNS scavenging feature is enabled by default on newly created zones.

0

Disables scavenging. This is the default setting.

1

Enables scavenging.

/defaultnorefreshinterval[0x1-0xFFFFFFFF|0xA8]

Sets a period of time in which no refreshes are accepted for dynamically updated records. Zones on the server inherit this value automatically. To change the default value, type a value in the range of **0x1-0xFFFFFFFF**. The default value from the server is **0xA8**.

/defaultrefreshinterval [0x1-0xFFFFFFFF|0xA8]

Sets a period of time that is allowed for dynamic updates to DNS records. Zones on the server inherit this value automatically. To change the default value, type a value in the range of **0x1-0xFFFFFFFF**. The default value from the server is **0xA8**.

/disableautoreversezones {0|1}

Enables or disables the automatic creation of reverse lookup zones. Reverse lookup zones provide resolution of Internet Protocol (IP) addresses to DNS domain names.

0

Enables the automatic creation of reverse lookup zones. This is the default setting.

1

Disables the automatic creation of reverse lookup zones.

/disablensrecordsautocreation {0|1}

Specifies whether the DNS server automatically creates name server (NS) resource records for zones that it hosts.

0

Automatically creates name server (NS) resource records for zones that the DNS server hosts.

1

Does not automatically create name server (NS) resource records for zones that the DNS server hosts.

/dspollinginterval 0-30

Specifies how often the DNS server polls AD DS for changes in active directory integrated zones.

/dstombstoneinterval [1-30]

The amount of time in seconds to retain deleted records in AD DS.

/ednscachetimeout []

Specifies the number of seconds that extended DNS (EDNS) information is cached. The minimum value is 3600, and the maximum value is 15,724,800. The default value is 604,800 seconds (one week).

/enableednsprobes {0|1}

Enables or disables the server to probe other servers to determine if they support EDNS.

0

Disables active support for EDNS probes.

1

Enables active support for EDNS probes.

/enablednssec {0|1}

Enables or disables support for DNS Security Extensions (DNSSEC).

0

Disables DNSSEC.

1

Enables DNSSEC.

/enableglobalnamesupport {0|1}

Enables or disables support for the GlobalNames zone. The GlobalNames zone supports resolution of single-label DNS names across a forest.

0

Disables support for the GlobalNames zone. When you set the value of this command to **0**, the DNS Server service does not resolve single-label names in the GlobalNames zone.

1

Enables support for the GlobalNames zone. When you set the value of this command to **1**, the DNS Server

service resolves single-label names in the GlobalNames zone.

/enableglobalqueryblocklist {0|1}

Enables or disables support for the global query block list that blocks name resolution for names in the list. The DNS Server service creates and enables the global query block list by default when the service starts the first time. To view the current global query block list, use the **dnscmd /info /globalqueryblocklist** command.

0

Disables support for the global query block list. When you set the value of this command to **0**, the DNS Server service responds to queries for names in the block list.

1

Enables support for the global query block list. When you set the value of this command to **1**, the DNS Server service does not respond to queries for names in the block list.

/eventloglevel [0|1|2|4]

Determines which events are logged in the DNS server log in Event Viewer.

0

Logs no events.

1

Logs only errors.

2

Logs only errors and warnings.

4

Logs errors, warnings, and informational events. This is the default setting.

/forwarddelegations [0|1]

Determines how the DNS server handles a query for a delegated subzone. These queries can be sent either to the subzone that is referred to in the query or to the list of forwarders that is named for the DNS server. Entries in the setting are used only when forwarding is enabled.

0

Automatically sends queries that refer to delegated subzones to the appropriate subzone. This is the default setting.

1

forwards queries that refer to the delegated subzone to the existing forwarders.

/forwardingtimeout []

Determines how many seconds (0x1-0xFFFFFFFF) a DNS server waits for a forwarder to respond before trying another forwarder. The default value is **0x5**, which is 5 seconds.

/globalneamesqueryorder {0|1}

Specifies whether the DNS Server service looks first in the GlobalNames zone or local zones when it resolves names.

0

The DNS Server service attempts to resolve names by querying the GlobalNames zone before it queries the zones for which it is authoritative.

1

The DNS Server service attempts to resolve names by querying the zones for which it is authoritative before it queries the GlobalNames zone.

/globalqueryblocklist[[]...]

replaces the current global query block list with a list of the names that you specify. If you do not specify any names, this command clears the block list. By default, the global query block list contains the following items:

- isatap
- wpad

The DNS Server service can remove either or both of these names when it starts the first time, if it finds these names in an existing zone.

/isslave [0|1]

Determines how the DNS server responds when queries that it forwards receive no response.

0

Specifies that the DNS server is not a subordinate (also known as a *slave*). If the forwarder does not respond, the DNS server attempts to resolve the query itself. This is the default setting.

1

Specifies that the DNS server is a subordinate. If the forwarder does not respond, the DNS server terminates the search and sends a failure message to the resolver.

/localnetpriority [0|1]

Determines the order in which host records are returned when the DNS server has multiple host records for the same name.

0

Returns the records in the order in which they are listed in the DNS database.

1

Returns the records that have similar IP network addresses first. This is the default setting.

/logfilemaxsize []

Specifies the maximum size in bytes (0x10000-0xFFFFFFFF) of the Dns.log file. When the file reaches its maximum size, DNS overwrites the oldest events. The default size is 0x400000, which is 4 megabytes (MB).

/logfilepath [<path+LogFileName>]

Specifies the path of the Dns.log file. The default path is %systemroot%\System32\Dns\Dns.log. You can specify a different path by using the format *path+LogFileName*.

/logipfilterlist [...]

Specifies which packets are logged in the debug log file. The entries are a list of IP addresses. Only packets going to and from the IP addresses in the list are logged.

/loglevel []

Determines which types of events are recorded in the Dns.log file. Each event type is represented by a hexadecimal number. If you want more than one event in the log, use hexadecimal addition to add the values, and then enter the sum.

0x0

The DNS server does not create a log. This is the default entry.

0x10

Logs queries.

0x10

Logs notifications.

0x20

Logs updates.

0xFE

Logs nonquery transactions.

0x100

Logs question transactions.

0x200

Logs answers.

0x1000

Logs send packets.

0x2000

Logs receive packets.

0x4000

Logs User Datagram Protocol (UDP) packets.

0x8000

Logs Transmission Control Protocol (TCP) packets.

0xFFFF

Logs all packets.

0x10000

Logs active directory write transactions.

0x20000

Logs active directory update transactions.

0x1000000

Logs full packets.

0x80000000

Logs write-through transactions.

/maxcachesize

Specifies the maximum size, in kilobytes (KB), of the DNS server's memory cache.

/maxcachettl []

Determines how many seconds (0x0-0xFFFFFFFF) a record is saved in the cache. If the **0x0** setting is used, the DNS server does not cache records. The default setting is **0x15180** (86,400 seconds or 1 day).

/maxnegativecachettl []

Specifies how many seconds (0x1-0xFFFFFFFF) an entry that records a negative answer to a query remains stored in the DNS cache. The default setting is **0x384** (900 seconds).

/namecheckflag [0|1|2|3]

Specifies which character standard is used when checking DNS names.

0

Uses ANSI characters that comply with Internet Engineering Task force (IETF) Request for Comments (Rfcs).

1

Uses ANSI characters that do not necessarily comply with IETF Rfcs.

2

Uses multibyte UCS Transformation format 8 (UTF-8) characters. This is the default setting.

3

Uses all characters.

/norecursion [0|1]

Determines whether a DNS server performs recursive name resolution.

0

The DNS server performs recursive name resolution if it is requested in a query. This is the default setting.

1

The DNS server does not perform recursive name resolution.

/notcp

This parameter is obsolete, and it has no effect in current versions of Windows Server.

/recursionretry []

Determines the number of seconds (0x1-0xFFFFFFFF) that a DNS server waits before again trying to contact a remote server. The default setting is 0x3 (three seconds). This value should be increased when recursion occurs over a slow wide area network (WAN) link.

/recursiontimeout []

Determines the number of seconds (0x1-0xFFFFFFFF) that a DNS server waits before discontinuing attempts to contact a remote server. The settings range from **0x1** through **0xFFFFFFFF**. The default setting is **0xF** (15 seconds). This value should be increased when recursion occurs over a slow WAN link.

/roundrobin [0|1]

Determines the order in which host records are returned when a server has multiple host records for the same name.

0

The DNS server does not use round robin. Instead, it returns the first record to every query.

1

The DNS server rotates among the records that it returns from the top to the bottom of the list of matching records. This is the default setting.

/rpcprotocol [0x0|0x1|0x2|0x4|0xFFFFFFFF]

Specifies the protocol that remote procedure call (RPC) uses when it makes a connection from the DNS server.

0x0

Disables RPC for DNS.

0x1

Uses TCP/IP.

0x2

Uses named pipes.

0x4

Uses local procedure call (LPC).

0xFFFFFFFF

All protocols. This is the default setting.

/scavenginginterval []

Determines whether the scavenging feature for the DNS server is enabled, and sets the number of hours (0x0-0xFFFFFFFF) between scavenging cycles. The default setting is **0x0**, which disables scavenging for the DNS server. A setting greater than **0x0** enables scavenging for the server and sets the number of hours between scavenging cycles.

/secureresponses [0|1]

Determines whether DNS filters records that are saved in a cache.

0

Saves all responses to name queries to a cache. This is the default setting.

1

Saves only the records that belong to the same DNS subtree to a cache.

/sendport []

Specifies the port number (0x0-0xFFFFFFFF) that DNS uses to send recursive queries to other DNS servers. The default setting is **0x0**, which means that the port number is selected randomly.

/serverlevelplugindll[]

Specifies the path of a custom plug-in. When *Dllpath* specifies the fully qualified path name of a valid DNS server plug-in, the DNS server calls functions in the plug-in to resolve name queries that are outside the scope of all locally hosted zones. If a queried name is out of the scope of the plug-in, the DNS server performs name resolution using forwarding or recursion, as configured. If *Dllpath* is not specified, the DNS server ceases to use a custom plug-in if a custom plug-in was previously configured.

/strictfileparsing [0|1]

Determines a DNS server's behavior when it encounters an erroneous record while loading a zone.

0

The DNS server continues to load the zone even if the server encounters an erroneous record. The error is recorded in the DNS log. This is the default setting.

1

The DNS server stops loading the zone, and it records the error in the DNS log.

/updateoptions

Prohibits dynamic updates of specified types of records. If you want more than one record type to be prohibited in the log, use hexadecimal addition to add the values, and then enter the sum.

0x0

Does not restrict any record types.

0x1

Excludes start of authority (SOA) resource records.

0x2

Excludes name server (NS) resource records.

0x4

Excludes delegation of name server (NS) resource records.

0x8

Excludes server host records.

0x100

During secure dynamic update, excludes start of authority (SOA) resource records.

0x200

During secure dynamic update, excludes root name server (NS) resource records.

0x30F

During standard dynamic update, excludes name server (NS) resource records, start of authority (SOA) resource records, and server host records. During secure dynamic update, excludes root name server (NS) resource records and start of authority (SOA) resource records. Allows delegations and server host updates.

0x400

During secure dynamic update, excludes delegation name server (NS) resource records.

0x800

During secure dynamic update, excludes server host records.

0x1000000

Excludes delegation signer (DS) records.

0x80000000

Disables DNS dynamic update.

/writeauthorityns [0|1]

Determines when the DNS server writes name server (NS) resource records in the Authority section of a response.

0

Writes name server (NS) resource records in the Authority section of referrals only. This setting complies with Rfc 1034, Domain names concepts and facilities, and with Rfc 2181, Clarifications to the DNS Specification. This is the default setting.

1

Writes name server (NS) resource records in the Authority section of all successful authoritative responses.

/xfrconnecttimeout []

Determines the number of seconds (0x0-0xFFFFFFFF) a primary DNS server waits for a transfer response from its secondary server. The default value is **0x1E** (30 seconds). After the time-out value expires, the connection is terminated.

Zone-level syntax

```
dnscmd /config <Parameters>
```

dnscmd /config

Modifies the configuration of the specified zone.

Parameters

Specify a setting, a zone name, and, as an option, a value. Parameter values use this syntax: *ZoneName Parameter [Value]*

The following parameter values are documented in the remainder of this section:

- **/aging**
- **/allowsrecordsautocreation**
- **/allowupdate**
- **/forwarderslave**
- **/forwardertimeout**
- **/norefreshinterval**
- **/refreshinterval**
- **/securesecondaries**

/aging

Enables or disables scavenging in a specific zone.

/allowsrecordsautocreation []

Overrides the DNS server's name server (NS) resource record autocreation setting. Name server (NS) resource records that were previously registered for this zone are not affected. Therefore, you must remove them manually if you do not want them.

/allowupdate

Determines whether the specified zone accepts dynamic updates.

/forwarderslave

Overrides the DNS server **/isslave** setting.

/forwardtimeout

Determines how many seconds a DNS zone waits for a forwarder to respond before trying another forwarder. This value overrides the value that is set at the server level.

/norefreshinterval

Sets a time interval for a zone during which no refreshes can dynamically update DNS records in a specified zone.

/refreshinterval

Sets a time interval for a zone during which refreshes can dynamically update DNS records in a specified zone.

/securesecondaries

Determines which secondary servers can receive zone updates from the master server for this zone.

Remarks

- The zone name must be specified only for zone-level parameters.

dnscmd /createbuiltindirectorypartitions

creates a DNS application directory partition. When DNS is installed, an application directory partition for the service is created at the forest and domain levels. Use this command to create DNS application directory partitions that were deleted or never created. With no parameter, this command creates a built-in DNS directory partition for the domain.

Syntax

```
dnscmd [<ServerName>] /createbuiltindirectorypartitions [/forest] [/alldomains]
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

/forest

creates a DNS directory partition for the forest.

/alldomains

creates DNS partitions for all domains in the forest.

dnscmd /createdirectorypartition

creates a DNS application directory partition. When DNS is installed, an application directory partition for the service is created at the forest and domain levels. This operation creates additional DNS application directory partitions.

Syntax

```
dnscmd [<ServerName>] /createdirectorypartition <PartitionFQDN>
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

The FQDN of the DNS application directory partition that will be created.

dnscmd /deletedirectorypartition

removes an existing DNS application directory partition.

Syntax

```
dnscmd [<ServerName>] /deletedirectorypartition <PartitionFQDN>
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

The FQDN of the DNS application directory partition that will be removed.

dnscmd /directorypartitioninfo

lists information about a specified DNS application directory partition.

Syntax

```
dnscmd [<ServerName>] /directorypartitioninfo <PartitionFQDN> [/detail]
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

The FQDN of the DNS application directory partition.

/detail

lists all information about the application directory partition.

dnscmd /enlistdirectorypartition

adds the DNS server to the specified directory partition's replica set.

Syntax

```
dnscmd [<ServerName>] /enlistdirectorypartition <PartitionFQDN>
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

The FQDN of the DNS application directory partition.

dnscmd /enumdirectorypartitions

lists the DNS application directory partitions for the specified server.

Syntax

```
dnscmd [<ServerName>] /enumdirectorypartitions [/custom]
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

/custom

lists only user-created directory partitions.

dnscmd /enumrecords

lists the resource records of a specified node in a DNS zone.

Syntax

```
dnscmd [<ServerName>] /enumrecords <ZoneName> <NodeName> [/type <RRtype> <Rrdata>] [/authority] [/glue] [/additional] [/node | /child | /startchild<ChildName>] [/continue | /detail]
```

Parameters

Specifies the DNS server that you plan to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

/enumrecords

lists resource records in the specified zone.

Specifies the name of the zone to which the resource records belong.

Specifies the name of the node of the resource records.

/type

Specifies the type of resource records to be listed and the type of data that is expected:

Specifies the type of resource records to be listed.

Specifies the type of data that is expected record.

/authority

Includes authoritative data.

/glue

Includes glue data.

/additional

Includes all additional information about the listed resource records.

{/node | /child | /startchild }

Filters or adds information to the resource record display:

/node

lists only the resource records of the specified node.

/child

lists only the resource records of a specified child domain.

/startchild

Begins the list at the specified child domain.

/continue | /detail

Specifies how the returned data is displayed.

/continue

lists only the resource records with their type and data.

/detail

lists all information about the resource records.

Sample usage

```
dnscmd /enumrecords test.contoso.com test /additional
```

dnscmd /enumzones

lists the zones that exist on the specified DNS server.

Syntax

```
dnscmd [<ServerName>] /enumzones [/primary | /secondary | /forwarder | /stub | /cache | /auto-created]
[/forward | /reverse | /ds | /file] [/domaindirectorypartition | /forestdirectorypartition |
/customdirectorypartition | /legacydirectorypartition | /directorypartition <PartitionFQDN>]
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

/primary | /secondary | /forwarder | /stub | /cache | /auto-created

Filters the types of zones to display:

/primary

lists all zones that are either standard primary zones or active directory integrated zones.

/secondary

lists all standard secondary zones.

/forwarder

lists zones that forward unresolved queries to another DNS server.

/stub

lists all stub zones.

/cache

lists only the zones that are loaded into the cache.

/auto-created

lists the zones that were created automatically during the DNS server installation.

/forward | /reverse | /ds | /file

Specifies additional filters of the types of zones to display:

/forward

lists forward lookup zones.

/reverse

lists reverse lookup zones.

/ds

lists active directory integrated zones.

/file

lists zones that are backed by files.

/domaindirectorypartition

lists zones that are stored in the domain directory partition.

/forestdirectorypartition

lists zones that are stored in the forest DNS application directory partition.

/customdirectorypartition

lists all zones that are stored in a user-defined application directory partition.

/legacydirectorypartition

lists all zones that are stored in the domain directory partition.

/directorypartition

lists all zones that are stored in the specified directory partition.

Remarks

- The **enumzones** parameters act as filters on the list of zones. If no filters are specified, a complete list of zones is returned. When a filter is specified, only the zones that meet that filter's criteria are included in the returned list of zones.

Example

See [Example 2: Display a complete list of zones on a DNS server](#) or [Example 3: Display a list of autocreated zones on a DNS server](#).

dnscmd /exportsettings

creates a text file that lists the configuration details of a DNS server. The text file is named DnsSettings.txt. It is located in the %systemroot%\system32\dns directory of the server.

Syntax

```
dnscmd [<ServerName>] /exportsettings
```

Parameters

Specifies the DNS server to manage, represented by local computer syntax, IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

Remarks

- You can use the information in the file that **dnscmd /exportsettings** creates to troubleshoot configuration problems or to ensure that you have configured multiple servers identically.

dnscmd /info

Displays settings from the DNS section of the registry of the specified server:

HKEY_LOCAL_MACHINE\SYSTEM\CurrentControlSet\Services\DNS\Parameters

Syntax

```
dnscmd [<ServerName>] /info [<Setting>]
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

Any setting that the **info** command returns can be specified individually. If a setting is not specified, a report of common settings is returned.

Remarks

- This command displays registry settings that are at the DNS server level. To display zone-level registry settings, use the [zoneinfo](#) command. To see a list of settings that can be displayed with this command, see the [config](#) description.

Example

See [Example 4: Display the IsSlave setting from a DNS server](#) or [Example 5: Display the Recursiontimeout setting from a DNS server](#).

dnscmd /ipvalidate

Tests whether an IP address identifies a functioning DNS server or whether the DNS server can act as a forwarder, a root hint server, or a master server for a specific zone.

Syntax

```
dnscmd [<ServerName>] /ipvalidate <Context> [<ZoneName>] [[<IPAddress>]]
```

Parameters

Specifies the DNS server to manage, represented by local computer syntax, IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

Specifies the type of test to perform. You can specify any of the following tests:

- **/dnsservers** tests that the computers with the addresses that you specify are functioning DNS servers.
- **/forwarders** tests that the addresses that you specify identify DNS servers that can act as forwarders.
- **/roothints** tests that the addresses that you specify identify DNS servers that can act as root hint name servers.
- **/zonemasters** tests that the addresses that you specify identify DNS servers that are master servers for *ZoneName*.

Identifies the zone. Use this parameter with the **/zonemasters** parameter.

Specifies the IP addresses that the command tests.

Sample usage

```
dnscmd dnssvr1.contoso.com /ipvalidate /dnsservers 10.0.0.1 10.0.0.2
dnscmd dnssvr1.contoso.com /ipvalidate /zonemasters corp.contoso.com 10.0.0.2
```

dnscmd /nodedelete

deletes all records for a specified host.

Syntax

```
dnscmd [<ServerName>] /nodedelete <ZoneName> <NodeName> [/tree] [/f]
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

Specifies the name of the zone.

Specifies the host name of the node to delete.

/tree

deletes all the child records.

/f

executes the command without asking for confirmation.

Example

See [Example 6: delete the records from a node](#).

dnscmd /recordadd

adds a record to a specified zone in a DNS server.

Syntax

```
dnscmd [<ServerName>] /recordadd <ZoneName> <NodeName> <RRtype> <Rrdata>
```

Parameters

Specifies the DNS server to manage, represented by local computer syntax, IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

Specifies the zone in which the record resides.

Specifies a specific node in the zone.

Specifies the type of record to be added.

Specifies the type of data that is expected.

> [!NOTE]

> When you add a record, make sure that you use the correct data type and data format. For a list of resource record types and the appropriate data types, see [Resource records reference](#).

Sample usage

```
dnscmd dnssvr1.contoso.com /recordadd test A 10.0.0.5
dnscmd /recordadd test.contoso.com test MX 10 mailserver.test.contoso.com
```

dnscmd /recorddelete

deletes a resource record from a specified zone.

Syntax

```
dnscmd <ServerName> /recorddelete <ZoneName> <NodeName> <RRtype> <Rrdata>[/f]
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

Specifies the zone in which the resource record resides.

Specifies the name of the host.

Specifies the type of resource record to be deleted.

Specifies the type of data that is expected.

/f

executes the command without asking for confirmation:

- Because nodes can have more than one resource record, this command requires you to be very specific about the type of resource record that you want to delete.
- if you specify a data type and you do not specify a type of resource record data, all records with that specific data type for the specified node are deleted.

Sample usage

```
dnscmd /recorddelete test.contoso.com test MX 10 mailserver.test.contoso.com
```

dnscmd /resetforwarders

selects or resets the IP addresses to which the DNS server forwards DNS queries when it cannot resolve them locally.

Syntax

```
dnscmd [<ServerName>] /resetforwarders [<IPaddress> [,<IPaddress>...][/timeout <timeOut>] [/slave|noslave]
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

lists the IP addresses to which the DNS server forwards unresolved queries.

/timeout

Sets the number of seconds that the DNS server waits for a response from the forwarder. By default, this value

is five seconds.

/slave|noslave

Determines whether the DNS server performs its own iterative queries if the forwarder fails to resolve a query:

/slave

Prevents the DNS server from performing its own iterative queries if the forwarder fails to resolve a query.

/noslave

Allows the DNS server to perform its own iterative queries if the forwarder fails to resolve a query. This is the default setting.

Remarks

- By default, a DNS server performs iterative queries when it cannot resolve a query.
- Setting IP addresses by using the **resetforwarders** command causes the DNS server to perform recursive queries to the DNS servers at the specified IP addresses. If the forwarders do not resolve the query, the DNS server can then perform its own iterative queries.
- if the **/slave** parameter is used, the DNS server does not perform its own iterative queries. This means that the DNS server forwards unresolved queries only to the DNS servers in the list, and it does not attempt iterative queries if the forwarders do not resolve them. It is more efficient to set one IP address as a forwarder for a DNS server. You can use the **resetforwarders** command for internal servers in a network to forward their unresolved queries to one DNS server that has an external connection.
- listing a forwarder's IP address twice causes the DNS server to attempt to forward to that server twice.

Sample usage

```
dnscmd dnssvr1.contoso.com /resetforwarders 10.0.0.1 /timeout 7 /slave
dnscmd dnssvr1.contoso.com /resetforwarders /noslave
```

dnscmd /resetlistenaddresses

Specifies the IP addresses on a server that listens for DNS client requests.

Syntax

```
dnscmd [<ServerName>] /resetlistenaddresses [<listenaddress>]
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

Specifies an IP address on the DNS server that listens for DNS client requests. If no listen address is specified, all IP addresses on the server listen for client requests.

Remarks

- By default, all IP addresses on a DNS server listen for client DNS requests.

Sample usage

```
dnscmd dnssvr1.contoso.com /resetlistenaddresses 10.0.0.1
```

dnscmd /startscavenging

Tells a DNS server to attempt an immediate search for stale resource records in a specified DNS server.

Syntax

```
dnscmd [<ServerName>] /startscavenging
```

Parameter

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

Remarks

- Successful completion of this command starts a scavenger immediately.
- Although the command to start the scavenger appears to complete successfully, the scavenger does not start

unless the following preconditions are met:

- Scavenging is enabled for both the server and the zone.
- The zone is started.
- The resource records have a time stamp.
- for information about how to enable scavenging for the server, see the **scavenginginterval** parameter under Server-level syntax in the [config](#) section.
- for information about how to enable scavenging for the zone, see the **aging** parameter under Zone-level syntax in the [config](#) section.
- for information about how to start a zone that is paused, see the [zoneresume](#) section.
- for information about how to check resource records for a time stamp, see the [ageallrecords](#) section.
- if the scavenger fails, no warning message appears.

Sample usage

```
dnscmd dnssvr1.contoso.com /startscavenging
```

dnscmd /statistics

Displays or clears data for a specified DNS server.

Syntax

```
dnscmd [<ServerName>] /statistics [<StatID>] [/clear]
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

Specifies which statistic or combination of statistics to display. An identification number is used to identify a statistic. If no statistic ID number is specified, all statistics display.

The following is a list of numbers that can be specified and the corresponding statistic that displays:

00000001

time

00000002

query

00000004

query2

00000008

Recurse

00000010

Master

00000020

Secondary

00000040

WINS

00000100

Update

00000200

SkwanSec

00000400

Ds

00010000

Memory

00100000

PacketMem

00040000

Dbase

00080000

Records

00200000

NbstatMem

/clear

resets the specified statistics counter to zero.

Remarks

- The **statistics** command displays counters that begin on the DNS server when it is started or resumed.

Examples

See [Example 7: Display time statistics for a DNS server](#) or [Example 8: Display NbstatMem statistics for a DNS server](#).

dnscmd /unenlistdirectorypartition

removes the DNS server from the specified directory partition's replica set.

Syntax

```
dnscmd [<ServerName>] /unenlistdirectorypartition <PartitionFQDN>
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

The FQDN of the DNS application directory partition that will be removed.

dnscmd /writebackfiles

Checks the DNS server memory for changes, and writes them to persistent storage.

Syntax

```
dnscmd [<ServerName>] /writebackfiles [<ZoneName>]
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

Specifies the name of the zone to be updated.

Remarks

- The **writebackfiles** command updates all dirty zones or a specified zone. A zone is dirty when there are changes in memory that have not yet been written to persistent storage. This is a server-level operation that checks all zones. You can specify one zone in this operation or you can use the [zonewriteback](#) operation.

Sample usage

```
dnscmd dnssvr1.contoso.com /writebackfiles
```

dnscmd /zoneadd

adds a zone to the DNS server.

Syntax

```
dnscmd [<ServerName>] /zoneadd <ZoneName> <Zonetype> [/dp <FQDN>| {/domain|/enterprise|/legacy}]
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

Specifies the name of the zone.

Specifies the type of zone to create. Each zone type has different required parameters:

/dsprimary

creates an active directory integrated zone.

/primary /file

creates a standard primary zone, and specifies the name of the file that will store the zone information.

/secondary [...]

creates a standard secondary zone.

/stub [...] /file

creates a file-backed stub zone.

/dsstub [...]

creates an active directory integrated stub zone.

/forwarder [...] /file

Specifies that the created zone forwards unresolved queries to another DNS server.

/dsforwarder

Specifies that the created active directory integrated zone forwards unresolved queries to another DNS server.

/dp {/domain | /enterprise | /legacy}

Specifies the directory partition on which to store the zone.

Specifies FQDN of the directory partition.

/domain

Stores the zone on the domain directory partition.

/enterprise

Stores the zone on the enterprise directory partition.

/legacy

Stores the zone on a legacy directory partition.

Remarks

- Specifying a zone type of **/forwarder** or **/dsforwarder** creates a zone that performs conditional forwarding.

Sample usage

```
dnscmd dnssvr1.contoso.com /zoneadd test.contoso.com /dsprimary
dnscmd dnssvr1.contoso.com /zoneadd secondtest.contoso.com /secondary 10.0.0.2
```

dnscmd /zonechangedirectorypartition

changes the directory partition on which the specified zone resides.

Syntax

```
dnscmd [<ServerName>] /zonechangedirectorypartition <ZoneName> { [<NewPartitionName>] | [<Zonetype>] }
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

The FQDN of the current directory partition on which the zone resides.

The FQDN of the directory partition that the zone will be moved to.

Specifies the type of directory partition that the zone will be moved to.

/domain

moves the zone to the built-in domain directory partition.

/forest

moves the zone to the built-in forest directory partition.

/legacy

moves the zone to the directory partition that is created for pre active directory domain controllers. These directory partitions are not necessary for native mode.

dnscmd /zonedelete

deletes a specified zone.

Syntax

```
dnscmd [<ServerName>] /zonedelete <ZoneName> [/dsdel] [/f]
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

Specifies the name of the zone to be deleted.

/dsdel

deletes the zone from AD DS.

/f

Runs the command without asking for confirmation.

Example

See [Example 9: delete a zone from a DNS server](#).

dnscmd /zoneexport

creates a text file that lists the resource records of a specified zone.

Syntax

```
dnscmd [<ServerName>] /zoneexport <ZoneName> <ZoneExportFile>
```

Parameters

Specifies the DNS server to manage, represented by local computer syntax, IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

Specifies the name of the zone.

Specifies the name of the file to create.

Remarks

- The **zoneexport** operation creates a file of resource records for an active directory integrated zone for troubleshooting purposes. By default, the file that this command creates is placed in the DNS directory, which is by default the %systemroot%/System32/Dns directory.

Example

See [Example 10: Export zone resource records list to a file](#).

dnscmd /zoneinfo

Displays settings from the section of the registry of the specified zone:

HKEY_LOCAL_MACHINE\SYSTEM\CurrentControlSet\Services\DNS\Parameters\Zones

Syntax

```
dnscmd [<ServerName>] /zoneinfo <ZoneName> [<Setting>]
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

Specifies the name of the zone.

You can individually specify any setting that the **zoneinfo** command returns. If you do not specify a setting, all settings are returned.

Remarks

- The **zoneinfo** command displays registry settings that are at the DNS zone level at **HKEY_LOCAL_MACHINE\SYSTEM\CurrentControlSet\Services\DNS\Parameters\Zones**.
- To display server-level registry settings, use the [info](#) command.
- To see a list of settings that you can display with this command, see the [config](#) command.

Example

See [Example 11: Display RefreshInterval setting from the registry](#) or [Example 12: Display Aging setting from the registry](#).

dnscmd /zonepause

pauses the specified zone, which then ignores query requests.

Syntax

```
dnscmd [<ServerName>] /zonepause <ZoneName>
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

Specifies the name of the zone to be paused.

Remarks

- To resume a zone and make it available after it has been paused, use the [zoneresume](#) command.

Sample usage

```
dnscmd dnssvr1.contoso.com /zonepause test.contoso.com
```

dnscmd /zoneprint

lists the records in a zone.

Syntax

```
dnscmd [<ServerName>] /zoneprint <ZoneName>
```

Parameters

Specifies the DNS server to manage, represented by local computer syntax, IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

Identifies the zone to be listed.

dnscmd /zonerefresh

forces a secondary DNS zone to update from the master zone.

Syntax

```
dnscmd <ServerName> /zonerefresh <ZoneName>
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

Specifies the name of the zone to be refreshed.

Remarks

- The **zonerefresh** command forces a check of the version number in the master server's start of authority (SOA) resource record. If the version number on the master server is higher than the secondary server's version number, a zone transfer is initiated that updates the secondary server. If the version number is the same, no zone transfer occurs.
- The forced check occurs by default every 15 minutes. To change the default, use the **dnscmd config refreshinterval** command.

Sample usage

```
dnscmd dnssvr1.contoso.com /zonerefresh test.contoso.com
```

dnscmd /zonereload

Copies zone information from its source.

Syntax

```
dnscmd <ServerName> /zonereload <ZoneName>
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

Specifies the name of the zone to be reloaded.

Remarks

- if the zone is active directory integrated, it reloads from AD DS.
- if the zone is a standard file-backed zone, it reloads from a file.

Sample usage

```
dnscmd dnssvr1.contoso.com /zonereload test.contoso.com
```

dnscmd /zoneresetmasters

resets the IP addresses of the master server that provides zone transfer information to a secondary zone.

Syntax

```
dnscmd <ServerName> /zoneresetmasters <ZoneName> [/local] [<IPAddress> [<IPAddress>]...]
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

Specifies the name of the zone to be reloaded.

/local

Sets a local master list. This parameter is used for active directory integrated zones.

The IP addresses of the master servers of the secondary zone.

Remarks

- This value is originally set when the secondary zone is created. Use the **zoneresetmasters** command on the secondary server. This value has no effect if it is set on the master DNS server.

Sample usage

```
dnscmd dnssvr1.contoso.com /zoneresetmasters test.contoso.com 10.0.0.1
dnscmd dnssvr1.contoso.com /zoneresetmasters test.contoso.com /local
```

dnscmd /zoneresetscavengeservers

changes the IP addresses of the servers that can scavenge the specified zone.

Syntax

```
dnscmd [<ServerName>] /zoneresetscavengeservers <ZoneName> [<IPAddress> [<IPAddress>]...]
```

Parameters

Specifies the DNS server to manage, represented by local computer syntax, IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

Identifies the zone to scavenge.

lists the IP addresses of the servers that can perform the scavenge. If this parameter is omitted, all servers that host this zone can scavenge it.

Remarks

- By default, all servers that host a zone can scavenge that zone.
- if a zone is hosted on more than one DNS server, you can use this command to reduce the number of times a zone is scavenged.
- Scavenging must be enabled on the DNS server and zone that is affected by this command.

Sample usage

```
dnscmd dnssvr1.contoso.com /zonereset scavengeservers test.contoso.com 10.0.0.1 10.0.0.2
```

dnscmd /zoneresetsecondaries

Specifies a list of IP addresses of secondary servers to which a master server responds when it is asked for a zone transfer.

Syntax

```
dnscmd [<ServerName>] /zoneresetsecondaries <ZoneName> {/noxfr | /nonsecure | /securens | /securelist  
<SecurityIPAddresses>} {/nonotify | /notify | /notifylist <NotifyIPAddresses>}
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If the is parameter is omitted, the local server is used.

Specifies the name of the zone that will have its secondary servers reset.

/noxfr | /nonsecure | /securens | /securelist

Specifies whether all or only some of the secondary servers requesting an update get an update.

/noxfr

Specifies that no zone transfers are allowed.

/nonsecure

Specifies that all zone transfer requests are granted.

/securens

Specifies that only the server that is listed in the name server (NS) resource record for the zone is granted a transfer.

/securelist

Specifies that zone transfers are granted only to the list of servers. This parameter must be followed by an IP address or addresses that the master server uses.

lists the IP addresses that receive zone transfers from the master server. This parameter is used only with the **/securelist** parameter.

/nonotify | /notify | /notifylist

Specifies that a change notification is sent only to certain secondary servers:

/nonotify

Specifies that no change notifications are sent to secondary servers.

/notify

Specifies that change notifications are sent to all secondary servers.

/notifylist

Specifies that change notifications are sent to only the list of servers. This command must be followed by an IP address or addresses that the master server uses.

Specifies the IP address or addresses of the secondary server or servers to which change notifications are sent. This list is used only with the **/notifylist** parameter.

Remarks

- Use the **zoneresetsecondaries** command on the master server to specify how it responds to zone transfer requests from secondary servers.

Sample usage

```
dnscmd dnssvr1.contoso.com /zoneresetsecondaries test.contoso.com /noxfr /nonotify  
dnscmd dnssvr1.contoso.com /zoneresetsecondaries test.contoso.com /securelist 11.0.0.2
```

dnscmd /zoneresettype

changes the type of the zone.

Syntax

```
dnscmd [<ServerName>] /zoneresettype <ZoneName> <Zonetype> [/overwrite_mem | /overwrite_ds]
```

Parameters

Specifies the DNS server to manage, represented by local computer syntax, IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

Identifies the zone on which the type will be changed.

Specifies the type of zone to create. Each type has different required parameters:

/dsprimary

creates an active directory integrated zone.

/primary /file

creates a standard primary zone.

/secondary [...]

creates a standard secondary zone.

/stub [...] /file

creates a file-backed stub zone.

/dsstub [...]

creates an active directory integrated stub zone.

/forwarder <MasterIPAddress[,...] /file

Specifies that the created zone forwards unresolved queries to another DNS server.

/dsforwarder

Specifies that the created active directory integrated zone forwards unresolved queries to another DNS server.

/overwrite_mem | /overwrite_ds

Specifies how to overwrite existing data:

/overwrite_mem

Overwrites DNS data from data in AD DS.

/overwrite_ds

Overwrites existing data in AD DS.

Remarks

- Setting the zone type as **/dsforwarder** creates a zone that performs conditional forwarding.

Sample usage

```
dnscmd dnssvr1.contoso.com /zoneresettype test.contoso.com /primary /file
test.contoso.com.dns
dnscmd dnssvr1.contoso.com /zoneresettype second.contoso.com /secondary 10.0.0.2
```

dnscmd /zoneresume

starts a specified zone that was previously paused.

Syntax

```
dnscmd <ServerName> /zoneresume <ZoneName>
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

Specifies the name of the zone to resume.

Remarks

- You can use this operation to reverse the [zonepause](#) operation.

Sample usage

```
dnscmd dnssvr1.contoso.com /zoneresume test.contoso.com
```

dnscmd /zoneupdatefromds

Updates the specified active directory integrated zone from AD DS.

Syntax

```
dnscmd <ServerName> /zoneupdatefromds <ZoneName>
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

Specifies the name of the zone to update.

Remarks

- active directory integrated zones perform this update by default every five minutes. To change this parameter, use the **dnscmd config dspollinginterval** command.

Sample usage

```
dnscmd dnssvr1.contoso.com /zoneupdatefromds
```

dnscmd /zonewriteback

Checks DNS server memory for changes that are relevant to a specified zone, and writes them to persistent storage.

Syntax

```
dnscmd <ServerName> /zonewriteback <ZoneName>
```

Parameters

Specifies the DNS server to manage, represented by IP address, FQDN, or host name. If this parameter is omitted, the local server is used.

Specifies the name of the zone to update.

Remarks

- This is a zone-level operation. You can update all zones on a DNS server with the [writebackfiles](#) operation.

Sample usage

```
dnscmd dnssvr1.contoso.com /zonewriteback test.contoso.com
```

doskey

4/13/2018 • 9 minutes to read • [Edit Online](#)

Calls Doskey.exe (which recalls previously entered command-line commands), edits command lines, and creates macros.

For examples of how to use this command, see [Examples](#).

Syntax

```
doskey [/reinstall] [/listsize=<Size>] [/macros:[all | <ExeName>] [/history] [/insert | /overstrike] [/exename=<ExeName>] [/macrofile=<FileName>] [<MacroName>=<Text>]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------------------|--|
| /reinstall | Installs a new copy of Doskey.exe and clears the command history buffer. |
| /listsize= <Size> | Specifies the maximum number of commands in the history buffer. |
| /macros | Displays a list of all doskey macros. You can use the redirection symbol (>) with /macros to redirect the list to a file. You can abbreviate /macros to /m . |
| /macros:all | Displays doskey macros for all executables. |
| /macros: <ExeName> | Displays doskey macros for the executable specified by <i>ExeName</i> . |
| /history | Displays all commands that are stored in memory. You can use the redirection symbol (>) with /history to redirect the list to a file. You can abbreviate /history as /h . |
| [/insert | /overstrike] |
| /exename= <ExeName> | Specifies the program (that is, executable) in which the doskey macro runs. |
| /macrofile= <FileName> | Specifies a file that contains the macros that you want to install. |
| <MacroName>=[] | Creates a macro that carries out the commands specified by <i>Text</i> . <i>MacroName</i> specifies the name you want to assign to the macro. <i>Text</i> specifies the commands you want to record. If <i>Text</i> is left blank, <i>MacroName</i> is cleared of any assigned commands. |

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------------|
| /? | Displays help at the command prompt. |

Remarks

- Using Doskey.exe

Doskey.exe is always available for all character-based, interactive programs (such as program debuggers or file transfer programs), and it maintains a command history buffer and macros for each program that it starts. You cannot use **doskey** command-line options from a program. You must run **doskey** command-line options before you start a program. Program key assignments override **doskey** key assignments.

- Recalling a command

To recall a command, you can use any of the following keys after you start Doskey.exe. If you use Doskey.exe within a program, that program's key assignments take precedence.

| KEY | DESCRIPTION |
|------------|---|
| UP ARROW | Recalls the command that you used before the one that is displayed. |
| DOWN ARROW | Recalls the command that you used after the one that is displayed. |
| PAGE UP | Recalls the first command that you used in the current session. |
| PAGE DOWN | Recalls the most recent command that you used in the current session. |

- Editing the command line

With Doskey.exe, you can edit the current command line. If you use Doskey.exe within a program, that program's key assignments take precedence and some Doskey.exe editing keys might not work.

The following table lists **doskey** editing keys and their functions.

| KEY OR KEY COMBINATION | DESCRIPTION |
|------------------------|---|
| LEFT ARROW | Moves the insertion point back one character. |
| RIGHT ARROW | Moves the insertion point forward one character. |
| CTRL+LEFT ARROW | Moves the insertion point back one word. |
| CTRL+RIGHT ARROW | Moves the insertion point forward one word. |
| HOME | Moves the insertion point to the beginning of the line. |
| END | Moves the insertion point to the end of the line. |
| ESC | Clears the command from the display. |

| KEY OR KEY COMBINATION | DESCRIPTION |
|------------------------|---|
| F1 | Copies one character from a column in the template to the same column in the Command Prompt window. (The template is a memory buffer that holds the last command you typed.) |
| F2 | Searches forward in the template for the next key that you type after you press F2. Doskey.exe inserts the text from the template—up to, but not including, the character you specify. |
| F3 | Copies the remainder of the template to the command line. Doskey.exe begins copying characters from the position in the template that corresponds to the position indicated by the insertion point on the command line. |
| F4 | Deletes all characters from the current insertion point position up to, but not including, the next occurrence of the character that you type after you press F4. |
| F5 | Copies the template into the current command line. |
| F6 | Places an end-of-file character (CTRL+Z) at the current insertion point position. |
| F7 | Displays (in a dialog box) all commands for this program that are stored in memory. Use the UP ARROW key and the DOWN ARROW key to select the command you want, and press ENTER to run the command. You can also note the sequential number in front of the command and use this number in conjunction with the F9 key. |
| ALT+F7 | Deletes all commands stored in memory for the current history buffer. |
| F8 | Displays all commands in the history buffer that start with the characters in the current command. |
| F9 | Prompts you for a history buffer command number, and then displays the command associated with the number that you specify. Press ENTER to run the command. To display all the numbers and their associated commands, press F7. |
| ALT+F10 | Deletes all macro definitions. |

- Using **doskey** within a program

Certain character-based, interactive programs, such as program debuggers or file transfer programs (FTP) automatically use Doskey.exe. To use Doskey.exe, a program must be a console process and use buffered input. Program key assignments override **doskey** key assignments. For example, if the program uses the F7 key for a function, you cannot get a **doskey** command history in a pop-up window.

With Doskey.exe, you can maintain a command history for each program that you start or repeat. You can edit previous commands at the program's prompt, and start **doskey** macros created for the program. If you exit and then restart a program from the same Command Prompt window, the command history from the previous program session is available.

You must run Doskey.exe before you start a program. You cannot use **doskey** command-line options from a program's command prompt, even if the program has a shell command.

If you want to customize how Doskey.exe works with a program and create **doskey** macros for that program, you can create a batch program that modifies Doskey.exe and starts the program.

- Specifying a default Insert mode

If you press the INSERT key, you can type text on the **doskey** command line in the midst of existing text without replacing the text. However, after you press ENTER, Doskey.exe returns your keyboard to Replace mode. You must press INSERT again to return to Insert mode.

Use **/insert** to switch your keyboard to Insert mode each time you press ENTER. Your keyboard effectively remains in Insert mode until you use **/overstrike**. You can temporarily return to Replace mode by pressing the INSERT key, but after you press ENTER, Doskey.exe returns your keyboard to Insert mode.

The insertion point changes shape when you use the INSERT key to change from one mode to the other.

- Creating a macro

You can use Doskey.exe to create macros that carry out one or more commands. The following table lists special characters that you can use to control command operations when you define a macro.

| CHARACTER | DESCRIPTION |
|------------------|---|
| \$G or \$g | Redirects output. Use either of these special characters to send output to a device or a file instead of to the screen. This character is equivalent to the redirection symbol for output (>). |
| \$G\$G or \$g\$g | Appends output to the end of a file. Use either of these double characters to append output to an existing file instead of replacing the data in the file. These double characters are equivalent to the append redirection symbol for output (>>). |
| \$L or \$l | Redirects input. Use either of these special characters to read input from a device or a file instead of from the keyboard. This character is equivalent to the redirection symbol for input (<). |
| \$B or \$b | Sends macro output to a command. These special characters are equivalent to using the pipe (**). |
| \$T or \$t | Separates commands. Use either of these special characters to separate commands when you create macros or type commands on the doskey command line. These special characters are equivalent to using the ampersand (&) on a command line. |
| \$\$ | Specifies the dollar-sign character (\$). |
| \$1 through \$9 | Represent any command-line information you want to specify when you run the macro. The special characters \$1 through \$9 are batch parameters that enable you to use different data on the command line each time you run the macro. The \$1 character in a doskey command is similar to the %1 character in a batch program. |

| CHARACTER | DESCRIPTION |
|-----------|--|
| \$* | Represents all the command-line information that you want to specify when you type the macro name. The special character \$* is a replaceable parameter that is similar to the batch parameters \$1 through \$9 , with one important difference: everything you type on the command line after the macro name is substituted for the \$* in the macro. |

- Running a **doskey** macro

To run a macro, type the macro name at the command prompt, starting at the first position. If the macro was defined with **\$*** or any of the batch parameters **\$1** through **\$9**, use a space to separate the parameters. You cannot run a **doskey** macro from a batch program.

- Creating a macro with the same name as a Windows Server 2003 family command

If you always use a particular command with specific command-line options, you can create a macro that has the same name as the command. To specify whether you want to run the macro or the command, follow these guidelines:

- To run the macro, type the macro name at the command prompt. Do not add a space before the macro name.
- To run the command, insert one or more spaces at the command prompt, and then type the command name.

- Deleting a macro

To delete a macro, type:

```
doskey <MacroName> =
```

Examples

The **/macros** and **/history** command-line options are useful for creating batch programs to save macros and commands. For example, to store all current **doskey** macros, type:

```
doskey /macros > macinit
```

To use the macros stored in Macinit, type:

```
doskey /macrofile=macinit
```

To create a batch program named Tmp.bat that contains recently used commands, type:

```
doskey /history> tmp.bat
```

To define a macro with multiple commands, use **\$t** to separate commands, as follows:

```
doskey tx=cd temp$tdir/w $*
```

In the preceding example, the TX macro changes the current directory to Temp and then displays a directory listing in wide display format. You can use **\$*** at the end of the macro to append other command-line options to **dir** when

you run TX.

The following macro uses a batch parameter for a new directory name:

```
doskey mc=md $1&tcd $1
```

The macro creates a new directory and then changes to the new directory from the current directory.

To use the preceding macro to create and change to a directory named Books, type:

```
mc books
```

To create a **doskey** macro for a program called Ftp.exe, include **/exename** as follows:

```
doskey /exename=ftp.exe go=open 172.27.1.100&tget *.TXT c:\reports&tbye
```

To use the preceding macro, start FTP. At the FTP prompt, type:

```
go
```

FTP runs the **open**, **mget**, and **bye** commands.

To create a macro that quickly and unconditionally formats a disk, type:

```
doskey qf=format $1 /q /u
```

To quickly and unconditionally format a disk in drive A, type:

```
qf a:
```

To delete a macro called vlist, type:

```
doskey vlist =
```

Additional references

[Command-Line Syntax Key](#)

driverquery

4/13/2018 • 2 minutes to read • [Edit Online](#)

Enables an administrator to display a list of installed device drivers and their properties. If used without parameters, **driverquery** runs on the local computer.

For examples of how to use this command, see [Examples](#).

Syntax

```
driverquery [/s <System> [/u [<Domain>\]<Username> [/p <Password>]]] [/fo {table | list | csv}] [/nh] [/v | /si]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------|--|
| /s <System> | Specifies the name or IP address of a remote computer. Do not use backslashes. The default is the local computer. |
| /u [<Domain>] | Runs the command with the credentials of the user account as specified by <i>User</i> or <i>Domain*User</i> . <i>By default, */s* uses the credentials of the user who is currently logged on to the computer that is issuing the command. /u cannot be used unless /s is specified.</i> |
| /p <Password> | Specifies the password of the user account that is specified in the /u parameter. /p cannot be used unless /u is specified. |
| /fo {table | list |
| /nh | Omits the header row from the displayed driver information. Not valid if the /fo parameter is set to list . |
| /v | Displays verbose output. /v is not valid for signed drivers. |
| /si | Provides information about signed drivers. |
| /? | Displays help at the command prompt. |

Examples

To display a list of installed device drivers on the local computer, type:

```
driverquery
```

To display the output in a comma-separated values (CSV) format, type:

```
driverquery /fo csv
```

To hide the header row in the output, type:

```
driverquery /nh
```

To use the **driverquery** command on a remote server named **server1** using your current credentials on the local computer, type:

```
driverquery /s server1
```

To use the **driverquery** command on a remote server named **server1** using the credentials for **user1** on the domain **maindom**, type:

```
driverquery /s server1 /u maindom\user1 /p p@ssw3d
```

Additional references

[Command-Line Syntax Key](#)

echo

5/7/2018 • 2 minutes to read • [Edit Online](#)

Displays messages or turns on or off the command echoing feature. If used without parameters, **echo** displays the current echo setting.

For examples of how to use this command, see [Examples](#).

Syntax

```
echo [<Message>]
echo [on | off]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|--|
| [on off] | Turns on or off the command echoing feature. Command echoing is on by default. |
| <Message> | Specifies the text to display on the screen. |
| /? | Displays help at the command prompt. |

Remarks

- The **echo Message** command is particularly useful when **echo** is turned off. To display a message that is several lines long without displaying any commands, you can include several **echo Message** commands after the **echo off** command in your batch program.
- When **echo** is turned off, the command prompt does not appear in the Command Prompt window. To display the command prompt, type **echo on**.
- If used in a batch file, **echo on** and **echo off** do not affect the setting at the command prompt.
- To prevent echoing a particular command in a batch file, insert an at sign (@) in front of the command. To prevent echoing all commands in a batch file, include the **echo off** command at the beginning of the file.
- To display a pipe (|) or redirection character (< or >) when you are using **echo**, use a caret (^) immediately before the pipe or redirection character (for example, ^|, ^>, or ^<). To display a caret, type two carets in succession (^ ^).

Examples

To display the current **echo** setting, type:

```
echo
```

To echo a blank line on the screen, type:

```
echo.
```

NOTE

Do not include a space before the period. Otherwise, the period will be displayed instead of a blank line.

To prevent echoing commands at the command prompt, type:

```
echo off
```

NOTE

When **echo** is turned off, the command prompt does not appear in the Command Prompt window. To display the command prompt again, type **echo on**.

To prevent all commands in a batch file (including the **echo off** command) from displaying on the screen, on the first line of the batch file type:

```
@echo off
```

You can use the **echo** command as part of an **if** statement. For example, to search the current directory for any file with the .rpt file name extension, and to echo a message if such a file is found, type:

```
if exist *.rpt echo The report has arrived.
```

The following batch file searches the current directory for files with the .txt file name extension, and displays a message indicating the results of the search:

```
@echo off
if not exist *.txt (
echo This directory contains no text files.
) else (
echo This directory contains the following text files:
echo.
dir /b *.txt
)
```

If no .txt files are found when the batch file is run, the following message displays:

```
This directory contains no text files.
```

If .txt files are found when the batch file is run the following output displays (for this example, assume the files File1.txt, File2.txt, and File3.txt exist):

```
This directory contains the following text files:
File1.txt
File2.txt
File3.txt
```

Additional references

edit

4/13/2018 • 2 minutes to read • [Edit Online](#)

Starts MS-DOS Editor, which creates and changes ASCII text files.

For examples of how to use this command, see [Examples](#).

Syntax

```
edit [/b] [/h] [/r] [/s] [/<NNN>] [[<Drive>:][<Path>]<FileName> [<FileName2> [...]]]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------------|---|
| [<Drive>:] [[...]] | Specifies the location and name of one or more ASCII text files. If the file does not exist, MS-DOS Editor creates it. If the file exists, MS-DOS Editor opens it and displays its contents on the screen. <i>FileName</i> can contain wildcard characters (* and ?). Separate multiple file names with spaces. |
| /b | Forces monochrome mode, so that MS-DOS Editor displays in black and white. |
| /h | Displays the maximum number of lines possible for the current monitor. |
| /r | Loads file(s) in read-only mode. |
| /s | Forces the use of short filenames. |
| <NNN> | Loads binary file(s), wrapping lines to <i>NNN</i> characters wide. |
| /? | Displays help at the command prompt. |

Remarks

- For additional help, open MS-DOS Editor, and then press the F1 key.
- Some monitors do not support the display of shortcut keys by default. If your monitor does not display shortcut keys, use **/b**.

Examples

To open MS-DOS Editor, type:

```
edit
```

To create and edit a file named newtextfile.txt in the current directory, type:

edit newtextfile.txt

endlocal

4/13/2018 • 2 minutes to read • [Edit Online](#)

Ends localization of environment changes in a batch file, and restores environment variables to their values before the corresponding **setlocal** command was run.

For examples of how to use this command, see [Examples](#).

Syntax

```
endlocal
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------------|
| /? | Displays help at the command prompt. |

Remarks

- The **endlocal** command has no effect outside a script or batch file.
- There is an implicit **endlocal** command at the end of a batch file.
- If command extensions are enabled (command extensions are enabled by default), the **endlocal** command restores the state of command extensions (that is, enabled or disabled) to what it was before the corresponding **setlocal** command was run.

NOTE

For more information about enabling and disabling command extensions, see [Cmd](#).

Examples

You can localize environment variables in a batch file. For example, the following program starts the superapp batch program on the network, directs the output to a file, and displays the file in Notepad:

```
@echo off
setlocal
path=g:\programs\superapp;%path%
call superapp>c:\superapp.out
endlocal
start notepad c:\superapp.out
```

Additional references

[Command-Line Syntax Key](#)

erase

4/13/2018 • 2 minutes to read • [Edit Online](#)

This command is the same as the **del** command. See [Del](#) for syntax and parameters.

eventcreate

10/29/2018 • 2 minutes to read • [Edit Online](#)

Enables an administrator to create a custom event in a specified event log. For examples of how to use this command, see [Examples](#).

Syntax

```
eventcreate [/s <Computer> [/u <Domain\User> [/p <Password>]] {[ /l {APPLICATION|SYSTEM}} [/so <SrcName>]] /t {ERROR|WARNING|INFORMATION|SUCCESSAUDIT|FAILUREAUDIT} /id <EventID> /d <Description>
```

Parameters

| PARAMETER | DESCRIPTION |
|--|---|
| /s <Computer> | Specifies the name or IP address of a remote computer (do not use backslashes). The default is the local computer. |
| /u <Domain\User> | Runs the command with the account permissions of the user specified by <User> or <Domain\User>. The default is the permissions of the current logged on user on the computer issuing the command. |
| /p <Password> | Specifies the password of the user account that is specified in the /u parameter. |
| /l {APPLICATION SYSTEM} | Specifies the name of the event log where the event will be created. The valid log names are APPLICATION and SYSTEM. |
| /so <SrcName> | Specifies the source to use for the event. A valid source can be any string and should represent the application or component that is generating the event. |
| /t {ERROR WARNING INFORMATION SUCCESSAUDIT FAILUREAUDIT} | Specifies the type of event to create. The valid types are ERROR, WARNING, INFORMATION, SUCCESSAUDIT, and FAILUREAUDIT. |
| /id <EventID> | Specifies the event ID for the event. A valid ID is any number from 1 to 1000. |
| /d <Description> | Specifies the description to use for the newly created event. |
| /? | Displays help at the command prompt. |

Remarks

- Custom events cannot be written to the security log.

Examples

The following examples show how you can use the eventcreate command:

```
eventcreate /t error /id 100 /l application /d "Create event in application log"
eventcreate /t information /id 1000 /so winmgmt /d "Create event in WinMgmt source"
eventcreate /t error /id 2001 /so winword /l application /d "new src Winword in application log"
eventcreate /s server /t error /id 100 /l application /d "Remote machine without user credentials"
eventcreate /s server /u user /p password /id 100 /t error /l application /d "Remote machine with user credentials"
eventcreate /s server1 /s server2 /u user /p password /id 100 /t error /so winmgmt /d "Creating events on Multiple remote machines"
eventcreate /s server /u user /id 100 /t warning /so winmgmt /d "Remote machine with partial user credentials"
```

Additional References

[Command-Line Syntax Key](#)

eventquery

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

eventquery is deprecated, and is not guaranteed to be supported in future releases of Windows. This tool is included in Windows Server 2003 . For more information see [eventquery](#).

eventtriggers

4/13/2018 • 2 minutes to read • [Edit Online](#)

Eventtriggers is deprecated, and is not guaranteed to be supported in future releases of Windows.

This tool is included in Windows Server 2003. For more information see [Eventtriggers](#).

Evntcmd

7/11/2018 • 4 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Configures the translation of events to traps, trap destinations, or both based on information in a configuration file.

Syntax

```
evntcmd [/s <computerName>] [/v <verbosityLevel>] [/n] <FileName>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| /s | Specifies, by name, the computer on which you want to configure the translation of events to traps, trap destinations, or both. If you do not specify a computer, the configuration occurs on the local computer. |
| /v | Specifies which types of status messages appear as traps and trap destinations are configured. This parameter must be an integer between 0 and 10. If you specify 10, all types of messages appear, including tracing messages and warnings about whether trap configuration was successful. If you specify 0, no messages appear. |
| /n | Specifies that the SNMP service should not be restarted if this computer receives trap configuration changes. |
| | Specifies, by name, the configuration file that contains information about the translation of events to traps and trap destinations you want to configure. |
| /? | Displays help at the command prompt. |

Remarks

- if you want to configure traps but not trap destinations, you can create a valid configuration file by using Event to Trap Translator, which is a graphical utility. If you have the SNMP service installed, you can start Event to Trap Translator by typing **evntwin** at a command prompt. After you have defined the traps you want, click Export to create a file suitable for use with **evntcmd**. You can use Event to Trap Translator to easily create a configuration file and then use the configuration file with **evntcmd** at the command prompt to quickly configure traps on multiple computers.
- The syntax for configuring a trap is as follows:
#pragma add []
 - The text **#pragma** must appear at the beginning of every entry in the file.
 - The parameter **add** specifies that you want to add an event to trap configuration.

- The parameters *EventLogFile*, *EventSource*, and *EventID* are required. The parameter *EventLogFile* specifies the file in which the event is recorded. The parameter *EventSource* specifies the application that generates the event. The *EventID* parameter specifies the unique number that identifies each event. To find out what values correspond to particular events, start Event to Trap Translator by typing **evntwin** at a command prompt. Click **Custom**, and then click **edit**. Under **Event Sources**, browse the folders until you locate the event you want to configure, click it, and then click **add**. Information about the event source, the event log file, and the event ID appear under **Source**, **Log**, and **Trap specific ID**, respectively.
- The *Count* parameter is optional, and it specifies how many times the event must occur before a trap message is sent. If you do not use the *Count* parameter, the trap message is sent after the event occurs once.
- The *Period* parameter is optional, but it requires you to use the *Count* parameter. The *Period* parameter specifies a length of time (in seconds) during which the event must occur the number of times specified with the *Count* parameter before a trap message is sent. If you do not use the *Period* parameter, a trap message is sent after the event occurs the number of times specified with the *Count* parameter, no matter how much time elapses between occurrences.

- The syntax for removing a trap is as follows:

#pragma delete

- The text **#pragma** must appear at the beginning of every entry in the file.
- The parameter *delete* specifies that you want to remove an event to trap configuration.
- The parameters *EventLogFile*, *EventSource*, and *EventID* are required. The parameter *EventLogFile* specifies the log in which the event is recorded. The parameter *EventSource* specifies the application that generates the event. The *EventID* parameter specifies the unique number that identifies each event.

- The syntax for configuring a trap destination is as follows:

#pragma add_TRAP_DEST

- The text **#pragma** must appear at the beginning of every entry in the file.
- The parameter **add_TRAP_DEST** specifies that you want trap messages to be sent to a specified host within a community.
- The parameter *CommunityName* specifies, by name, the community in which trap messages are sent.
- The parameter *HostID* specifies, by name or IP address, the host to which you want trap messages to be sent.

- The syntax for removing a trap destination is as follows:

#pragma delete_TRAP_DEST

- The text **#pragma** must appear at the beginning of every entry in the file.
- The parameter *delete_TRAP_DEST* specifies that you do not want trap messages to be sent to a specified host within a community.
- The parameter *CommunityName* specifies, by name, the community in which trap messages are sent.
- The parameter *HostID* specifies, by name or IP address, the host to which you do not want trap messages to be sent.

Examples

The following examples illustrate entries in the configuration file for the **evntcmd** command. They are not designed to be typed at a command prompt.

To send a trap message if the Event Log service is restarted, type:

```
#pragma add System "Eventlog" 2147489653
```

To send a trap message if the Event Log service is restarted twice in three minutes, type:

```
#pragma add System "Eventlog" 2147489653 2 180
```

To stop sending a trap message whenever the Event Log service is restarted, type:

```
#pragma delete System "Eventlog" 2147489653
```

To send trap messages within the community named Public to the host with the IP address 192.168.100.100, type:

```
#pragma add_TRAP_DEST public 192.168.100.100
```

To send trap messages within the community named Private to the host named Host1, type:

```
#pragma add_TRAP_DEST private Host1
```

To stop sending trap messages within the community named Private to the same computer on which you are configuring trap destinations, type:

```
#pragma delete_TRAP_DEST private localhost
```

additional references

- [Command-Line Syntax Key](#)

exit

4/13/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

exits the Cmd.exe program (the command interpreter) or the current batch script.
for examples of how to use this command, see [Examples](#).

Syntax

```
exit [/b] [<exitCode>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| /b | exits the current batch script instead of exiting Cmd.exe. If executed from outside a batch script, exits Cmd.exe. |
| | Specifies a numeric number. If /b is specified, the ERRORLEVEL environment variable is set to that number. If you are quitting Cmd.exe , the process exit code is set to that number. |
| /? | Displays help at the command prompt. |

Examples

To close the command interpreter, Cmd.exe, type:

```
exit
```

additional references

- [Command-Line Syntax Key](#)

expand

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

expands one or more compressed files. You can use this command to retrieve compressed files from distribution disks.

Syntax

```
expand [/r] <source> <destination>
expand /r <source> [<destination>]
expand /i <source> [<destination>]
expand /d <source>.cab [/f:<files>]
expand <source>.cab /f:<files> <destination>
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------|--|
| /r | renames expanded files. |
| source | Specifies the files to expand. <i>Source</i> can consist of a drive letter and colon, a directory name, a file name, or a combination of these. You can use wildcards (* or ?). |
| destination | <p>Specifies where files are to be expanded.</p> <p>if <i>source</i> consists of multiple files and you do not specify /r, <i>destination</i> must be a directory.</p> <p><i>Destination</i> can consist of a drive letter and colon, a directory name, a file name, or a combination of these.</p> <p>Destination file path specification.</p> |
| /i | <p>renames expanded files but ignores the directory structure.</p> <p>This parameter applies to: Windows Server 2008 R2 and Windows 7 .</p> |
| /d | Displays a list of files in the source location. Does not expand or extract the files. |
| /f: | Specifies the files in a cabinet (.cab) file that you want to expand. You can use wildcards (* or ?). |
| /? | Displays help at the command prompt. |

Remarks

- Using **expand** at the recovery Console

The **expand** command, with different parameters, is available from the recovery Console. For more information about the recovery Console, see [article 314058](#) in the Microsoft Knowledge Base.

additional references

[Command-Line Syntax Key](#)

extract

4/13/2018 • 2 minutes to read • [Edit Online](#)

Syntax

```
EXTRACT [/Y] [/A] [/D | /E] [/L dir] cabinet [filename ...]  
EXTRACT [/Y] source [newname]  
EXTRACT [/Y] /C source destination
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| cabinet | File contains two or more files. |
| filename | Name of the file to extract from the cabinet. Wild cards and multiple filenames (separated by blanks) may be used. |
| source | Compressed file (a cabinet with only one file). |
| newname | New filename to give the extracted file. If not supplied, the original name is used. |
| /A | Process ALL cabinets. Follows cabinet chain starting in first cabinet mentioned. |
| /C | Copy source file to destination (to copy from DMF disks). |
| /D | Display cabinet directory (use with filename to avoid extract). |
| /E | Extract (use instead of . to extract all files). |
| /L dir | Location to place extracted files (default is current directory). |
| /Y | Do not prompt before overwriting an existing file. |

Additional references

[Command-Line Syntax Key](#)

Compares two files or sets of files and displays the differences between them.

For examples of how to use this command, see [Examples](#).

Syntax

```
fc /a [/c] [/l] [/lb<N>] [/n] [/off[line]] [/t] [/u] [/w] [/<NNNN>] [<Drive1>:][<Path1>]<FileName1> [<Drive2>:][<Path2>]<FileName2>  
fc /b [<Drive1>:][<Path1>]<FileName1> [<Drive2>:][<Path2>]<FileName2>
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------------|---|
| /a | Abbreviates the output of an ASCII comparison. Instead of displaying all of the lines that are different, fc displays only the first and last line for each set of differences. |
| /b | Compares the two files in binary mode, byte by byte, and does not attempt to resynchronize the files after finding a mismatch. This is the default mode for comparing files that have the following file extensions: .exe, .com, .sys, .obj, .lib, or .bin. |
| /c | Ignores the letter case. |
| /l | Compares the files in ASCII mode, line-by-line, and attempts to resynchronize the files after finding a mismatch. This is the default mode for comparing files, except files with the following file extensions: .exe, .com, .sys, .obj, .lib, or .bin. |
| /lb<N> | Sets the number of lines for the internal line buffer to <i>N</i> . The default length of the line buffer is 100 lines. If the files that you are comparing have more than 100 consecutive differing lines, fc cancels the comparison. |
| /n | Displays the line numbers during an ASCII comparison. |
| /off[<i>line</i>] | Does not skip files that have the offline attribute set. |
| /t | Prevents fc from converting tabs to spaces. The default behavior is to treat tabs as spaces, with stops at each eighth character position. |
| /u | Compares files as Unicode text files. |

| PARAMETER | DESCRIPTION |
|-------------|--|
| /w | Compresses white space (that is, tabs and spaces) during the comparison. If a line contains many consecutive spaces or tabs, /w treats these characters as a single space. When used with /w , fc ignores white space at the beginning and end of a line. |
| /<NNNN> | Specifies the number of consecutive lines that must match following a mismatch, before fc considers the files to be resynchronized. If the number of matching lines in the files is less than <i>NNNN</i> , fc displays the matching lines as differences. The default value is 2. |
| [<Drive1>:] | Specifies the location and name of the first file or set of files to compare. <i>FileName1</i> is required. |
| [<Drive2>:] | Specifies the location and name of the second file or set of files to compare. <i>FileName2</i> is required. |
| /? | Displays help at the command prompt. |

Remarks

- This command is implemented by c:\WINDOWS\fc.exe. You can use this command within PowerShell, but be sure to spell out the full executable (fc.exe) since 'fc' is an alias for Format-Custom.
- Reporting differences between files for an ASCII comparison

When you use **fc** for an ASCII comparison, **fc** displays the differences between two files in the following order:

- Name of the first file
 - Lines from *FileName1* that differ between the files
 - First line to match in both files
 - Name of the second file
 - Lines from *FileName2* that differ
 - First line to match
- Using **/b** for binary comparisons

/b displays mismatches that are found during a binary comparison in the following syntax:

```
\<XXXXXXXX: YY ZZ>
```

The value of XXXXXXXX specifies the relative hexadecimal address for the pair of bytes, measured from the beginning of the file. Addresses start at 00000000. The hexadecimal values for YY and ZZ represent the mismatched bytes from *FileName1* and *FileName2*, respectively.

- Using wildcard characters

You can use wildcard characters (*) and (?) in *FileName1* and *FileName2*. If you use a wildcard in *FileName1*, **fc** compares all the specified files to the file or set of files specified by *FileName2*. If you use a wildcard in *FileName2*, **fc** uses the corresponding value from *FileName1*.

- Working with memory

When comparing ASCII files, **fc** uses an internal buffer (large enough to hold 100 lines) as storage. If the

files are larger than the buffer, **fc** compares what it can load into the buffer. If **fc** does not find a match in the loaded portions of the files, it stops and displays the following message:

```
Resynch failed. Files are too different.
```

When comparing binary files that are larger than the available memory, **fc** compares both files completely, overlaying the portions in memory with the next portions from the disk. The output is the same as that for files that fit completely in memory.

Examples

To make an ASCII comparison of two text files, Monthly.rpt and Sales.rpt, and display the results in abbreviated format, type:

```
fc /a monthly.rpt sales.rpt
```

To make a binary comparison of two batch files, Profits.bat and Earnings.bat, type:

```
fc /b profits.bat earnings.bat
```

Results similar to the following appear:

```
00000002: 72 43
00000004: 65 3A
0000000E: 56 92
...
...
...
000005E8: 00 6E
FC: Earnings.bat longer than Profits.bat
```

If the Profits.bat and Earnings.bat files are identical, **fc** displays the following message:

```
Comparing files Profits.bat and Earnings.bat
FC: no differences encountered
```

To compare every .bat file in the current directory with the file New.bat, type:

```
fc *.bat new.bat
```

To compare the file New.bat on drive C with the file New.bat on drive D, type:

```
fc c:new.bat d:*.bat
```

To compare each batch file in the root directory on drive C to the file with the same name in the root directory on drive D, type:

```
fc c:*.bat d:*.bat
```

Additional references

[Command-Line Syntax Key](#)

find

4/13/2018 • 3 minutes to read • [Edit Online](#)

Searches for a string of text in a file or files, and displays lines of text that contain the specified string.

For examples of how to use this command, see [Examples](#).

Syntax

```
find [/v] [/c] [/n] [/i] [/off[line]] "<String>" [[<Drive>:][<Path>]<FileName>[...]]
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------|--|
| /v | Displays all lines that do not contain the specified <String>. |
| /c | Counts the lines that contain the specified <String> and displays the total. |
| /n | Precedes each line with the file's line number. |
| /i | Specifies that the search is not case-sensitive. |
| /off[line] | Does not skip files that have the offline attribute set. |
| "<String>" | Required. Specifies the group of characters (enclosed in quotation marks) that you want to search for. |
| [<Drive>:[] | Specifies the location and name of the file in which to search for the specified string. |
| /? | Displays help at the command prompt. |

Remarks

- Specifying a string

If you do not use **/i**, **find** searches for exactly what you specify for *String*. For example, the **find** command treats the characters "a" and "A" differently. If you use **/i**, however, **find** is not case sensitive, and it treats "a" and "A" as the same character.

If the string you want to search for contains quotation marks, you must use double quotation marks for each quotation mark contained within the string (for example, "This ""string"" contains quotation marks").

- Using **find** as a filter

If you omit a file name, **find** acts as a filter, taking input from the standard input source (usually the keyboard, a pipe (|), or a redirected file) and then displaying any lines that contain *String*.

- Ordering command syntax

You can type parameters and command-line options for the **find** command in any order.

- Using wildcards

You cannot use wildcards (***** and **?**) in file names or extensions that you specify with the **find** command. To search for a string in a set of files that you specify with wildcards, you can use the **find** command within a **for** command.

- Using **/v** or **/n** with **/c**

If you use **/c** and **/v** in the same command line, **find** displays a count of the lines that do not contain the specified string. If you specify **/c** and **/n** in the same command line, **find** ignores **/n**.

- Using **find** with carriage returns

The **find** command does not recognize carriage returns. When you use **find** to search for text in a file that includes carriage returns, you must limit the search string to text that can be found between carriage returns (that is, a string that is not likely to be interrupted by a carriage return). For example, **find** does not report a match for the string "tax file" if a carriage return occurs between the words "tax" and "file."

Examples

To display all lines from Pencil.ad that contain the string "Pencil Sharpener", type:

```
find "Pencil Sharpener" pencil.ad
```

To find a string that contains text within quotation marks, you must enclose the entire string in quotation marks. Then you must use two quotation marks for each quotation mark contained within the string. To find "The scientists labeled their paper "for discussion only." It is not a final report." in Report.doc, type:

```
find "The scientists labeled their paper ""for discussion only."" It is not a final report." report.doc
```

If you want to search for a set of files, you can use the **find** command within the **for** command. To search the current directory for files that have the extension .bat and that contain the string "PROMPT", type:

```
for %f in (*.bat) do find "PROMPT" %f
```

To search your hard disk to find and display the file names on drive C that contain the string "CPU", use the pipe (|) to direct the output of the **dir** command to the **find** command as follows:

```
dir c:\ /s /b | find "CPU"
```

Because **find** searches are case-sensitive and **dir** produces uppercase output, you must either type the string "CPU" in uppercase letters or use the **/i** command-line option with **find**.

Additional references

[Command-Line Syntax Key](#)

findstr

4/13/2018 • 3 minutes to read • [Edit Online](#)

Searches for patterns of text in files.

For examples of how to use this command, see [Examples](#).

Syntax

```
findstr [/b] [/e] [/l | /r] [/s] [/i] [/x] [/v] [/n] [/m] [/o] [/p] [/f:<File>] [/c:<String>] [/g:<File>] [/d:<DirList>] [/a:<ColorAttribute>] [/off[line]] <Strings> [<Drive>:][<Path>]<FileName>[ ...]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------------|---|
| /b | Matches the text pattern if it is at the beginning of a line. |
| /e | Matches the text pattern if it is at the end of a line. |
| /l | Processes search strings literally. |
| /r | Processes search strings as regular expressions. This is the default setting. |
| /s | Searches the current directory and all subdirectories. |
| /i | Ignores the case of the characters when searching for the string. |
| /x | Prints lines that match exactly. |
| /v | Prints only lines that do not contain a match. |
| /n | Prints the line number of each line that matches. |
| /m | Prints only the file name if a file contains a match. |
| /o | Prints character offset before each matching line. |
| /p | Skips files with non-printable characters. |
| /off[<i>line</i>] | Does not skip files that have the offline attribute set. |
| /f:<File> | Gets a file list from the specified file. |
| /c:<String> | Uses the specified text as a literal search string. |

| PARAMETER | DESCRIPTION |
|---------------------|---|
| /g:<File> | Gets search strings from the specified file. |
| /d:<DirList> | Searches the specified list of directories. Each directory must be separated with a semicolon (;), for example <code>dir1;dir2;dir3</code> . |
| /a:<ColorAttribute> | Specifies color attributes with two hexadecimal digits. Type <code>color /?</code> for additional information. |
| <Strings> | Specifies the text to search for in <i>FileName</i> . Required. |
| [<Drive>:][...] | Specifies the location and file or files to search. At least one file name is required. |
| /? | Displays Help at the command prompt. |

Remarks

- All **findstr** command-line options must precede *Strings* and *FileName* in the command string.
- Regular expressions use both literal characters and metacharacters to find patterns of text, rather than exact strings of characters. A literal character is a character that does not have a special meaning in the regular-expression syntax—it matches an occurrence of that character. For example, letters and numbers are literal characters. A metacharacter is a symbol with special meaning (an operator or delimiter) in the regular-expression syntax.

The following table lists the metacharacters that **findstr** accepts.

| METACHARACTER | VALUE |
|---------------|---|
| . | Wildcard: any character |
| * | Repeat: zero or more occurrences of the previous character or class |
| ^ | Line position: beginning of the line |
| \$ | Line position: end of the line |
| [class] | Character class: any one character in a set |
| [^class] | Inverse class: any one character not in a set |
| [x-y] | Range: any characters within the specified range |
| \x | Escape: literal use of a metacharacter x |
| \<string | Word position: beginning of the word |
| string> | Word position: end of the word |

The special characters in regular expression syntax have the most power when you use them together. For

example, use the following combination of the wildcard character (.) and repeat (*) character to match any string of characters:

```
.*
```

Use the following expression as part of a larger expression to match any string beginning with "b" and ending with "ing":

```
b.*ing
```

Examples

Use spaces to separate multiple search strings unless the argument is prefixed with **/c**.

To search for "hello" or "there" in file x.y, type:

```
findstr "hello there" x.y
```

To search for "hello there" in file x.y, type:

```
findstr /c:"hello there" x.y
```

To find all occurrences of the word "Windows" (with an initial capital letter W) in the file Proposal.txt, type:

```
findstr Windows proposal.txt
```

To search every file in the current directory and all subdirectories that contained the word Windows, regardless of the letter case, type:

```
findstr /s /i Windows *.*
```

To find all occurrences of lines that begin with "FOR" and are preceded by zero or more spaces (as in a computer program loop), and to display the line number where each occurrence is found, type:

```
findstr /b /n /r /c:"^ *FOR" *.bas
```

To search for multiple strings in a set of files, create a text file that contains each search criterion on a separate line. You can also list the exact files that you want to search in a text file. For example, to use the search criteria in the file Stringlist.txt, search the files listed in Filelist.txt, and then store the results in the file Results.out, type:

```
findstr /g:stringlist.txt /f:filelist.txt > results.out
```

To list every file containing the word "computer" within the current directory and all subdirectories, regardless of case, type:

```
findstr /s /i /m "\\<computer\>" *.*
```

To list every file containing the word "computer" and any other words that begin with "comp", (such as "compliment" and "compete"), type:

```
findstr /s /i /m "\\<comp.*" *.*
```

Additional references

[Command-Line Syntax Key](#)

finger

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Displays information about a user or users on a specified remote computer (typically a computer running UNIX) that is running the finger service or daemon. The remote computer specifies the format and output of the user information display. Used without parameters, **finger** displays help.

Syntax

```
finger [-l] [<User>] [@<Host>] [...]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| -l | Displays user information in long list format. |
| | Specifies the user about which you want information. If you omit the <i>User</i> parameter, finger displays information about all users on the specified computer. |
| @ | Specifies the remote computer running the finger service where you are looking for user information. You can specify a computer name or IP address. |
| /? | Displays help at the command prompt. |

Remarks

Multiple User@Host parameters can be specified. You must prefix **finger** parameters with a hyphen (-) rather than a slash (/). This command is available only if the Internet Protocol (TCP/IP) protocol is installed as a component in the properties of a network adapter in Network Connections. The Windows Server 2003 does not provide a finger service.

Examples

To display information for user1 on the computer users.microsoft.com, type:

```
finger user1@users.microsoft.com
```

To display information for all users on the computer users.microsoft.com, type:

```
finger @users.microsoft.com
```

additional references

- [Command-Line Syntax Key](#)

flattemp

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Enables or disables flat temporary folders. for examples of how to use this command, see [Examples](#).

NOTE

In Windows Server 2008 R2, Terminal Services was renamed Remote Desktop Services. To find out what's new in the latest version, see [What's New in Remote Desktop Services in Windows Server 2012](#) in the Windows Server TechNet Library.

Syntax

```
flattemp {/query | /enable | /disable}
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| /query | Queries the current setting. |
| /enable | Enables flat temporary folders. Users will share the temporary folder unless the temporary folder resides in the user's home folder. |
| /disable | Disables flat temporary folders. Each user's temporary folder will reside in a separate folder (determined by the user's Session ID). |
| /? | Displays help at the command prompt. |

Remarks

- The **flattemp** command is only available when you have installed the Terminal Server role service on a computer running Windows Server 2008 or the rd Session Host role service on a computer running Windows Server 2008 R2.
- You must have administrative credentials to run **flattemp**.
- After each user has a unique temporary folder, use **flattemp /enable** to enable flat temporary folders.
- The default method for creating temporary folders for multiple users (usually pointed to by the TEMP and TMP environment variables) is to create subfolders in the **\Temp** folder, by using the logonID as the subfolder name. For example, if the TEMP environment variable points to C:\Temp, the temporary folder assigned to the user logonID 4 is C:\Temp\4. Using **flattemp**, you can point directly to the \Temp folder and prevent subfolders from forming. This is useful when you want the user temporary folders to be contained in home folders, whether on an rd Session Host server local drive or on a shared network drive. You should use the **flattemp /enable** command only when each user has a separate temporary folder.

- You might encounter application errors if the user's temporary folder is on a network drive. This occurs when the shared network drive becomes momentarily inaccessible on the network. Because the temporary files of the application are either inaccessible or out of synchronization, it responds as if the disk has stopped. Moving the temporary folder to a network drive is not recommended. The default is to keep temporary folders on the local hard disk. If you experience unexpected behavior or disk-corruption errors with certain applications, stabilize your network or move the temporary folders back to the local hard disk.
- if you disable using separate temporary folders per-session, **flattemp** settings are ignored. This option is set in the Remote Desktop Services Configuration tool.

Examples

- To display the current setting for flat temporary folders, type: `flattemp /query`
- To enable flat temporary folders, type: `flattemp /enable`
- To disable flat temporary folders, type: `flattemp /disable`

additional references

[Command-Line Syntax Key](#)

[Remote Desktop Services \(Terminal Services\) Command Reference](#)

fondue

10/24/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Enables Windows optional features by downloading required files from Windows Update or another source specified by Group Policy. The manifest file for the feature must already be installed in your Windows image.

Syntax

```
fondue.exe /enable-feature:<feature_name> [/caller-name:<program_name>] [/hide-ux:{all | rebootRequest}]
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------------------------|--|
| /enable-feature:<feature_name> | Specifies the name of the Windows optional feature you want to enable. You can only enable one feature per command line. To enable multiple features, use fondue.exe for each feature. |
| /caller-name:<program_name> | Specifies the program or process name when you call fondue.exe from a script or batch file. You can use this option to add the program name to the SQM report if there is an error. |
| /hide-ux:{all rebootRequest} | <p>Use all to hide all messages to the user including progress and permission requests to access Windows Update. If permission is required, the operation will fail.</p> <p>Use rebootRequest to only hide user messages asking for permission to reboot the computer. Use this option if you have a script that controls reboot requests.</p> |

Examples

To enable Microsoft .NET Framework 3.5, type:

```
fondue.exe /enable-feature:NETFX3
```

To enable Microsoft .NET Framework 3.5, add the program name to the SQM report, and not display messages to the user, type:

```
fondue.exe /enable-feature:NETFX3 /caller-name:Admin.bat /hide-ux:all
```

additional references

- [Command-Line Syntax Key](#) ## See Also [Microsoft .NET Framework 3.5 Deployment Considerations](#)

for

8/3/2018 • 8 minutes to read • [Edit Online](#)

Runs a specified command for each file in a set of files.

For examples of how to use this command, see [Examples](#).

Syntax

```
for {%%|%}<Variable> in (<Set>) do <Command> [<CommandLineOptions>]
```

Parameters

| PARAMETER | DESCRIPTION |
|----------------------|--|
| {%% %}<Variable> | Required. Represents a replaceable parameter. Use a single percent sign (%) to carry out the for command at the command prompt. Use double percent signs (%%) to carry out the for command within a batch file. Variables are case sensitive, and they must be represented with an alphabetical value such as %A, %B, or %C. |
| (<Set>) | Required. Specifies one or more files, directories, or text strings, or a range of values on which to run the command. The parentheses are required. |
| <Command> | Required. Specifies the command that you want to carry out on each file, directory, or text string, or on the range of values included in <i>Set</i> . |
| <CommandLineOptions> | Specifies any command-line options that you want to use with the specified command. |
| /? | Displays help at the command prompt. |

Remarks

- Using **for**

You can use the **for** command within a batch file or directly from the command prompt.

- Using batch parameters

The following attributes apply to the **for** command:

- The **for** command replaces %*Variable* or %%*Variable* with each text string in the specified set until the specified command processes all of the files.
- Variable names are case sensitive, global, and no more than 52 can be active at a time.
- To avoid confusion with the batch parameters %0 through %9, you can use any character for *Variable* except the numerals 0 through 9. For simple batch files, a single character such as %%f will work.
- You can use multiple values for *Variable* in complex batch files to distinguish different replaceable

variables.

- Specifying a group of files

The *Set* parameter can represent a single group of files or several groups of files. You can use wildcard characters (***** and **?**) to specify a file set. The following are valid file sets:

```
(*.doc)
(*.doc *.txt *.me)
(jan*.doc jan*.rpt feb*.doc feb*.rpt)
(ar??1991.* ap??1991.*)
```

When you use the **for** command, the first value in *Set* replaces **%Variable** or **%%Variable**, and then the specified command processes this value. This continues until all of the files (or groups of files) that correspond to the *Set* value are processed.

- Using the **in** and **do** keywords

In and **do** are not parameters, but you must use them with **for**. If you omit either of these keywords, an error message appears.

- Using additional forms of **for**

If command extensions are enabled (that is the default), the following additional forms of **for** are supported:

- Directories only

If *Set* contains wildcard characters (***** or **?**), the specified *Command* executes for each directory (instead of a set of files in a specified directory) that matches *Set*.

The syntax is:

```
for /d {%%}%<Variable> in (<Set>) do <Command> [<CommandLineOptions>]
```

- Recursive

Walks the directory tree that is rooted at *Drive:Path* and executes the **for** statement in each directory of the tree. If no directory is specified after **/r**, the current directory is used as the root directory. If *Set* is just a single period (.), it only enumerates the directory tree.

The syntax is:

```
for /r [[<Drive>:]<Path>] {%%}%<Variable> in (<Set>) do <Command> [<CommandLineOptions>]
```

- Iterating a range of values

Use an iterative variable to set the starting value (*Start#*) and then step through a set range of values until the value exceeds the set ending value (*End#*). **/I** will execute the iterative by comparing *Start#* with *End#*. If *Start#* is less than *End#* the command will execute. When the iterative variable exceeds *End#*, the command shell exits the loop. You can also use a negative *Step#* to step through a range in decreasing values. For example, (1,1,5) generates the sequence 1 2 3 4 5 and (5,-1,1) generates the sequence 5 4 3 2 1.

The syntax is:

```
for /I {%%}%<Variable> in (<Start#>,<Step#>,<End#>) do <Command> [<CommandLineOptions>]
```

- o Iterating and file parsing

Use file parsing to process command output, strings, and file content. Use iterative variables to define the content or strings that you want to examine, and use the various *ParsingKeywords* options to further modify the parsing. Use the *ParsingKeywords* token option to specify which tokens should be passed as iterative variables. Note that when used without the token option, **/f** will only examine the first token.

File parsing consists of reading the output, string, or file content, and then breaking it into individual lines of text and parsing each line into zero or more tokens. The **for** loop is then called with the iterative variable value set to the token. By default, **/f** passes the first blank separated token from each line of each file. Blank lines are skipped.

The syntaxes are:

```
for /f ["<ParsingKeywords>"] {%%}%<Variable> in (<Set>) do <Command> [<CommandLineOptions>]
for /f ["<ParsingKeywords>"] {%%}%<Variable> in ("<LiteralString>") do <Command>
[<CommandLineOptions>]
for /f ["<ParsingKeywords>"] {%%}%<Variable> in ('<Command>') do <Command>
[<CommandLineOptions>]
```

The *Set* argument specifies one or more file names. Each file is opened, read, and processed before moving to the next file in *Set*. To override the default parsing behavior, specify *ParsingKeywords*. This is a quoted string that contains one or more keywords to specify different parsing options.

If you use the **usebackq** option, use one of the following syntaxes:

```
for /f ["usebackq <ParsingKeywords>"] {%%}%<Variable> in (<Set>) do <Command>
[<CommandLineOptions>]
for /f ["usebackq <ParsingKeywords>"] {%%}%<Variable> in ("<LiteralString>") do <Command>
[<CommandLineOptions>]
for /f ["usebackq <ParsingKeywords>"] {%%}%<Variable> in ('<Command>') do <Command>
[<CommandLineOptions>]
```

The following table lists the parsing keywords that you can use for *ParsingKeywords*.

| KEYWORD | DESCRIPTION |
|-------------------|--|
| eol= <c> | Specifies an end of line character (just one character). |
| skip= <N> | Specifies the number of lines to skip at the beginning of the file. |
| delims= <xxx> | Specifies a delimiter set. This replaces the default delimiter set of space and tab. |
| tokens= <X,Y,M-N> | Specifies which tokens from each line are to be passed to the for loop for each iteration. As a result, additional variable names are allocated. <i>M-N</i> specifies a range, from the <i>M</i> th through the <i>N</i> th tokens. If the last character in the tokens= string is an asterisk (*), an additional variable is allocated, and it receives the remaining text on the line after the last token that is parsed. |

| KEYWORD | DESCRIPTION |
|----------|--|
| usebackq | Specifies to execute a back-quoted string as a command, and a single-quoted string as a literal string command. Also, allows file names in <i>Set</i> to be enclosed in quotation marks. |

- Variable substitution

The following table lists optional syntax (for any variable *I*).

| VARIABLE WITH MODIFIER | DESCRIPTION |
|------------------------|--|
| %~I | Expands %I which removes any surrounding quotation marks (" "). |
| %~fI | Expands %I to a fully qualified path name. |
| %~dI | Expands %I to a drive letter only. |
| %~pI | Expands %I to a path only. |
| %~nI | Expands %I to a file name only. |
| %~xI | Expands %I to a file name extension only. |
| %~sI | Expands path to contain short names only. |
| %~aI | Expands %I to the file attributes of file. |
| %~tI | Expands %I to the date and time of file. |
| %~zI | Expands %I to the size of the file. |
| %~\$PATH:I | Searches the directories listed in the PATH environment variable and expands %I to the fully qualified name of the first directory found. If the environment variable name is not defined or the file is not found by the search, this modifier expands to the empty string. |

The following table lists modifier combinations that you can use to get compound results.

| VARIABLE WITH COMBINED MODIFIERS | DESCRIPTION |
|----------------------------------|---|
| %~dpi | Expands %I to a drive letter and path only. |
| %~nxi | Expands %I to a file name and extension only. |
| %~fsI | Expands %I to a full path name with short names only. |
| %~dp\$PATH:I | Searches the directories that are listed in the PATH environment variable for %I and expands to the drive letter and path of the first one found. |

| VARIABLE WITH COMBINED MODIFIERS | DESCRIPTION |
|----------------------------------|--|
| %~ftzal | Expands %l to an output line that is like dir . |

In the above examples, you can replace %l and PATH with other valid values. A valid **for** variable name terminates the %~ syntax.

By using uppercase variable names such as %l, you can make your code more readable and avoid confusion with the modifiers, which are not case sensitive.

- Parsing a string

You can use the **for /f** parsing logic on an immediate string by wrapping *Set* in single quotes--for example, ('Set'). *Set* is treated as a single line of input from a file, and then it is parsed.

- Parsing output

You can use the **for /f** command to parse the output of a command by making a back-quoted string from the *Set* between the parentheses. It is treated as a command line, which is passed to a child Cmd.exe. The output is captured into memory and parsed as if it is a file.

Examples

To use **for** in a batch file, use the following syntax:

```
for {%%|}%<Variable> in (<Set>) do <Command> [<CommandLineOptions>]
```

To display the contents of all the files in the current directory that have the extension .doc or .txt by using the replaceable variable %f, type:

```
for %f in (*.doc *.txt) do type %f
```

In the preceding example, each file that has the .doc or .txt extension in the current directory is substituted for the %f variable until the contents of every file are displayed. To use this command in a batch file, replace every occurrence of %f with %%f. Otherwise, the variable is ignored and an error message is displayed.

To parse a file, ignoring commented lines, type:

```
for /f "eol=; tokens=2,3* delims=," %i in (myfile.txt) do @echo %i %j %k
```

This command parses each line in Myfile.txt. It ignores lines that begin with a semicolon and passes the second and third token from each line to the **for** body (tokens are delimited by commas or spaces). The body of the **for** statement references %i to get the second token, %j to get the third token, and %k to get all of the remaining tokens. If the file names that you supply contain spaces, use quotation marks around the text (for example, "File Name"). To use quotation marks, you must use **usebackq**. Otherwise, the quotation marks are interpreted as defining a literal string to parse.

%i is explicitly declared in the **for** statement. %j and %k are implicitly declared by using **tokens=**. You can use **tokens=** to specify up to 26 tokens, provided that it does not cause an attempt to declare a variable higher than the letter "z" or "Z."

The following example enumerates the environment variable names in the current environment. To parse the output of a command by placing *Set* between the parentheses, type:

```
for /f "usebackq delims==" %i in ('set') do @echo %i
```

Additional references

[Command-Line Syntax Key](#)

forfiles

8/21/2018 • 3 minutes to read • [Edit Online](#)

Selects and executes a command on a file or set of files. This command is useful for batch processing.

For examples of how to use this command, see [Examples](#).

Syntax

```
forfiles [/p <Path>] [/m <SearchMask>] [/s] [/c "<Command>"] [/d [{+|-}][{<Date>|<Days>}]]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------------------|--|
| /p <Path> | Specifies the path from which to start the search. By default, searching starts in the current working directory. |
| /m <SearchMask> | Searches files according to the specified search mask. The default search mask is *.*. |
| /s | Instructs the forfiles command to search into subdirectories recursively. |
| /c "<Command>" | Runs the specified command on each file. Command strings should be enclosed in quotation marks. The default command is " cmd /c echo @file ". |
| /d [{+ -}][{<Date> <Days>}] | Selects files with a last modified date within the specified time frame. <ul style="list-style-type: none">- Selects files with a last modified date later than or equal to (+) or earlier than or equal to (-) the specified date, where <i>Date</i> is in the format MM/DD/YYYY.- Selects files with a last modified date later than or equal to (+) the current date plus the number of days specified, or earlier than or equal to (-) the current date minus the number of days specified.- Valid values for <i>Days</i> include any number in the range 0–32,768. If no sign is specified, + is used by default. |
| /? | Displays help at the command prompt. |

Remarks

- **Forfiles** is most commonly used in batch files.
- **Forfiles /s** is similar to **dir /s**.
- You can use the following variables in the command string as specified by the **/c** command-line option.

| VARIABLE | DESCRIPTION |
|----------|---|
| @FILE | File name. |
| @FNAME | File name without extension. |
| @EXT | File name extension. |
| @PATH | Full path of the file. |
| @RELPATH | Relative path of the file. |
| @ISDIR | Evaluates to TRUE if a file type is a directory. Otherwise, this variable evaluates to FALSE. |
| @FSIZE | File size, in bytes. |
| @FDATE | Last modified date stamp on the file. |
| @FTIME | Last modified time stamp on the file. |

- With **forfiles**, you can run a command on or pass arguments to multiple files. For example, you could run the **type** command on all files in a tree with the .txt file name extension. Or you could execute every batch file (*.bat) on drive C, with the file name "Myinput.txt" as the first argument.
- With **forfiles**, you can do any of the following:
 - Select files by an absolute date or a relative date by using the **/d** parameter.
 - Build an archive tree of files by using variables such as @FSIZE and @FDATE.
 - Differentiate files from directories by using the @ISDIR variable.
 - Include special characters in the command line by using the hexadecimal code for the character, in **0xHH** format (for example, 0x09 for a tab).
- **Forfiles** works by implementing the **recurse subdirectories** flag on tools that are designed to process only a single file.

Examples

To list all of the batch files on drive C, type:

```
forfiles /p c:\ /s /m *.bat /c "cmd /c echo @file is a batch file"
```

To list all of the directories on drive C, type:

```
forfiles /p c:\ /s /m *.* /c "cmd /c if @isdir==TRUE echo @file is a directory"
```

To list all of the files in the current directory that are at least one year old, type:

```
forfiles /s /m *.* /d -365 /c "cmd /c echo @file is at least one year old."
```

To display the text "*File is outdated*" for each of the files in the current directory that are older than January 1, 2007, type:

```
forfiles /s /m *.* /d -01/01/2007 /c "cmd /c echo @file is outdated."
```

To list the file name extensions of all the files in the current directory in column format, and add a tab before the extension, type:

```
forfiles /s /m *.* /c "cmd /c echo The extension of @file is 0x09@ext"
```

Additional references

[Command-Line Syntax Key](#)

Format

10/24/2017 • 5 minutes to read • [Edit Online](#)

Applies To: Windows 10, Windows Server 2016

Formats a disk to accept Windows files.

For examples of how to use this command, see [Examples](#).

Syntax

```
format <Volume> [/fs:{FAT|FAT32|NTFS}] [/v:<Label>] [/q] [/a:<UnitSize>] [/c] [/x] [/p:<Passes>]
format <Volume> [/v:<Label>] [/q] [/f:<Size>] [/p:<Passes>]
format <Volume> [/v:<Label>] [/q] [/t:<Tracks> /n:<Sectors>] [/p:<Passes>]
format <Volume> [/v:<Label>] [/q] [/p:<Passes>]
format <Volume> [/q]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------|---|
| <Volume> | Specifies the mount point, volume name, or drive letter (followed by a colon) of the drive that you want to format. If you do not specify any of the following command-line options, format uses the volume type to determine the default format for the disk. |
| /fs:{FAT | FAT32 |
| /v:<Label> | Specifies the volume label. If you omit the /v command-line option or use it without specifying a volume label, format prompts you for the volume label after the formatting is complete. Use the syntax /v: to prevent the prompt for a volume label. If you use a single format command to format more than one disk, all of the disks will be given the same volume label. |
| /a:<UnitSize> | <p>Specifies the allocation unit size to use on FAT, FAT32, or NTFS volumes. If you do not specify <i>UnitSize</i>, it is chosen based on volume size. Default settings are strongly recommended for general use. The following list presents valid values for NTFS, FAT, and FAT32 <i>UnitSize</i>:</p> <p>512 1024 2048 4096 8192 16K 32K 64K</p> <p>FAT and FAT32 also support 128K and 256K for a sector size greater than 512 bytes.</p> |

| PARAMETER | DESCRIPTION |
|--------------|--|
| /q | Performs a quick format. Deletes the file table and the root directory of a previously formatted volume, but does not perform a sector-by-sector scan for bad areas. You should use the /q command-line option to format only previously formatted volumes that you know are in good condition. Note that /q overrides /p . |
| /f:<Size> | Specifies the size of the floppy disk to format. When possible, use this command-line option instead of the /t and /n command-line options. Windows accepts the following values for size: <ul style="list-style-type: none"> - 1440 or 1440k or 1440kb - 1.44 or 1.44m or 1.44mb - 1.44-MB, double-sided, quadruple-density, 3.5-inch disk |
| /t:<Tracks> | Specifies the number of tracks on the disk. When possible, use the /f command-line option instead. If you use the /t option, you must also use the /n option. These options together provide an alternative method of specifying the size of the disk that is being formatted. This option is not valid with the /f option. |
| /n:<Sectors> | Specifies the number of sectors per track. When possible, use the /f command-line option instead of /n . If you use /n , you must also use /t . These two options together provide an alternative method of specifying the size of the disk that is being formatted. This option is not valid with the /f option. |
| /p:<Passes> | Zeros every sector on the volume for the number of passes specified. This option is not valid with the /q option. |
| /c | NTFS only. Files created on the new volume will be compressed by default. |
| /x | Causes the volume to dismount, if necessary, before it is formatted. Any open handles to the volume will no longer be valid. |
| /? | Displays help at the command prompt. |

Remarks

- Administrative credentials

You must be a member of the Administrators group to format a hard drive.

- Using **format**

The **format** command creates a new root directory and file system for the disk. It can also check for bad areas on the disk, and it can delete all data on the disk. To be able to use a new disk, you must first use this command to format the disk.

- Adding a volume label

After formatting a floppy disk, **format** displays the following message:

```
Volume label (11 characters, ENTER for none)?
```

To add a volume label, type up to 11 characters (including spaces). If you do not want to add a volume label to the disk, press ENTER.

- Formatting a hard disk

NOTE

You must be a member of the Administrators group to format a hard disk.

When you use the **format** command to format a hard disk, a warning message similar to the following displays:

```
WARNING, ALL DATA ON NON-REMOVABLE DISK
DRIVE x: WILL BE LOST!
Proceed with Format (Y/N)? _
```

To format the hard disk, press Y; if you do not want to format the disk, press N.

- Unit size

FAT file systems restrict the number of clusters to no more than 65526. FAT32 file systems restrict the number of clusters to between 65527 and 4177917.

NTFS compression is not supported for allocation unit sizes above 4096.

NOTE

Format will immediately stop processing if it determines that the previous requirements cannot be met using the specified cluster size.

- **Format** messages

When formatting is complete, **format** displays messages that show the total disk space, the spaces marked as defective, and the space available for your files.

- Quick formatting

You can speed up the formatting process by using the **/q** command-line option. Use this option only if there are no bad sectors on your hard disk.

- Using **format** with a reassigned drive or a network drive

You should not use the **format** command on a drive that was prepared by using the **subst** command. You cannot format disks over a network.

- **Format** exit codes

The following table lists each exit code and a brief description of its meaning.

| EXIT CODE | DESCRIPTION |
|-----------|--------------------------------------|
| 0 | The format operation was successful. |
| 1 | Incorrect parameters were supplied. |

| EXIT CODE | DESCRIPTION |
|-----------|--|
| 4 | A fatal error occurred (which is any error other than 0, 1, or 5). |
| 5 | The user pressed N in response to the prompt "Proceed with Format (Y/N)?" to stop the process. |

You can check these exit codes by using the ERRORLEVEL environment variable with the **if** batch command.

- Using **format** at the Recovery Console

The **format** command, with different parameters, is available from the Recovery Console.

Examples

To format a new floppy disk in drive A using the default size, type:

```
format a:
```

To perform a quick format operation on a previously formatted floppy disk in drive A, type:

```
format a: /q
```

To format a floppy disk in drive A and assign it the volume label "DATA," type:

```
format a: /v:DATA
```

Additional references

[Command-Line Syntax Key](#)

freedisk

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Checks to see if the specified amount of disk space is available before continuing with an installation process.

Syntax

```
freedisk [/s <computer> [/u [<Domain>\]<User> [/p [<Password>]]]] [/d <Drive>] [<Value>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| /s | Specifies the name or IP address of a remote computer (do not use backslashes). The default is the local computer. This parameter applies to all files and folders specified in the command. |
| /u [N] | Runs the script with the permissions of the specified user account. The default is system permissions. |
| /p [] | Specifies the password of the user account that is specified in /u . |
| /d | Specifies the drive for which you want to find out the availability of free space. You must specify for a remote computer. |
| | Checks for a specific amount of free disk space. You can specify in bytes, KB, MB, GB, TB, PB, EB, ZB or YB. |

Remarks

- Using the **/s**, **/u**, and **/p** command-line options are available only when you use **/s**. You must use **/p** with **/u** to provide the user's password.
- for unattended installations, you can use **freedisk** in installation batch files to check for the prerequisite amount of free space before continuing with the installation.
- When you use **freedisk** in a batch file, it returns a **0** if there is enough space and a **1** if there is not enough space. ## Examples To determine whether there are at least 50 MB of free space available on drive C:, type:

```
freedisk 50mb
```

 Output similar to the following appears on the screen:

```
INFO: The specified 52,428,800 byte(s) of free space is available on current drive. ## additional references
```

[Command-Line Syntax Key](#)

Fsutil

8/21/2018 • 4 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows 10, Windows Server 2012 R2, Windows 8.1, Windows Server 2012, Windows 8, Windows Server 2008 R2, Windows 7

Performs tasks that are related to file allocation table (FAT) and NTFS file systems, such as managing reparse points, managing sparse files, or dismounting a volume. If it is used without parameters, **Fsutil** displays a list of supported subcommands.

NOTE

You must be logged on as an administrator or a member of the Administrators group to use Fsutil. The Fsutil command is quite powerful and should be used only by advanced users who have a thorough knowledge of Windows operating systems.

You have to enable Windows Subsystem for Linux before you can run **Fsutil**. Run the following command as Administrator in PowerShell to enable this optional feature:

```
Enable-WindowsOptionalFeature -Online -FeatureName Microsoft-Windows-Subsystem-Linux
```

You'll be prompted to restart your computer once it's installed. After your computer restarts, you'll be able to run **Fsutil** as an administrator.

Parameters

| SUBCOMMAND | DESCRIPTION |
|----------------------------------|--|
| Fsutil 8dot3name | Queries or changes the settings for short name behavior on the system, for example, generates 8.3 character-length file names. Removes short names for all files within a directory. Scans a directory and identifies registry keys that might be impacted if short names were stripped from the files in the directory. |
| Fsutil behavior | Queries or sets volume behavior. |
| Fsutil dirty | Queries whether the volume's dirty bit is set or sets a volume's dirty bit. When a volume's dirty bit is set, autochk automatically checks the volume for errors the next time the computer is restarted. |
| Fsutil file | Finds a file by user name (if Disk Quotas are enabled), queries allocated ranges for a file, sets a file's short name, sets a file's valid data length, sets zero data for a file, creates a new file of a specified size, finds a file ID if given the name, or finds a file link name for a specified file ID. |
| Fsutil fsinfo | Lists all drives and queries the drive type, volume information, NTFS-specific volume information, or file system statistics. |

| SUBCOMMAND | DESCRIPTION |
|-------------------------------------|---|
| Fsutil hardlink | Lists hard links for a file, or creates a hard link (a directory entry for a file). Every file can be considered to have at least one hard link. On NTFS volumes, each file can have multiple hard links, so a single file can appear in many directories (or even in the same directory, with different names). Because all of the links reference the same file, programs can open any of the links and modify the file. A file is deleted from the file system only after all links to it are deleted. After you create a hard link, programs can use it like any other file name. |
| Fsutil objectid | Manages object identifiers, which are used by the Windows operating system to track objects such as files and directories. |
| Fsutil quota | Manages disk quotas on NTFS volumes to provide more precise control of network-based storage. Disk quotas are implemented on a per-volume basis and enable both hard- and soft-storage limits to be implemented on a per-user basis. |
| Fsutil repair | Queries or sets the self-healing state of the volume. Self-healing NTFS attempts to correct corruptions of the NTFS file system online without requiring Chkdsk.exe to be run. Includes initiating on-disk verification and waiting for repair completion. |
| Fsutil reparsepoint | Queries or deletes reparse points (NTFS file system objects that have a definable attribute containing user-controlled data). Reparse points are used to extend functionality in the input/output (I/O) subsystem. They are used for directory junction points and volume mount points. They are also used by file system filter drivers to mark certain files as special to that driver. |
| Fsutil resource | Creates a Secondary Transactional Resource Manager, starts or stops a Transactional Resource Manager, displays information about a Transactional Resource Manager or modifies its behavior. |
| Fsutil sparse | Manages sparse files. A sparse file is a file with one or more regions of unallocated data in it. A program will see these unallocated regions as containing bytes with the value zero, but no disk space is used to represent these zeros. All meaningful or nonzero data is allocated, whereas all non-meaningful data (large strings of data composed of zeros) is not allocated. When a sparse file is read, allocated data is returned as stored and unallocated data is returned as zeros (by default in accordance with the C2 security requirement specification). Sparse file support allows data to be deallocated from anywhere in the file. |
| Fsutil tiering | Enables management of storage tier functions, such as setting and disabling flags and listing of tiers. |
| Fsutil transaction | Commits a specified transaction, rolls back a specified transaction, or displays info about the transaction. |

| SUBCOMMAND | DESCRIPTION |
|-------------------------------|---|
| Fsutil usn | Manages the update sequence number (USN) change journal, which provides a persistent log of all changes made to files on the volume. |
| Fsutil volume | Manages a volume. Dismounts a volume, queries to see how much free space is available on a disk, or finds a file that is using a specified cluster. |
| Fsutil wim | Provides functions to discover and manage WIM-backed files. |

See also

[Command-Line Syntax Key](#)

Fsutil 8dot3name

10/17/2017 • 3 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows 10, Windows Server 2012 R2, Windows 8.1, Windows Server 2012, Windows 8, Windows Server 2008 R2, Windows 7

Queries or changes the settings for short name (8dot3 name) behavior, which include:

- Query the current setting for the short name behavior
- Scan the specified directory path for registry keys that might be impacted if short names were stripped from the specified directory path
- Change the setting that controls the short name behavior. This setting can be applied to a specified volume or to the default volume setting.
- Remove the short names for all files within a directory

For examples of how to use this command, see [Examples](#).

Syntax

```
fsutil 8dot3name [query] [<VolumePath>]
fsutil 8dot3name [scan] [/s] [/l [<log file>] ] [/v] <DirectoryPath>
fsutil 8dot3name [set] { <DefaultValue> | <VolumePath> {1|0}}
fsutil 8dot3name [strip] [/t] [/s] [/f] [/l [<log file.>] ] [/v] <DirectoryPath>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| query [] | <p>Queries the file system for the state of the 8dot3 short name creation behavior.</p> <p>If a <i>VolumePath</i> is not specified as a parameter, the default 8dot3name creation behavior setting for all volumes is displayed.</p> |
| scan | <p>Scans the files that are located in the specified <i>DirectoryPath</i> for registry keys that might be impacted if 8dot3 short names were stripped from the file names.</p> |

| PARAMETER | DESCRIPTION |
|-----------|--|
| set { } | <p>Changes the file system behavior for 8dot3 name creation in the following instances:</p> <ul style="list-style-type: none"> When <i>DefaultValue</i> is specified, the registry key, HKLM\System\CurrentControlSet\Control\FileSystem\NtfsDisable8dot3NameCreationNtfsDisable8dot3NameCreationNtfsDisable8dot3NameCreation, will be set to the <i>DefaultValue</i>. <p>The <i>DefaultValue</i> can have the following values:</p> <ul style="list-style-type: none"> 0: Enables 8dot3 name creation for all volumes on the system. 1: Disables 8dot3 name creation for all volumes on the system. 2: Sets 8dot3 name creation on a per volume basis. 3: Disables 8dot3 name creation for all volumes except the system volume. <ul style="list-style-type: none"> When a <i>VolumePath</i> is specified, the specified volumes on disk flag 8dot3name properties will be set to enable 8dot3 name creation for a specified volume (0) or set to disable 8dot3 name creation on the specified volume (1). <p>You must set the default file system behavior for 8dot3 name creation to the value 2 before you can enable or disable 8dot3 name creation for a specified volume.</p> |
| strip | <p>Removes the 8dot3 file names for all files that are located in the specified <i>DirectoryPath</i>. The 8dot3 file name is not removed for any files where the <i>DirectoryPath</i> combined with the file name contains more than 260 characters.</p> <p>This command lists, but does not modify the registry keys that point to the files that had 8dot3 file names permanently removed.</p> <p>For more information about the effects of permanently removing the 8dot3 file names from files, see Remarks.</p> |
| | <p>Specifies the drive name followed by a colon or the GUID in the format Volume{GUID}.</p> |
| /f | <p>Specifies that all files that are located in the specified <i>DirectoryPath</i> will have the 8dot3 file names removed even if there are registry keys that point to files using the 8dot3 file name. In this case, the operation removes the 8dot3 file names, but does not modify any registry keys that point to the files that are using the 8dot3 file names. Warning: It is recommended that you back up your directory or volume prior to using the /f parameter because it may lead to unexpected application failures, including the inability to uninstall programs.</p> |

| PARAMETER | DESCRIPTION |
|-----------|--|
| /l [] | <p>Specifies a log file where information will be written.</p> <p>If the /l parameter is not specified, all information is written to the default log file:</p> <p>%temp%\8dot3_removal_log@(GMT YYYY-MM-DD HH-MM-SS).log</p> |
| /s | Specifies that the operation should be applied to the subdirectories of the specified <i>DirectoryPath</i> . |
| /t | Specifies that the removal of 8dot3 file names should be run in test mode. All operations except the actual removal of the 8dot3 file names are performed. You can use test mode to discover which registry keys point to files that use the 8dot3 file names. |
| /v | Specifies that all information that is written to the log file is also displayed on the command-line. |

Remarks

- Permanently removing 8dot3 file names and not modifying registry keys that point to the 8dot3 file names may lead to unexpected application failures, including the inability to uninstall an application. It is recommended you first back up your directory or volume before you attempt to remove 8dot3 file names.

Examples

To query for the disable 8dot3 name behavior for a disk volume that is specified with the GUID, {928842df-5a01-11de-a85c-806e6f6e6963}, type:

```
fsutil 8dot3name query Volume{928842df-5a01-11de-a85c-806e6f6e6963}
```

You can also query the 8dot3 name behavior by using the **behavior** subcommand.

To remove 8dot3 file names in the *D:\MyData* directory and all subdirectories, while writing the information to the log file that is specified as *mylogfile.log*, type:

```
fsutil 8dot3name scan /l mylogfile.log /s d:\MyData
```

Additional references

[Command-Line Syntax Key](#)

[Fsutil](#)

[Fsutil 8dot3name](#)

[Fsutil behavior](#)

Fsutil behavior

10/17/2017 • 9 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows 10, Windows Server 2012 R2, Windows 8.1, Windows Server 2012, Windows 8, Windows Server 2008 R2, Windows 7

Queries or sets NTFS volume behavior, which includes:

- The creation of 8.3 character-length file names
- Extended character use in 8.3 character-length short file names on NTFS volumes
- The updating of the Last Access Time stamp when directories are listed on NTFS volumes
- The frequency with which quota events are written to the system log and to NTFS paged pool and NTFS non-paged pool memory cache levels
- The size of the master file table zone (MFT Zone)
- Silent deletion of data when the system encounters corruption on an NTFS volume.
- File-delete notification (also known as trim or unmap)

For examples of how to use this command, see [Examples](#).

Syntax

```
fsutil behavior query {allowextchar | bugcheckoncorrupt | disable8dot3 [<VolumePath>] | disablecompression |  
disablecompressionlimit | disableencryption | disablefilemetadataoptimization | disablelastaccess |  
disablespotcorruptionhandling | disabletxf | disablewriteautotiering | encryptpagingfile | mftzone |  
memoryusage | quotanotify | symlinkevaluation | disabledeletenotify}
```

```
fsutil behavior set {allowextchar {1|0} | bugcheckoncorrupt {1|0} | disable8dot3 [ <Value> | [<VolumePath>  
{1|0}] ] | disablecompression {1|0} | disablecompressionlimit {1|0} | disableencryption {1|0} |  
disablefilemetadataoptimization {1|0} | disablelastaccess {1|0} | disablespotcorruptionhandling {1|0} |  
disabletxf {1|0} | disablewriteautotiering {1|0} | encryptpagingfile {1|0} | mftzone <Value> | memoryusage  
<Value> | quotanotify <Frequency> | symlinkevaluation <SymbolicLinkType> | disabledeletenotify {1|0}}
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------------|--|
| query | Queries the file system behavior parameters. |
| set | Changes the file system behavior parameters. |
| allowextchar {1 0} | <p>Allows (1) or disallows (0) characters from the extended character set (including diacritic characters) to be used in 8.3 character-length short file names on NTFS volumes.</p> <p>You must restart your computer for this parameter to take effect.</p> |

| PARAMETER | DESCRIPTION |
|---------------------------------------|--|
| Bugcheckoncorrupt {1 0} | <p>Allows (1) or disallows (0) generation of a bug check when there is corruption on an NTFS volume. This feature can be used to prevent NTFS from silently deleting data when used with the Self-Healing NTFS feature.</p> <p>You must restart your computer for this parameter to take effect.</p> <p>This parameter applies to: Windows Server 2008 R2 and Windows 7 .</p> |
| disable8dot3 [] {1 0} | <p>Disables (1) or enables (0) the creation of 8.3 character-length file names on FAT- and NTFS-formatted volumes. Optionally, prefix with the <i>VolumePath</i> specified as a drive name followed by a colon or GUID.</p> |
| disablecompression {1 0} | <p>Disables (1) or enables (0) NTFS compression.</p> <p>You must restart your computer for this parameter to take effect.</p> |
| disablecompressionlimit {1 0} | <p>Disables (1) or enables (0) NTFS compression limit on NTFS volume. When a compressed file reaches a certain level of fragmentation, rather than failing to extend the file, NTFS stops compressing additional extents of the file. This was done to allow compressed files to be larger than they normally would be. Setting this value to TRUE disables this feature which will limit the size of compressed files on the system. We do not recommend disabling this feature.</p> <p>You must restart your computer for this parameter to take effect.</p> |
| disableencryption {1 0} | <p>Disables (1) or enables (0) the encryption of folders and files on NTFS volumes.</p> <p>You must restart your computer for this parameter to take effect.</p> |
| disablefilemetadataoptimization {1 0} | <p>Disables (1) or enables (0) file metadata optimization. NTFS has a limit on how many extents a given file can have. Compressed and spares files can become very fragmented. By default, NTFS periodically compacts its internal metadata structures to allow for more fragmented files. Setting this value to TRUE disables this internal optimization. We do not recommend disabling this feature.</p> <p>You must restart your computer for this parameter to take effect.</p> |
| disablelastaccess {1 0} | <p>Disables (1) or enables (0) updates to the Last Access Time stamp on each directory when directories are listed on an NTFS volume.</p> <p>You must restart your computer for this parameter to take effect.</p> |

| PARAMETER | DESCRIPTION |
|-------------------------------------|--|
| disablespotcorruptionhandling {1 0} | <p>Disables (1) or enables (0) spot corruption handling. One of the new features introduced in Windows 8 and Windows Server 2012 is a new high availability form of CHKDSK. This feature allows system administrators to run CHKDSK to analyze the state of a volume without taking it offline. We do not recommend disabling this feature.</p> <p>You must restart your computer for this parameter to take effect.</p> |
| disabletxf {1 0} | <p>Disables (1) or enables (0) txf on the specified NTFS volume. TxF is an NTFS feature that provides transaction like semantics to file system operations. TxF is presently deprecated though the functionality is still available. We do not recommend disabling this feature on the C: volume.</p> <p>You must restart your computer for this parameter to take effect.</p> |
| disablewriteautotiering {1 0} | <p>Disables ReFS v2 auto tiering logic for tiered volumes.</p> <p>You must restart your computer for this parameter to take effect.</p> |
| encryppagingfile {1 0} | <p>Encrypts (1) or does not encrypt (0) the memory paging file in the Windows operating system.</p> <p>You must restart your computer for this parameter to take effect.</p> |
| mftzone | <p>Sets the size of the MFT Zone, and is expressed as a multiple of 200MB units. Set <i>Value</i> to a number from 1 (default is 200 MB) to 4 (maximum is 800 MB).</p> <p>You must restart your computer for this parameter to take effect.</p> |
| memoryusage | <p>Configures the internal cache levels of NTFS paged-pool memory and NTFS nonpaged-pool memory. Set to 1 or 2. When set to 1 (the default), NTFS uses the default amount of paged-pool memory. When set to 2, NTFS increases the size of its lookaside lists and memory thresholds. (A lookaside list is a pool of fixed-size memory buffers that the kernel and device drivers create as private memory caches for file system operations, such as reading a file.)</p> <p>You must restart your computer for this parameter to take effect.</p> |
| quotanotify | <p>Configures how frequently NTFS quota violations are reported in the system log. Valid values for are in the range 0 – 4294967295. The default frequency is 3600 seconds (one hour).</p> <p>You must restart your computer for this parameter to take effect.</p> |

| PARAMETER | DESCRIPTION |
|---------------------|--|
| symlinkevaluation | <p>Controls the kind of symbolic links that can be created on a computer. Valid choices are:</p> <ol style="list-style-type: none"> 1. Local to local symbolic links, L2L:{0 1} 2. Local to remote symbolic links, L2R:{0 1} 3. Remote to local symbolic links, R2R:{0 1} 4. Remote to remote symbolic links, R2L:{0 1} |
| DisableDeleteNotify | <p>Disables (1) or enables (0) delete notifications. Delete notification (also known as trim or unmap) is a feature that will notify the underlying storage device of clusters that have been freed due to a file delete operation. In addition:</p> <ul style="list-style-type: none"> - For systems using ReFS v2, trim is disabled by default. Applies to Windows Server 2016. - For systems using ReFS v1, trim is enabled by default. Applies to Windows Server 2012, Windows Server 2012 R2, and Windows Server 2016. - For systems using NTFS, trim is enabled by default unless an administrator disables it. - If your hard disk drive or SAN reports that it does not support trim, then your hard disk drive and SANs will not get trim notifications. - Enabling or disabling does not require a restart. - Trim is effective when the next unmap command is issued. - Existing inflight IO are not impacted by the registry change. - Does not require any service restart when you enable or disable trim. <p>This parameter was introduced in Windows Server 2008 R2 and Windows 7.</p> |

Remarks

- The MFT Zone is a reserved area that enables the master file table (MFT) to expand as needed to prevent MFT fragmentation. If the average file size on the volume is 2 KB or less, it can be beneficial to set the **mftzone** value to 2. If the average file size on the volume is 1 KB or less, it can be beneficial to set the **mftzone** value to 4.
- When **disable8dot3** is set to 0, every time you create a file with a long file name, NTFS creates a second file entry that has an 8.3 character-length file name. When NTFS creates files in a directory, it must look up the 8.3 character-length file names that are associated with the long file names. This parameter updates the **HKLM\SYSTEM\CurrentControlSet\Control\FileSystem\NtfsDisable8dot3NameCreation** registry key.
- The **allowextchar** parameter updates the **HKLM\SYSTEM\CurrentControlSet\Control\FileSystem\NtfsAllowExtendedCharacterIn8dot3Name** registry key.

The **disablelastaccess** parameter reduces the impact of logging updates to the Last Access Time stamp on files and directories. Disabling the **Last Access Time** feature improves the speed of file and directory access. This parameter updates the **HKLM\SYSTEM\CurrentControlSet\Control\FileSystem\NtfsDisableLastAccessUpdate** registry key.

Notes:

- File-based **Last Access Time** queries are accurate even if all on-disk values are not current. NTFS returns the correct value on queries because the accurate value is stored in memory.
- One hour is the maximum amount of time that NTFS defers updating **Last Access Time** on disk. If NTFS updates other file attributes such as **Last Modify Time**, and a **Last Access Time** update is pending, NTFS updates **Last Access Time** with the other updates without additional performance impact.
- The **disablelastaccess** parameter can affect programs such as Backup and Remote Storage that rely on this feature.
- Increasing physical memory does not always increase the amount of paged pool memory available to NTFS. Setting **memoryusage** to **2** raises the limit of paged pool memory. This might improve performance if your system is opening and closing many files in the same file set and is not already using large amounts of system memory for other applications or for cache memory. If your computer is already using large amounts of system memory for other applications or for cache memory, increasing the limit of NTFS paged and non-paged pool memory reduces the available pool memory for other processes. This might reduce overall system performance. This parameter updates the **HKLM\SYSTEM\CurrentControlSet\Control\FileSystem\NtfsMemoryUsage** registry key.
- The value specified in the **mftzone** parameter is an approximation of the initial size of the MFT plus the MFT Zone on a new volume, and it is set at mount time for each file system. As space on the volume is used, NTFS adjusts the space reserved for future MFT growth. If the MFT Zone is already large, the full MFT Zone size is not reserved again. Because the MFT Zone is based on the contiguous range past the end of the MFT, it shrinks as the space is used.

The file system does not determine the new MFT Zone location until the current MFT Zone is completely used. Note that this never occurs on a typical system.

- Some devices may experience performance degradation when the delete notification feature is turned on. In this case, use the **disabledeletenotify** option to turn off the notification feature.

Examples

To query for the disable 8dot3 name behavior for a disk volume specified with the GUID, {928842df-5a01-11de-a85c-806e6f6e6963}, type:

```
fsutil behavior query disable8dot3 Volume{928842df-5a01-11de-a85c-806e6f6e6963}
```

You can also query the 8dot3 name behavior by using the **8dot3name** subcommand.

To query the system to see if TRIM is enabled or not, type:

```
fsutil behavior query DisableDeleteNotify
```

This yields an output similar to this:

```
NTFS DisableDeleteNotify = 1
ReFS DisableDeleteNotify is not currently set
```

To override the default behavior for TRIM (disabledeletenotify) for ReFS v2, type:

```
fsutil behavior set DisableDeleteNotify ReFS 0
```

To override the default behavior for TRIM (disabledeletenotify) for NTFS and ReFS v1, type:

```
fsutil behavior set DisableDeleteNotify 1
```

Additional references

[Command-Line Syntax Key](#)

[Fsutil](#)

[Fsutil 8dot3name](#)

Fsutil file

10/17/2017 • 4 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows 10, Windows Server 2012 R2, Windows 8.1, Windows Server 2012, Windows 8, Windows Server 2008 R2, Windows 7

Finds a file by user name (if Disk Quotas are enabled), queries allocated ranges for a file, sets a file's short name, sets a file's valid data length, sets zero data for a file, or creates a new file.

For examples of how to use this command, see [Examples](#).

Syntax

```
fsutil file [createnew] <filename> <length>
fsutil file [findbysid] <username> <directory>
fsutil file [optimizemetadata] [/A] <filename>
fsutil file [queryallocranges] offset=<offset> length=<length> <filename>
fsutil file [queryextents] [/R] <filename> [<startingvcn> [<numvcns>]]
fsutil file [queryfileid] <filename>
fsutil file [queryfilenamebyid] <volume> <fileid>
fsutil file [queryoptimizemetadata] <filename>
fsutil file [queryvaliddata] [/R] [/D] <filename>
fsutil file [seteof] <filename> <length>
fsutil file [setshortname] <filename> <shortname>
fsutil file [setvaliddata] <filename> <datalength>
fsutil file [setzerodata] offset=<offset> length=<length> <filename>
```

Parameters

| PARAMETER | DESCRIPTION |
|------------------|---|
| createnew | Creates a file of the specified name and size, with content that consists of zeroes. |
| <filename> | Specifies the full path to the file including the file name and extension, for example C:\documents\filename.txt. |
| <length> | Specifies the file's valid data length. |
| findbysid | Finds files that belong to a specified user on NTFS volumes where Disk Quotas are enabled. |
| <username> | Specifies the user's user name or logon name. |
| <directory> | Specifies the full path to the directory, for example C:\users. |
| optimizemetadata | This performs an immediate compaction of the metadata for a given file. |
| /A | Analyze file metadata before and after optimization. |

| PARAMETER | DESCRIPTION |
|-----------------------|--|
| queryallocranges | Queries the allocated ranges for a file on an NTFS volume. Useful for determining whether a file has sparse regions. |
| offset= <offset> | Specifies the start of the range that should be set to zeroes. |
| length= <length> | Specifies the length of the range (in bytes). |
| queryextents | Queries extents for a file. |
| /R | If is a reparse point, open it rather than its target. |
| <startingvcn> | Specifies first VCN to query. If omitted, start at VCN 0. |
| <numvcns> | Number of VCNs to query. If omitted or 0, query until EOF. |
| queryfileid | Queries the file ID of a file on an NTFS volume. This parameter applies to: Windows Server 2008 R2 and Windows 7 . |
| <volume> | Specifies the volume as drive name followed by a colon. |
| queryfilenamebyid | Displays a random link name for a specified file ID on an NTFS volume. Since a file can have more than one link name pointing to that file, it is not guaranteed which file link will be provided as a result of the query for the file name. This parameter applies to: Windows Server 2008 R2 and Windows 7 . |
| <fileid> | Specifies the ID of the file on an NTFS volume. |
| queryoptimizemetadata | Queries the metadata state of a file. |
| queryvaliddata | Queries the valid data length for a file. |
| /D | Display detailed valid data information. |
| seteof | Sets the EOF of the given file. |
| setshortname | Sets the short name (8.3 character-length file name) for a file on an NTFS volume. |
| <shortname> | Specifies the file's short name. |
| setvaliddata | Sets the valid data length for a file on an NTFS volume. |
| <datalength> | Specifies the length of the file in bytes. |
| setzerodata | Sets a range (specified by <i>Offset</i> and <i>Length</i>) of the file to zeroes, which empties the file. If the file is a sparse file, the underlying allocation units are decommitted. |

Remarks

- In NTFS, there are two IMPORTANT concepts of file length: the end-of-file (EOF) marker and the Valid Data Length (VDL). The EOF indicates the actual length of the file. The VDL identifies the length of valid data on disk. Any reads between VDL and EOF automatically return 0 to preserve the C2 object reuse requirement.
- The **setvaliddata** parameter is only available for administrators because it requires the Perform volume maintenance tasks (SeManageVolumePrivilege) privilege. This feature is only required for advanced multimedia and system area network scenarios. The **setvaliddata** parameter must be a positive value that is greater than the current VDL, but less than the current file size.

It is useful for programs to set a VDL when:

- Writing raw clusters directly to disk through a hardware channel. This allows the program to inform the file system that this range contains valid data that can be returned to the user.
- Creating large files when performance is an issue. This avoids the time it takes to fill the file with zeroes when the file is created or extended.

Examples

To find files that are owned by scottb on drive C, type:

```
fsutil file findbysid scottb c:\users
```

To query the allocated ranges for a file on an NTFS volume, type:

```
fsutil file queryallocranges offset=1024 length=64 c:\temp\sample.txt
```

To optimize metadata for a file, type:

```
fsutil file optimizemetadata C:\largefragmentedfile.txt
```

To query the extents for a file, type:

```
fsutil file queryextents C:\Temp\sample.txt
```

To set the EOF for a file, type:

```
fsutil file seteof C:\testfile.txt 1000
```

To set the short name for the file Longfilename.txt on drive C to Longfile.txt, type:

```
fsutil file setshortname c:\longfilename.txt longfile.txt
```

To set the valid data length to 4096 bytes for a file named Testfile.txt on an NTFS volume, type:

```
fsutil file setvaliddata c:\testfile.txt 4096
```

To set a range of a file on an NTFS volume to zeros to empty it, type:

```
fsutil file setzerodata offset=100 length=150 c:\temp\sample.txt
```

Additional references

[Command-Line Syntax Key](#)

[Fsutil](#)

Fsutil fsinfo

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows 10, Windows Server 2012 R2, Windows 8.1, Windows Server 2012, Windows 8, Windows Server 2008 R2, Windows 7

Lists all drives, queries the drive type, queries volume information, queries NTFS-specific volume information, or queries file system statistics.

For examples of how to use this command, see [Examples](#).

Syntax

```
fsutil fsinfo [drives]
fsutil fsinfo [drivetype] <VolumePath>
fsutil fsinfo [ntfsinfo] <RootPath>
fsutil fsinfo [statistics] <VolumePath>
fsutil fsinfo [volumeinfo] <RootPath>
```

Parameters

| PARAMETER | DESCRIPTION |
|------------------|--|
| drives | Lists all drives in the computer. |
| drivetype | Queries a drive and lists its type, for example CD-ROM drive. |
| ntfsinfo | Lists NTFS specific volume information for the specified volume, such as the number of sectors, total clusters, free clusters, and the start and end of the MFT Zone. |
| sectorinfo | Lists information about the hardware's sector size and alignment. |
| statistics | Lists file system statistics for the specified volume, such as metadata, log file, and MFT reads and writes. |
| volumeinfo | Lists information for the specified volume, such as the file system, and whether the volume supports case-sensitive file names, unicode in file names, disk quotas, or is a DirectAccess (DAX) volume. |
| <"VolumePath"> | Specifies the drive letter (followed by a colon). |
| <"RootPathname"> | Specifies the drive letter (followed by a colon) of the root drive. |

Examples

To list all of the drives in the computer, type:

```
fsutil fsinfo drives
```

Output similar to the following displays:

```
Drives: A:\ C:\ D:\ E:\
```

To query the drive type of drive C, type:

```
fsutil fsinfo drivetype c:
```

Possible results of the query include:

```
Unknown Drive
No such Root Directory
Removable Drive, for example floppy
Fixed Drive
Remote/Network Drive
CD-ROM Drive
Ram Disk
```

To query the volume information for volume E, type:

```
fsinfo volumeinfo e:\
```

Output similar to the following displays:

```
Volume Name :Volume
Serial Number : 0xd0b634d9
Max Component Length : 255
File System Name : NTFS
.
.
.
Supports Named Streams
Is DAX Volume
```

To query drive F for NTFS-specific volume information, type:

```
fsutil fsinfo ntfsinfo f:
```

Output similar to the following displays:

```
NTFS Volume Serial Number : 0xe660d46a60d442cb
Number Sectors :          0x00000000010ea04f
Total Clusters :          0x00000000021d409
.
.
.
Mft Zone End :            0x000000000004700
```

To query the file system's underlying hardware for sector information, type:

```
fsinfo sectorinfo d:
```

Output similar to the following displays:

```
D:\>fsutil fsinfo sectorinfo d:
LogicalBytesPerSector :          4096
PhysicalBytesPerSectorForAtomicity : 4096
.
.
.
Trim Not Supported
DAX capable
```

To query the file system statistics for drive E, type:

```
fsinfo statistics e:
```

Output similar to the following displays:

```
File System Type :    NTFS
Version :            1
UserFileReads :       75021
UserFileReadBytes :   1305244512
.
.
.
LogFileWriteBytes :   180936704
```

Additional references

[Command-Line Syntax Key Fsutil](#)

Fsutil hardlink

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows 10, Windows Server 2012 R2, Windows 8.1, Windows Server 2012, Windows 8, Windows Server 2008 R2, Windows 7

Creates a hard link between an existing file and a new file.

Syntax

```
fsutil hardlink create <NewFileName> <ExistingFileName>
fsutil hardlink list <Filename>
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------------|---|
| create | Establishes an NTFS hard link between an existing file and a new file. (An NTFS hard link is similar to a POSIX hard link.) |
| <NewFileName> | Specifies the file that you want to create a hard link to. |
| <ExistingFileName> | Specifies the file that you want to create a hard link from. |
| list | <p>Lists the hardlinks to <i>Filename</i>.</p> <p>This parameter applies to: Windows Server 2008 R2 and Windows 7 .</p> |

Remarks

- A hard link is a directory entry for a file. Every file can be considered to have at least one hard link. On NTFS volumes, each file can have multiple hard links, so a single file can appear in many directories (or even in the same directory with different names). Because all of the links reference the same file, programs can open any of the links and modify the file. A file is deleted from the file system only after all links to it have been deleted. After you create a hard link, programs can use it like any other file name.

Additional references

[Command-Line Syntax Key](#)

[Fsutil](#)

Fsutil objectid

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows 10, Windows Server 2012 R2, Windows 8.1, Windows Server 2012, Windows 8, Windows Server 2008 R2, Windows 7

Manages object identifiers (OIDs), which are internal objects used by the Distributed Link Tracking (DLT) Client service and File Replication Service (FRS) to track other objects such as files, directories, and links. Object identifiers are invisible to most programs and should never be modified.

Caution

Do not delete, set, or otherwise modify an object identifier. Deleting or setting an object identifier can result in the loss of data from portions of a file, up to and including entire volumes of data. In addition, you might cause adverse behavior in the Distributed Link Tracking (DLT) Client service and File Replication Service (FRS).

For examples of how to use this command, see [Examples](#).

Syntax

```
fsutil objectid [create] <FileName>
fsutil objectid [delete] <FileName>
fsutil objectid [query] <FileName>
fsutil objectid [set] <ObjectID> <BirthVolumeID> <BirthObjectID> <DomainID> <FileName>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------|--|
| create | Creates an object identifier if the specified file does not already have one. If the file already has an object identifier, this subcommand is equivalent to the query subcommand. |
| delete | Deletes an object identifier. |
| query | Queries an object identifier. |
| set | Sets an object identifier. |
| <ObjectID> | Sets a file-specific 16 byte hexadecimal identifier that is guaranteed to be unique within a volume. The object identifier is used by the Distributed Link Tracking (DLT) Client service and the File Replication Service (FRS) to identify files. |
| <BirthVolumeID> | Indicates the volume on which the file was located when it first obtained an object identifier. This value is a 16 byte hexadecimal identifier that is used by the DLT Client service. |
| <BirthObjectID> | Indicates the file's original object identifier (The <i>ObjectID</i> may change when a file is moved). This value is a 16 byte hexadecimal identifier that is used by the DLT Client service. |

| PARAMETER | DESCRIPTION |
|------------|---|
| <DomainID> | 16 byte hexadecimal domain identifier. This value is not currently used and must be set to all zeros. |
| <FileName> | Specifies the full path to the file including the file name and extension, for example C:\documents\filename.txt. |

Remarks

- Any file that has an object identifier also has a birth volume identifier, a birth object identifier, and a domain identifier. When you move a file, the object identifier may change, but the birth volume and birth object identifiers remain the same. This behavior enables the Windows operating system to always find a file, no matter where it has been moved.

Examples

To create an object identifier, type:

```
fsutil objectid create c:\temp\sample.txt
```

To delete an object identifier, type:

```
fsutil objectid delete c:\temp\sample.txt
```

To query an object identifier, type:

```
fsutil objectid query c:\temp\sample.txt
```

To set an object identifier, type:

```
fsutil objectid set 40dff02fc9b4d4118f120090273fa9fc f86ad6865fe8d21183910008c709d19e
40dff02fc9b4d4118f120090273fa9fc 00000000000000000000000000000000 c:\temp\sample.txt
```

Additional references

[Command-Line Syntax Key](#)

[Fsutil](#)

Fsutil quota

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows 10, Windows Server 2012 R2, Windows 8.1, Windows Server 2012, Windows 8, Windows Server 2008 R2, Windows 7

Manages disk quotas on NTFS volumes to provide more precise control of network-based storage.

For examples of how to use this command, see [Examples](#).

Syntax

```
fsutil quota [disable] <VolumePath>
fsutil quota [enforce] <VolumePath>
fsutil quota [modify] <VolumePath> <Threshold> <Limit> <UserName>
fsutil quota [query] <VolumePath>
fsutil quota [track] <VolumePath>
fsutil quota [violations]
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------|---|
| disable | Disables quota tracking and enforcement on the specified volume. |
| enforce | Enforces quota usage on the specified volume. |
| modify | Modifies an existing disk quota or creates a new quota. |
| query | Lists existing disk quotas. |
| track | Tracks disk usage on the specified volume. |
| violations | Searches the system and application logs and displays a message to indicate that quota violations have been detected or that a user has reached a quota threshold or quota limit. |
| <VolumePath> | Required. Specifies the drive name followed by a colon or the GUID in the format Volume{GUID} . |
| <Threshold> | Sets the limit (in bytes) at which warnings are issued. This parameter is required for the fsutil quota modify command. |
| <Limit> | Sets the maximum allowed disk usage (in bytes). This parameter is required for the fsutil quota modify command. |
| <UserName> | Specifies the domain or user name. This parameter is required for the fsutil quota modify command. |

Remarks

- Disk quotas are implemented on a per-volume basis, and they enable both hard and soft storage limits to be implemented on a per-user basis.
- You can use write scripts that use **fsutil quota** to set the quota limits every time you add a new user or to automatically track quota limits, compile them into a report, and automatically send them to the system administrator in e-mail.

Examples

To list existing disk quotas for a disk volume that is specified with the GUID, {928842df-5a01-11de-a85c-806e6f6e6963}, type:

```
fsutil quota query Volume{928842df-5a01-11de-a85c-806e6f6e6963}
```

To list existing disk quotas for a disk volume that is specified with the drive letter, **C:**, type:

```
Fsutil quota query C:
```

Additional references

[Command-Line Syntax Key](#)

[Fsutil](#)

Fsutil repair

10/24/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows 10, Windows Server 2012 R2, Windows 8.1, Windows Server 2012, Windows 8, Windows Server 2008 R2, Windows 7

Administers and monitors NTFS self-healing repair operations.

For examples of how to use this command, see [Examples](#).

Syntax

```
fsutil repair [enumerate] <volumePath> [<LogName>]
fsutil repair [initiate] <VolumePath> <FileReference>
fsutil repair [query] <VolumePath>
fsutil repair [set] <VolumePath> <Flags>
fsutil repair [wait][<WaitType>] <VolumePath>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------|---|
| enumerate | Enumerates the entire of a volume's corruption log. |
| <volumePath> | Specifies the volume as the drive name followed by a colon. |
| <LogName> | \$Corrupt - The set of confirmed corruptions in the volume. \$Verify - A set of potential, unverified corruptions in the volume. |
| initiate | Initiates NTFS self-healing. |
| <FileReference> | Specifies the NTFS volume-specific file ID (file reference number). The file reference includes the segment number of the file. |
| query | Queries the self-healing state of the NTFS volume. |
| set | Sets the self-healing state of the volume. |
| <Flags> | <p>Specifies the repair method to be used when setting the self-healing state of the volume.</p> <p>The Flags parameter can be set to three values:</p> <ul style="list-style-type: none">- 0x01: Enables general repair.- 0x09: Warns about potential data loss without repair.- 0x00: Disables NTFS self-healing repair operations. |

| PARAMETER | DESCRIPTION |
|------------------|---|
| state | Queries the corruption state of the system or for a given volume. |
| wait | Waits for repair(s) to complete. If NTFS has detected a problem on a volume on which it is performing repairs, this option allows the system to wait until the repair is complete before it runs any pending scripts. |
| [WaitType {0 1}] | <p>Indicates whether to wait for the current repair to complete or to wait for all repairs to complete. <i>WaitType</i> can be set to the following values:</p> <ul style="list-style-type: none"> - 0: Waits for all repairs to complete. (default value) - 1: Waits for the current repair to complete. |

Remarks

- Self-healing NTFS attempts to correct corruptions of the NTFS file system online, without requiring **Chkdsk.exe** to be run. This feature was introduced in Windows Server 2008. For more information, see [Self Healing NTFS](#).

Examples

To enumerate the confirmed corruptions of a volume, type:

```
fsutil repair enumerate C: $Corrupt
```

To enable self-healing repair on drive C, type:

```
fsutil repair set c: 1
```

To disable self-healing repair on drive C, type:

```
fsutil repair set c: 0
```

Additional references

[Command-Line Syntax Key](#)

[Fsutil](#)

[Self Healing NTFS](#)

Fsutil reparsepoint

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows 10, Windows Server 2012 R2, Windows 8.1, Windows Server 2012, Windows 8, Windows Server 2008 R2, Windows 7, Windows 2008, Windows Vista

Queries or deletes reparse points. The **fsutil reparsepoint** command is typically used by support professionals.

For examples of how to use this command, see [Examples](#).

Syntax

```
fsutil reparsepoint [query] <FileName>
fsutil reparsepoint [delete] <FileName>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| query | Retrieves the reparse point data that is associated with the file or directory identified by the specified handle. |
| delete | Deletes a reparse point from the file or directory that is identified by the specified handle, but does not delete the file or directory. |
| | Specifies the full path to the file including the file name and extension, for example C:\documents\filename.txt. |

Remarks

- Reparse points are NTFS file system objects that have a definable attribute that contains user-defined data, and they are used to extend functionality in the input/output (I/O) subsystem.
- Reparse points are used for directory junction points and volume mount points. They are also used by file system filter drivers to mark certain files as special to that driver.
- When a program sets a reparse point, it stores this data, plus a reparse tag, which uniquely identifies the data it is storing. When the file system opens a file with a reparse point, it attempts to find the associated file system filter. If the file system filter is found, the filter processes the file as directed by the reparse data. If no file system filter is found, the File open operation fails.

Examples

To retrieve reparse point data associated with C:\Server, type:

```
fsutil reparsepoint query c:\server
```

To delete a reparse point from a specified file or directory, use the following format:

```
fsutil reparsepoint delete c:\server
```

Additional references

[Command-Line Syntax Key](#)

[Fsutil](#)

Fsutil resource

10/24/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows 10, Windows Server 2012 R2, Windows 8.1, Windows Server 2012, Windows 8, Windows Server 2008 R2, Windows 7, Windows 2008, Windows Vista

Creates a Secondary Transactional Resource Manager, starts or stops a Transactional Resource Manager, or displays information about a Transactional Resource Manager and modifies the following behavior:

- Whether a default Transactional Resource Manager will clean its transactional metadata at the next mount
- The specified Transactional Resource Manager to prefer consistency over availability
- The specified Transaction Resource Manager to prefer availability over consistency
- The characteristics of a running Transactional Resource Manager

For examples of how to use this command, see [Examples](#) .

Syntax

```
fsutil resource [create] <RmRootPathname>
fsutil resource [info] <RmRootPathname>
fsutil resource [setautoreset] {true|false} <DefaultRmRootPathname>
fsutil resource [setavailable] <RmRootPathname>
fsutil resource [setconsistent] <RmRootPathname>
fsutil resource [setlog] [growth {<Containers> containers|<Percent> percent} <RmRootPathname>] [maxextents
<Containers> <RmRootPathname>] [minextents <Containers> <RmRootPathname>] [mode {full|undo} <RmRootPathname>]
[rename <RmRootPathname>] [shrink <percent> <RmRootPathname>] [size <Containers> <RmRootPathname>]
fsutil resource [start] <RmRootPathname> [<RmLogPathname> <TmLogPathname>]
fsutil resource [stop] <RmRootPathname>
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------|---|
| create | Creates a secondary Transactional Resource Manager. |
| | Specifies the full path to a Transactional Resource Manager root directory. |
| info | Displays the specified Transactional Resource Manager's information. |
| setautoreset | <p>Specifies whether a default Transactional Resource Manager will clean the transactional metadata on the next mount.</p> <ul style="list-style-type: none">- Set the setautoreset parameter to true to specify that the Transaction Resource Manager will clean the transactional metadata on the next mount, by default.- Set the setautoreset parameter to false to specify that the Transaction Resource Manager will not clean the transactional metadata on the next mount, by default. |

| PARAMETER | DESCRIPTION |
|------------------|---|
| | Specifies the drive name followed by a colon. |
| setavailable | Specifies that a Transactional Resource Manager will prefer availability over consistency. |
| setconsistent | Specifies that a Transactional Resource Manager will prefer consistency over availability. |
| setlog | Changes the characteristics of a Transactional Resource Manager that is already running. |
| growth | <p>Specifies the amount by which the Transactional Resource Manager log can grow.</p> <p>The growth parameter can be specified as follows:</p> <ul style="list-style-type: none"> - Number of containers using the format: <i>Containers</i>containers - Percentage using the format: <i>Percent</i>percent |
| | Specifies the data objects that are used by the Transactional Resource Manager. |
| maxextent | Specifies the maximum number of containers for the specified Transactional Resource Manager. |
| minextent | Specifies the minimum number of containers for the specified Transactional Resource Manager. |
| mode {full undo} | Specifies whether all transactions are logged (full) or only rolled back events are logged (undo). |
| rename | Changes the GUID for the Transactional Resource Manager. |
| shrink | Specifies percentage by which the Transactional Resource Manager log can automatically decrease. |
| size | Specifies the size of the Transactional Resource Manager as a specified number of <i>Containers</i> . |
| start | Starts the specified Transactional Resource Manager. |
| stop | Stops the specified Transactional Resource Manager. |

Examples

To set the log for the Transactional Resource Manager that is specified by c:\test, to have an automatic growth of five containers, type:

```
fsutil resource setlog growth 5 containers c:test
```

To set the log for the Transactional Resource Manager that is specified by c:\test, to have an automatic growth of two percent, type:

```
fsutil resource setlog growth 2 percent c:test
```

To specify that the default Transactional Resource Manager will clean the transactional metadata on the next mount on drive C, type:

```
fsutil resource setautoreset true c:\
```

Additional references

[Command-Line Syntax Key](#)

[Fsutil](#)

[Transactional NTFS](#)

Fsutil sparse

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows 10, Windows Server 2012 R2, Windows 8.1, Windows Server 2012, Windows 8, Windows Server 2008 R2, Windows 7

Manages sparse files.

For examples of how to use this command, see [Examples](#).

Syntax

```
fsutil sparse [queryflag] <FileName>
fsutil sparse [queryrange] <FileName>
fsutil sparse [setflag] <FileName>
fsutil sparse [setrange] <FileName> <BeginningOffset> <Length>
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|---|
| queryflag | Queries sparse. |
| queryrange | Scans a file and searches for ranges that may contain nonzero data. |
| setflag | Marks the indicated file as sparse. |
| setrange | Fills a specified range of a file with zeros. |
| | Specifies the full path to the file including the file name and extension, for example C:\documents\filename.txt. |
| | Specifies the offset within the file to mark as sparse. |
| | Specifies the length of the region in the file to be marked as sparse (in bytes). |

Remarks

- A sparse file is a file with one or more regions of unallocated data in it. A program will see these unallocated regions as containing bytes with the value zero, but no disk space is used to represent these zeros. All meaningful or nonzero data is allocated, whereas all nonmeaningful data (large strings of data that is composed of zeros) is not allocated. When a sparse file is read, allocated data is returned as stored, and unallocated data is returned, by default, as zeros, in accordance with the C2 security requirement specification. Sparse file support allows data to be deallocated from anywhere in the file.
- In a sparse file, large ranges of zeroes may not require disk allocation. Space for nonzero data will be allocated as needed when the file is written.

- Only compressed or sparse files can have zeroed ranges known to the operating system.
- If the file is sparse or compressed, NTFS may deallocate disk space within the file. This sets the range of bytes to zeroes without extending the file size.

Examples

To mark a file named Sample.txt in the C:\Temp directory as sparse, type:

```
fsutil sparse setflag c:\temp\sample.txt
```

Additional references

[Command-Line Syntax Key](#)

[Fsutil](#)

Fsutil tiering

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows 10

Enables management of storage tier functions, such as setting and disabling flags and listing of tiers.

Syntax

```
fsutil tiering [clearflags] <volume> <flags>
fsutil tiering [queryflags] <volume>
fsutil tiering [regionlist] <volume>
fsutil tiering [setflags] <volume> <flags>
fsutil tiering [tierlist] <volume>
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|--|
| clearflags | Disables the tiering behavior flags of a volume. |
| <volume> | Specifies the volume. |
| /TrNH | For volumes with tiered storage, causes Heat gathering to be disabled. Applies to NTFS and ReFS only. |
| queryflags | Queries the tiering behavior flags of a volume. |
| regionlist | Lists the tiered regions of a volume and their respective storage tiers. |
| setflags | Enables the tiering behavior flags of a volume. |
| tierlist | Lists the storage tiers associated with a volume. |

Examples

To query the flags on volume C, type:

```
fsutil tiering queryflags C:
```

To set the flags on volume C, type:

```
fsutil tiering setflags C: /TrNH
```

To clear the flags on volume C, type:

```
fsutil tiering clearflags C: /TrNH
```

To list the regions of volume C and their respective storage tiers, type:

```
fsutil tiering regionlist C:
```

To list the tiers of volume C, type:

```
fsutil tiering tierlist C:
```

Additional references

[Command-Line Syntax Key](#)

[Fsutil](#)

Fsutil transaction

10/24/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows 10, Windows Server 2012 R2, Windows 8.1, Windows Server 2012, Windows 8, Windows Server 2008 R2, Windows 7, Windows 2008, Windows Vista

Manages NTFS transactions.

For examples of how to use this command, see [Examples](#) .

Syntax

```
fsutil transaction [commit] <GUID>
fsutil transaction [fileinfo] <Filename>
fsutil transaction [list]
fsutil transaction [query] [{Files|All}] <GUID>
fsutil transaction [rollback] <GUID>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| commit | Marks the end of a successful implicit or explicit specified transaction. |
| | Specifies the GUID value that represents a transaction. |
| fileinfo | Displays transaction information for the specified file. |
| | Specifies full path and file name. |
| list | Displays a list of currently running transactions. |
| query | Displays information for the specified transaction. - If fsutil transaction query Files is specified, the file information will be displayed only for the specified transaction. - If fsutil transaction query All is specified, all information for the transaction will be displayed. |
| rollback | Rolls back a specified transaction to the beginning. |

Remarks

- Transactional NTFS was introduced in Windows Server 2008 .

Examples

To display transaction information for file c:\test.txt, type:

```
fsutil transaction fileinfo c:\test.txt
```

Additional references

[Command-Line Syntax Key](#)

[Fsutil](#)

[Transactional NTFS](#)

Fsutil usn

10/17/2017 • 4 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows 10, Windows Server 2012 R2, Windows 8.1, Windows Server 2012, Windows 8, Windows Server 2008 R2, Windows 7

Manages the update sequence number (USN) change journal.

Syntax

```
fsutil usn [createjournal] m=<MaxSize> a=<AllocationDelta> <VolumePath>
fsutil usn [deletejournal] [/D | /N] <volumepath>
fsutil usn [enablerangetracking] <volumepath> [options]
fsutil usn [enumdata] <FileRef> <LowUSN> <HighUSN> <VolumePath>
fsutil usn [queryjournal] <VolumePath>
fsutil usn [readdata] <FileName>
fsutil usn [readjournal] [c= <chunk-size> s=<file-size-threshold>] <volumepath>
```

Parameters

| PARAMETER | DESCRIPTION |
|----------------------|---|
| createjournal | Creates a USN change journal. |
| m= <MaxSize> | Specifies the maximum size, in bytes, that NTFS allocates for the change journal. |
| a= <AllocationDelta> | Specifies the size, in bytes, of memory allocation that is added to the end and removed from the beginning of the change journal. |
| <VolumePath> | Specifies the drive letter (followed by a colon). |
| deletejournal | Deletes or disables an active USN change journal. Caution: Deleting the change journal impacts the File Replication Service (FRS) and the Indexing Service, because it would require these services to perform a complete (and time-consuming) scan of the volume. This in turn negatively impacts FRS SYSVOL replication and replication between DFS link alternates while the volume is being rescanned. |
| /d | Disables an active USN change journal, and returns input/output (I/O) control while the change journal is being disabled. |
| /n | Disables an active USN change journal and returns I/O control only after the change journal is disabled. |
| enablerangetracking | Enables USN write range tracking for a volume. |
| c= <chunk-size> | Specifies the chunk size to track on a volume. |

| PARAMETER | DESCRIPTION |
|--------------------------|--|
| s= <file-size-threshold> | Specifies the file size threshold for range tracking. |
| enumdata | Enumerates and lists the change journal entries between two specified boundaries. |
| <FileRef> | Specifies the ordinal position within the files on the volume at which the enumeration is to begin. |
| <LowUSN> | Specifies the lower boundary of the range of USN values used to filter the records that are returned. Only records whose last change journal USN is between or equal to the <i>LowUSN</i> and <i>HighUSN</i> member values are returned. |
| <HighUSN> | Specifies the upper boundary of the range of USN values used to filter the files that are returned. |
| queryjournal | Queries a volume's USN data to gather information about the current change journal, its records, and its capacity. |
| readdata | Reads the USN data for a file. |
| <FileName> | Specifies the full path to the file, including the file name and extension For example: C:\documents\filename.txt |
| readjournal | Reads the USN records in the USN journal. |
| minver= <number> | Minimum Major Version of USN_RECORD to return. Default = 2. |
| maxver= <number> | Maximum Major Version of USN_RECORD to return. Default = 4. |
| startusn= <USN number> | USN to start reading the USN journal from. Default = 0. |

Remarks

- About the USN change journal

The USN change journal provides a persistent log of all changes made to files on the volume. As files, directories, and other NTFS objects are added, deleted, and modified, NTFS enters records into the USN change journal, one for each volume on the computer. Each record indicates the type of change and the object changed. New records are appended to the end of the stream.

Programs can consult the USN change journal to determine all the modifications made to a set of files. The USN change journal is much more efficient than checking time stamps or registering for file notifications. The USN change journal is enabled and used by the Indexing Service, File Replication Service (FRS), Remote Installation Services (RIS), and Remote Storage.

- Using **createjournal**

If a change journal already exists on a volume, the **createjournal** parameter updates the change journal's *MaxSize* and *AllocationDelta* parameters. This enables you to expand the number of records that an active

journal maintains without having to disable it.

- Using **m**

The change journal can grow larger than this target value, but the change journal is truncated at the next NTFS checkpoint to less than this value. NTFS examines the change journal and trims it when its size exceeds the value of *MaxSize* plus the value of *AllocationDelta*. At NTFS checkpoints, the operating system writes records to the NTFS log file that enable NTFS to determine what processing is required to recover from a failure.

- Using **a**

The change journal can grow to more than the sum of the values of *MaxSize* and *AllocationDelta* before being trimmed.

- Using **deletejournal**

Deleting or disabling an active change journal is very time consuming, because the system must access all the records in the master file table (MFT) and set the last USN attribute to 0 (zero). This process can take several minutes, and it can continue after the system restarts, if a restart is necessary. During this process, the change journal is not considered active, nor is it disabled. While the system is disabling the journal, it cannot be accessed, and all journal operations return errors. You should use extreme care when disabling an active journal, because it adversely affects other applications that are using the journal.

Examples

To create a USN change journal on drive C, type:

```
fsutil usn createjournal m=1000 a=100 c:
```

To delete an active USN change journal on drive C, type:

```
fsutil usn deletejournal /d c:
```

To enable range tracking with a specified chunk-size and file-size-threshold, type:

```
fsutil usn enablerangetracking c=16384 s=67108864 C:
```

To enumerate and list the change journal entries between two specified boundaries on drive C, type:

```
fsutil usn enumdata 1 0 1 c:
```

To query USN data for a volume on drive C, type:

```
fsutil usn queryjournal c:
```

To read the USN data for a file in the \Temp folder on drive C, type:

```
fsutil usn readdata c:\temp\sample.txt
```

To read the USN journal with a specific start USN, type:

```
fsutil usn readjournal startusn=0xF00
```

Additional references

[Command-Line Syntax Key](#)

[Fsutil](#)

Fsutil volume

10/24/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows 10, Windows Server 2012 R2, Windows 8.1, Windows Server 2012, Windows 8, Windows Server 2008 R2, Windows 7

Dismounts a volume, or queries the hard disk drive to determine how much free space is currently available on the hard disk drive or which file is using a particular cluster.

For examples of how to use this command, see [Examples](#).

Syntax

```
fsutil volume [allocationreport] <VolumePath>
fsutil volume [diskfree] <VolumePath>
fsutil volume [dismount] <VolumePath>
fsutil volume [filelayout] <VolumePath> <fileid>
fsutil volume [list]
fsutil volume [querycluster] <VolumePath> <Cluster> [<Cluster>] ... ..
```

Parameters

| PARAMETER | DESCRIPTION |
|------------------|--|
| allocationreport | Displays information about how storage is used on a given volume. |
| <VolumePath> | Specifies the drive letter (followed by a colon). |
| diskfree | Queries the hard disk drive to determine the amount of free space on it. |
| dismount | Dismounts a volume. |
| filelayout | Displays NTFS metadata for the given file. |
| <fileid> | Specifies the file id. |
| list | Lists all of the volumes on the system. |
| querycluster | <p>Finds which file is using a specified cluster. You can specify multiple clusters with the querycluster parameter.</p> <p>This parameter applies to: Windows Server 2008 R2 and Windows 7 .</p> |
| <cluster> | Specifies the logical cluster number (LCN). |

Examples

To display an allocated clusters report, type:

```
fsutil volume allocationreport C:
```

To dismount a volume on drive C, type:

```
fsutil volume dismount c:
```

To query the amount of free space of a volume on drive C, type:

```
fsutil volume diskfree c:
```

To display all the information about a specified file(s), type:

```
fsutil volume C: *  
fsutil volume C:\Windows  
fsutil volume C: 0x00040000000001bf
```

To list the volumes on disk, type:

```
fsutil volume list
```

To find the file(s) that are using the clusters, specified by the logical cluster numbers 50 and 0x2000, on drive C, type:

```
fsutil volume querycluster C: 50 0x2000
```

Additional references

[Command-Line Syntax Key](#)

[Fsutil](#)

[How NTFS Works](#)

Fsutil wim

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows 10

Provides functions to discover and manage Windows Image (WIM)-backed files.

Syntax

```
fsutil wim [enumfiles] <drive name> <data source>
fsutil wim [enumwims] <drive name>
fsutil wim [queryfile] <filename>
fsutil wim [removewim] <drive name> <data source>
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------|---|
| enumfiles | Enumerates WIM backed files. |
| <drive name> | Specifies the drive name. |
| <data source> | Specifies the data source. |
| enumwims | Enumerates backing WIM files. |
| queryfile | Queries if the file is backed by WIM, and if so, displays details about the WIM file. |
| <filename> | Specifies the filename. |
| removewim | Removes a WIM from backing files. |

Examples

To enumerate the files for drive C: from data source 0, type:

```
fsutil wim enumfiles C: 0
```

To enumerate backing WIM files for drive C:, type:

```
fsutil wim enumwims C:
```

To see if a file is backed by WIM, type:

```
fsutil wim C:\Windows\notepad.exe
```

To remove the WIM from backing files for volume C: and data source 2, type:

```
fsutil wim removewims C: 2
```

Additional references

[Command-Line Syntax Key](#)

[Fsutil](#)

ftype

4/13/2018 • 2 minutes to read • [Edit Online](#)

Displays or modifies file types that are used in file name extension associations. If used without an assignment operator (=), **ftype** displays the current open command string for the specified file type. If used without parameters, **ftype** displays the file types that have open command strings defined.

For examples of how to use this command, see [Examples](#).

Syntax

```
ftype [<FileType>[=[<OpenCommandString>]]]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------------|---|
| <FileType> | Specifies the file type to display or change. |
| <OpenCommandString> | Specifies the open command string to use when opening files of the specified file type. |
| /? | Displays help at the command prompt. |

Remarks

The following table describes how **ftype** substitutes variables within an open command string:

| VARIABLE | REPLACEMENT VALUE |
|-------------|---|
| %0 or %1 | Gets substituted with the file name being launched through the association. |
| %* | Gets all of the parameters. |
| %2, %3, ... | Gets the first parameter (%2), the second parameter (%3), and so on. |
| %~<N> | Gets all of the remaining parameters starting with the <i>N</i> th parameter, where <i>N</i> can be any number from 2 to 9. |

Examples

To display the current file types that have open command strings defined, type:

```
ftype
```

To display the current open command string for the *txtfile* file type, type:

```
ftype txtfile
```

This command produces output similar to the following:

```
txtfile=%SystemRoot%\system32\notepad.exe %1
```

To delete the open command string for a file type called *Example*, type:

```
ftype example=
```

To associate the .pl file name extension with the PerlScript file type and enable the PerlScript file type to run PERL.EXE, type the following commands:

```
assoc .pl=PerlScript  
ftype PerlScript=perl.exe %1 %*
```

To eliminate the need to type the .pl file name extension when invoking a Perl script, type:

```
set PATHEXT=.pl;%PATHEXT%
```

Additional references

[Command-Line Syntax Key](#)

fveupdate

4/13/2018 • 2 minutes to read • [Edit Online](#)

FveUpdate is an internally used tool that is run by setup when a computer is upgraded from Windows Vista or Windows Server 2008 to Windows 7 or Windows Server 2008 R2. It updates the metadata associated with BitLocker to the latest version. This tool cannot be run independently.

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Returns the media access control (MAC) address and list of network protocols associated with each address for all network cards in each computer, either locally or across a network.

Syntax

```
getmac[.exe][/s <computer> [/u <Domain\User> [/p <Password>]]][fo {TABLE | list | CSV}][nh][v]
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------------------|---|
| /s | Specifies the name or IP address of a remote computer (do not use backslashes). The default is the local computer. |
| /u \ | Runs the command with the account permissions of the user specified by User or Domain\User. The default is the permissions of the current logged on user on the computer issuing the command. |
| /p | Specifies the password of the user account that is specified in the /u parameter. |
| /fo { TABLE list CSV} | Specifies the format to use for the query output. Valid values are TABLE , list , and CSV . The default format for output is TABLE . |
| /nh | Suppresses column header in output. Valid when the /fo parameter is set to TABLE or CSV . |
| /v | Specifies that the output display verbose information. |
| /? | |

Remarks

getmac can be useful either when you want to enter the MAC address into a network analyzer or when you need to know what protocols are currently in use on each network adapter in a computer.

Examples

The following examples show how you can use the **getmac** command:

```
getmac /fo table /nh /v
```

```
getmac /s srvmain
```

```
getmac /s srvmain /u maindom\hiropln
```

```
getmac /s srvmain /u maindom\hiropln /p p@ssw23
```

```
getmac /s srvmain /u maindom\hiropln /p p@ssw23 /fo list /v
```

```
getmac /s srvmain /u maindom\hiropln /p p@ssw23 /fo table /nh
```

additional references

- [Command-Line Syntax Key](#)

gettype

4/13/2018 • 2 minutes to read • [Edit Online](#)

Gettype is deprecated, and is not guaranteed to be supported in future releases of Windows.

This tool is included in Windows Server 2003 . For more information see [Gettype](#).

goto

4/13/2018 • 2 minutes to read • [Edit Online](#)

Directs cmd.exe to a labeled line in a batch program. Within a batch program, **goto** directs command processing to a line that is identified by a label. When the label is found, processing continues starting with the commands that begin on the next line.

For examples of how to use this command, see [Examples](#).

Syntax

```
goto <Label>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| <Label> | Specifies a text string that is used as a label in the batch program. |
| /? | Displays help at the command prompt. |

Remarks

- Working with command extensions

If command extensions are enabled (the default), and you use the **goto** command with a target label of **:EOF**, you transfer control to the end of the current batch script file and exit the batch script file without defining a label. When you use **goto** with the **:EOF** label, you must insert a colon before the label. For example:

```
goto:EOF
```

- Using valid *Label* values

You can use spaces in the *Label* parameter, but you cannot include other separators (for example, semicolons or equal signs).

- Matching *Label* with the label in the batch program

The *Label* value that you specify must match a label in the batch program. The label within the batch program must begin with a colon (:). If a line begins with a colon, it is treated as a label and any commands on that line are ignored. If your batch program does not contain the label that you specify in *Label*, the batch program stops and displays the following message:

```
Label not found
```

- Using **goto** for conditional operations

You can use **goto** with other commands to perform conditional operations. For more information about using **goto** for conditional operations, see the [If](#) command reference.

Examples

The following batch program formats a disk in drive A as a system disk. If the operation is successful, the **goto** command directs processing to the **:end** label:

```
echo off
format a: /s
if not errorlevel 1 goto end
echo An error occurred during formatting.
:end
echo End of batch program.
```

Additional references

[Command-Line Syntax Key](#)

[Cmd](#)

[If](#)

Fix domain name dependencies in Group Policy Objects and Group Policy links after a domain rename operation. For examples of how this command can be used, see [Examples](#).

Syntax

```
Gpfixup [/v]
[/olddns:<OLDDNSNAME> /newdns:<NEWDNSNAME>]
[/oldnb:<OLDFLATNAME> /newnb:<NEWFLATNAME>]
[/dc:<DCNAME>] [/sionly]
[/user:<USERNAME> [/pwd:{<PASSWORD>|*}]] [/?]
```

Parameters

| PARAMETER | DESCRIPTION |
|----------------------|---|
| /v | Displays detailed status messages. If this parameter is not used, only error messages or a summary status message of SUCCESS or FAILURE appears. |
| /olddns:<OLDDNSNAME> | Specifies the old DNS name of the renamed domain as <OLDDNSNAME> when the domain rename operation changes the DNS name of a domain. You can use this parameter only if you also use the /newdns parameter to specify a new domain DNS name. |
| /newdns:<NEWDNSNAME> | Specifies the new DNS name of the renamed domain as <NEWDNSNAME> when the domain rename operation changes the DNS name of a domain. You can use this parameter only if you also use the /olddns parameter to specify the old domain DNS name. |
| /oldnb:<OLDFLATNAME> | Specifies the old NetBIOS name of the renamed domain as <OLDFLATNAME> when the domain rename operation changes the NetBIOS name of a domain. You can use this parameter only if you use the /newnb parameter to specify a new domain NetBIOS name. |
| /newnb:<NEWFLATNAME> | Specifies the new NetBIOS name of the renamed domain as <NEWFLATNAME> when the domain rename operation changes the NetBIOS name of a domain. You can use this parameter only if you use the /oldnb parameter to specify the old domain NetBIOS name. |

| PARAMETER | DESCRIPTION |
|------------------|--|
| /dc:<DCNAME> | Connect to the domain controller named <DCNAME> (a DNS name or a NetBIOS name). <DCNAME> must host a writable replica of the domain directory partition as indicated by one of the following: <ul style="list-style-type: none"> - The DNS name <NEWDNSNAME> by using /newdns - The NetBIOS name <NEWFLATNAME> by using /newnb If this parameter is not used, connect to any domain controller in the renamed domain indicated by <NEWDNSNAME> or <NEWFLATNAME>. |
| /sionly | Performs only the Group Policy fix that relates to managed software installation (the Software Installation extension for Group Policy). Skip the actions that fix Group Policy links and the SYSVOL paths in GPOs. |
| /user:<USERNAME> | Runs this command in the security context of the user <USERNAME>, where <USERNAME> is in the format domain\user. If this parameter is not used, runs this command as the logged in user. |
| /pwd:{<PASSWORD> | *} |
| /? | Displays Help at the command prompt. |

Remarks

- The **gpfixup** command is available in Windows Server 2008 R2 and Windows Server 2008, except on Server Core installations.
- Although the Group Policy Management Console (GPMC) is distributed with Windows Server 2008 R2 and Windows Server 2008, you must install Group Policy Management as a feature through Server Manager.

Examples

This example assumes that you have already performed a domain rename operation in which you changed the DNS name from **MyOldDnsName** to **MyNewDnsName**, and the NetBIOS name from **MyOldNetBIOSName** to **MyNewNetBIOSName**. In this example, you use the **gpfixup** command to connect to the domain controller named **MyDcDnsName** and repair GPOs and Group Policy links by updating the old domain name embedded in the GPOs and links. Status and error output is saved to a file that is named **gpfixup.log**.

```
gpfixup /olddns: MyOldDnsName /newdns:MyNewDnsName /oldnb:MyOldNetBIOSName /newnb:MyNewNetBIOSName /dc:MyDcDnsName 2>&1 >gpfixup.log
```

This example is the same as the previous one, except that it assumes the NetBIOS name of the domain was not changed during the domain rename operation.

```
gpfixup /olddns: MyOldDnsName /newdns:MyNewDnsName /dc:MyDcDnsName 2>&1 >gpfixup.log
```

Additional references

- [Administering Active Directory Domain Rename](#)
- [Group Policy TechCenter](#)
- [Command-Line Syntax Key](#)

gpresult

8/21/2018 • 3 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Displays the Resultant Set of Policy (RSOP) information for a remote user and computer. To use RSOP reporting for remotely targeted computers through the firewall, you must have firewall rules that enable inbound network traffic on the ports.

Syntax

```
gpresult [/s <system> [/u <USERNAME> [/p [<PASSWOrd>]]] [/user [<TARGETDOMAIN>\]<TARGETUSER>] [/scope {user | computer}] [/r | /v | /z | [/x | /h] <FILENAME> [/f] | /?]
```

Parameters

NOTE

Except when you use **/?**, you must include an output option, either **/r**, **/v**, **/z**, **/x**, or **/h**.

| PARAMETER | DESCRIPTION |
|-------------------------------------|---|
| /s <system> | Specifies the name or IP address of a remote computer. Do not use backslashes. The default is the local computer. |
| /u <USERNAME> | Uses the credentials of the specified user to run the command. The default user is the user who is logged on to the computer that issues the command. |
| /p [<PASSWOrd>] | Specifies the password of the user account that is provided in the /u parameter. If /p is omitted, gpresult prompts for the password. /p cannot be used with /x or /h . |
| /user [<TARGETDOMAIN>\]<TARGETUSER> | Specifies the remote user whose RSOP data is to be displayed. |
| /scope {user computer} | Displays RSOP data for either the user or the computer. If /scope is omitted, gpresult displays RSOP data for both the user and the computer. |
| /x /h | Saves the report in either XML (/x) or HTML (/h) format at the location and with the file name that is specified by the <i>FILENAME</i> parameter. Cannot be used with /u , /p , /r , /v , or /z . |
| /f | forces gpresult to overwrite the file name that is specified in the /x or /h option. |
| /r | Displays RSOP summary data. |

| PARAMETER | DESCRIPTION |
|-----------|---|
| /v | Displays verbose policy information. This includes detailed settings that were applied with a precedence of 1. |
| /z | Displays all available information about Group Policy. This includes detailed settings that were applied with a precedence of 1 and higher. |
| /? | Displays help at the command prompt. |

Remarks

- Group Policy is the primary administrative tool for defining and controlling how programs, network resources, and the operating system operate for users and computers in an organization. In an active directory environment, Group Policy is applied to users or computers based on their membership in sites, domains, or organizational units.
- Because you can apply overlapping policy settings to any computer or user, the Group Policy feature generates a resulting set of policy settings when the user logs on. **gpresult** displays the resulting set of policy settings that were enforced on the computer for the specified user when the user logged on.
- Because **/v** and **/z** produce lots of information, it is useful to redirect output to a text file (for example, **gpresult/z >policy.txt**).
- The **gpresult** command is available in Windows Server 2012 , Windows Server 2008 R2, Windows Server2008, Windows 8, Windows 7, and Windows Vista. ## Examples The following example retrieves RSoP data for the remote user **targetusername** of the computer **srvmain**, and displays RSoP data about the user only. The command is run with the credentials of the user **maindom\hiropln**, and **p@ssW23** is entered as the password for that user. `gpresult /s srvmain /u maindom\hiropln /p p@ssW23 /user targetusername /scope user /r` The following example saves all available information about Group Policy for the remote user **targetusername** of the computer **srvmain** to a file that is named **policy.txt**. No data is included about the computer. The command is run with the credentials of the user **maindom\hiropln**, and **p@ssW23** is entered as the password for that user. `gpresult /s srvmain /u maindom\hiropln /p p@ssW23 /user targetusername /z > policy.txt` The following example displays RSoP data for the computer **srvmain** and the logged-on user. Data is included about both the user and the computer. The command is run with the credentials of the user **maindom\hiropln**, and **p@ssW23** is entered as the password for that user. `gpresult /s srvmain /u maindom\hiropln /p p@ssW23 /r`
additional references
- [Group Policy TechCenter](#)
- [Command-Line Syntax Key](#)

gpupdate

4/13/2018 • 2 minutes to read • [Edit Online](#)

Updates Group Policy settings. For examples of how this command can be used, see [Examples](#).

Syntax

```
gpupdate [/target:{Computer | User}] [/force] [/wait:<VALUE>] [/logoff] [/boot] [/sync] [/?]
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------------|--|
| /target:{Computer | User} |
| /force | Reapplies all policy settings. By default, only policy settings that have changed are applied. |
| /wait:<VALUE> | <p>Sets the number of seconds to wait for policy processing to finish before returning to the command prompt. When the time limit is exceeded, the command prompt appears, but policy processing continues. The default value is 600 seconds. The value 0 means not to wait. The value -1 means to wait indefinitely.</p> <p>In a script, by using this command with a time limit specified, you can run gpupdate and continue with commands that do not depend upon the completion of gpupdate. Alternatively, you can use this command with no time limit specified to let gpupdate finish running before other commands that depend on it are run.</p> |
| /logoff | Causes a logoff after the Group Policy settings are updated. This is required for those Group Policy client-side extensions that do not process policy on a background update cycle but do process policy when a user logs on. Examples include user-targeted Software Installation and Folder Redirection. This option has no effect if there are no extensions called that require a logoff. |
| /boot | Causes a computer restart after the Group Policy settings are applied. This is required for those Group Policy client-side extensions that do not process policy on a background update cycle but do process policy at computer startup. Examples include computer-targeted Software Installation. This option has no effect if there are no extensions called that require a restart. |
| /sync | Causes the next foreground policy application to be done synchronously. Foreground policy is applied at computer boot and user logon. You can specify this for the user, computer, or both, by using the /target parameter. The /force and /wait parameters are ignored if you specify them. |

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------------|
| /? | Displays Help at the command prompt. |

Remarks

- The **gpupdate** command is available in Windows Server 2008 R2, Windows Server 2008, Windows 7 Ultimate, Windows 7 Professional, Windows Vista Ultimate, Windows Vista Enterprise, and Windows Vista Business.

Examples

Force a background update of all Group Policy settings, regardless of whether they have changed.

```
gpupdate /force
```

Additional references

- [Group Policy TechCenter](#)
- [Command-Line Syntax Key](#)

graftabl

4/13/2018 • 2 minutes to read • [Edit Online](#)

Enables Windows operating systems to display an extended character set in graphics mode. If used without parameters, **graftabl** displays the previous and the current code page.

For examples of how to use this command, see [Examples](#).

Syntax

```
graftabl <CodePage>  
graftabl /status
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|---|
| <CodePage> | Specifies a code page to define the appearance of extended characters in graphics mode. Valid code page identification numbers are: 437: United States 850: Multilingual (Latin I) 852: Slavic (Latin II) 855: Cyrillic (Russian) 857: Turkish 860: Portuguese 861: Icelandic 863: Canadian-French 865: Nordic 866: Russian 869: Modern Greek |
| /status | Displays the current code page that graftabl is using. |
| /? | Displays help at the command prompt. |

Remarks

- **Graftabl** affects only the monitor display of extended characters of the code page that you specify. It does not change the actual console input code page. To change the console input code page, use the **mode** or **chcp** command.
- The following table lists each exit code and a brief description of it.
|Exit code|Description| |-----|-----| |0|Character set was loaded successfully. No previous code page was loaded.| |1|An incorrect parameter was specified. No action was taken.| |2|A file error occurred.|
- You can use the ERRORLEVEL environment variable in a batch program to process exit codes that are returned by **graftabl**.

Examples

To view the current code page used by **graftabl**, type:

```
graftabl /status
```

To load the graphics character set for code page 437 (United States) into memory, type:

```
graftabl 437
```

To load the graphics character set for code page 850 (multilingual) into memory, type:

```
graftabl 850
```

Additional references

[Command-Line Syntax Key](#)

[Freedisk](#)

[Chcp](#)

help

4/13/2018 • 2 minutes to read • [Edit Online](#)

Provides online information about system commands (that is, non-network commands). If used without parameters, **help** lists and briefly describes every system command.

For examples of how to use this command, see [Examples](#).

Syntax

```
help [<Command>]
[<Command>] /?
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| <Command> | Specifies the name of the command that you want information about. |

Examples

To view information about the **robocopy** command, type either of the following:

```
help robocopy
robocopy /?
```

Additional references

[Command-Line Syntax Key](#)

helpctr

4/13/2018 • 2 minutes to read • [Edit Online](#)

Helpctr is deprecated, and is not guaranteed to be supported in future releases of Windows.

This tool is included in Windows Server 2003 . For more information see [Helpctr](#).

hostname

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Displays the host name portion of the full computer name of the computer.

Syntax

```
hostname
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------------|
| /? | Displays help at the command prompt. |

Remarks

This command is available only if the Internet Protocol (TCP/IP) protocol is installed as a component in the properties of a network adapter in Network.

Examples

To display the name of the computer, type:

```
hostname
```

additional references

- [Command-Line Syntax Key](#)

icacs

8/21/2018 • 3 minutes to read • [Edit Online](#)

Displays or modifies discretionary access control lists (DACLS) on specified files, and applies stored DACLS to files in specified directories.

For examples of how to use this command, see [Examples](#).

Syntax

```
icacs <FileName> [/grant[:r] <Sid>:<Perm>[...]] [/deny <Sid>:<Perm>[...]] [/remove[:g|:d]] <Sid>[...]] [/t]
[/c] [/l] [/q] [/setintegritylevel <Level>:<Policy>[...]]
icacs <Directory> [/substitute <SidOld> <SidNew> [...]] [/restore <ACLfile> [/c] [/l] [/q]]
```

Parameters

| PARAMETER | DESCRIPTION |
|--|---|
| <FileName> | Specifies the file for which to display DACLS. |
| <Directory> | Specifies the directory for which to display DACLS. |
| /t | Performs the operation on all specified files in the current directory and its subdirectories. |
| /c | Continues the operation despite any file errors. Error messages will still be displayed. |
| /l | Performs the operation on a symbolic link versus its destination. |
| /q | Suppresses success messages. |
| [/save <ACLfile> [/t] [/c] [/l] [/q]] | Stores DACLS for all matching files into <i>ACLfile</i> for later use with /restore . |
| [/setowner <Username> [/t] [/c] [/l] [/q]] | Changes the owner of all matching files to the specified user. |
| [/findSID <Sid> [/t] [/c] [/l] [/q]] | Finds all matching files that contain a DACL explicitly mentioning the specified security identifier (SID). |
| [/verify [/t] [/c] [/l] [/q]] | Finds all files with ACLs that are not canonical or have lengths inconsistent with ACE (access control entry) counts. |
| [/reset [/t] [/c] [/l] [/q]] | Replaces ACLs with default inherited ACLs for all matching files. |

| PARAMETER | DESCRIPTION |
|---|--|
| <code>[/grant[:r] <Sid>:[...]]</code> | Grants specified user access rights. Permissions replace previously granted explicit permissions. Without :r , permissions are added to any previously granted explicit permissions. |
| <code>[/deny <Sid>:[...]]</code> | Explicitly denies specified user access rights. An explicit deny ACE is added for the stated permissions and the same permissions in any explicit grant are removed. |
| <code>[/remove[:g]:d]] <Sid> [...] [/t] [/c] [/l] [/q]</code> | Removes all occurrences of the specified SID from the DACL. :g removes all occurrences of granted rights to the specified SID. :d removes all occurrences of denied rights to the specified SID. |
| <code>[/setintegritylevel [(CI)(OI)]<Level>:[...]]</code> | Explicitly adds an integrity ACE to all matching files. <i>Level</i> is specified as: - L [ow] - M [edium] - H [igh] Inheritance options for the integrity ACE may precede the level and are applied only to directories. |
| <code>[/substitute <SidOld> [...]]</code> | Replaces an existing SID (<i>SidOld</i>) with a new SID (<i>SidNew</i>). Requires the <i>Directory</i> parameter. |
| <code>/restore <ACLfile> [/c] [/l] [/q]</code> | Applies stored DACLs from <i>ACLfile</i> to files in the specified directory. Requires the <i>Directory</i> parameter. |
| <code>/inheritancelevel:[e]d[r]</code> | Sets the inheritance level: e - Enables inheritance d - Disables inheritance and copies the ACEs r - Removes all inherited ACEs |

Remarks

- SIDs may be in either numerical or friendly name form. If you use a numerical form, affix the wildcard character ***** to the beginning of the SID.
- icacls** preserves the canonical order of ACE entries as:
 - Explicit denials
 - Explicit grants
 - Inherited denials
 - Inherited grants
- Perm* is a permission mask that can be specified in one of the following forms:
 - A sequence of simple rights:
 - F** (full access)
 - M** (modify access)
 - R**X (read and execute access)
 - R** (read-only access)
 - W** (write-only access)

- A comma-separated list in parenthesis of specific rights:

D (delete)

RC (read control)

WDAC (write DAC)

WO (write owner)

S (synchronize)

AS (access system security)

MA (maximum allowed)

GR (generic read)

GW (generic write)

GE (generic execute)

GA (generic all)

RD (read data/list directory)

WD (write data/add file)

AD (append data/add subdirectory)

REA (read extended attributes)

WEA (write extended attributes)

X (execute/traverse)

DC (delete child)

RA (read attributes)

WA (write attributes)

- Inheritance rights may precede either *Perm* form, and they are applied only to directories:

(OI): object inherit

(CI): container inherit

(IO): inherit only

(NP): do not propagate inherit

Examples

To save the DACLs for all files in the C:\Windows directory and its subdirectories to the ACLFile file, type:

```
icaccls c:\windows\* /save aclfile /t
```

To restore the DACLs for every file within ACLFile that exists in the C:\Windows directory and its subdirectories, type:

```
icacIs c:\windows\ /restore aclfile
```

To grant the user User1 Delete and Write DAC permissions to a file named "Test1", type:

```
icacIs test1 /grant User1:(d,wdac)
```

To grant the user defined by SID S-1-1-0 Delete and Write DAC permissions to a file, named "Test2", type:

```
icacIs test2 /grant *S-1-1-0:(d,wdac)
```

Additional references

[Command-Line Syntax Key](#)

if

4/13/2018 • 3 minutes to read • [Edit Online](#)

Performs conditional processing in batch programs.

For examples of how to use this command, see [Examples](#).

Syntax

```
if [not] ERRORLEVEL <Number> <Command> [else <Expression>]
if [not] <String1>==<String2> <Command> [else <Expression>]
if [not] exist <FileName> <Command> [else <Expression>]
```

If command extensions are enabled, use the following syntax:

```
if [/i] <String1> <CompareOp> <String2> <Command> [else <Expression>]
if cmdextversion <Number> <Command> [else <Expression>]
if defined <Variable> <Command> [else <Expression>]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------------|---|
| not | Specifies that the command should be carried out only if the condition is false. |
| errorlevel <Number> | Specifies a true condition only if the previous program run by Cmd.exe returned an exit code equal to or greater than <i>Number</i> . |
| <Command> | Specifies the command that should be carried out if the preceding condition is met. |
| <String1>== | Specifies a true condition only if <i>String1</i> and <i>String2</i> are the same. These values can be literal strings or batch variables (for example, %1). You do not need to enclose literal strings in quotation marks. |
| exist <FileName> | Specifies a true condition if the specified file name exists. |
| <CompareOp> | Specifies a three-letter comparison operator. The following list represents valid values for <i>CompareOp</i> : EQU Equal to NEQ Not equal to LSS Less than LEQ Less than or equal to GTR Greater than GEQ Greater than or equal to |

| PARAMETER | DESCRIPTION |
|------------------------|---|
| /i | Forces string comparisons to ignore case. You can use /i on the <i>String1==String2</i> form of if . These comparisons are generic, in that if both <i>String1</i> and <i>String2</i> are comprised of numeric digits only, the strings are converted to numbers and a numeric comparison is performed. |
| cmdextversion <Number> | Specifies a true condition only if the internal version number associated with the command extensions feature of Cmd.exe is equal to or greater than the number specified. The first version is 1. It increases by increments of one when significant enhancements are added to the command extensions. The cmdextversion conditional is never true when command extensions are disabled (by default, command extensions are enabled). |
| defined <Variable> | Specifies a true condition if <i>Variable</i> is defined. |
| <Expression> | Specifies a command-line command and any parameters to be passed to the command in an else clause. |
| /? | Displays help at the command prompt. |

Remarks

- If the condition specified in an **if** clause is true, the command that follows the condition is carried out. If the condition is false, the command in the **if** clause is ignored and the command executes any command that is specified in the **else** clause.
- When a program stops, it returns an exit code. To use exit codes as conditions, use **errorlevel**.
- If you use **defined**, the following three variables are added to the environment: **%errorlevel%**, **%cmdcmdline%**, and **%cmdextversion%**.
 - **%errorlevel%** expands into a string representation of the current value of the ERRORLEVEL environment variable. This assumes that there is not an existing environment variable with the name ERRORLEVEL—if there is, you will get that ERRORLEVEL value instead.
 - **%cmdcmdline%** expands into the original command line that was passed to Cmd.exe prior to any processing by Cmd.exe. This assumes that there is not an existing environment variable with the name CMDCMDLINE—if there is, you will get the CMDCMDLINE value instead.
 - **%cmdextversion%** expands into the string representation of the current value of **cmdextversion**. This assumes that there is not an existing environment variable with the name CMDEXTVERSION—if there is, you will get the CMDEXTVERSION value instead.
- You must use the **else** clause on the same line as the command after the **if**.

Examples

To display the message "Cannot find data file" if the file Product.dat cannot be found, type:

```
if not exist product.dat echo Cannot find data file
```

To format a disk in drive A and display an error message if an error occurs during the formatting process, type the following lines in a batch file:

```
:begin
@echo off
format a: /s
if not errorlevel 1 goto end
echo An error occurred during formatting.
:end
echo End of batch program.
```

To delete the file Product.dat from the current directory or display a message if Product.dat is not found, type the following lines in a batch file:

```
IF EXIST Product.dat (
del Product.dat
) ELSE (
echo The Product.dat file is missing.
)
```

NOTE

These lines can be combined into a single line as follows:

```
IF EXIST Product.dat (del Product.dat) ELSE (echo The Product.dat file is missing.)
```

To echo the value of the ERRORLEVEL environment variable after running a batch file, type the following lines in the batch file:

```
goto answer%errorlevel%
:answer1
echo Program had return code 1
:answer0
echo Program had return code 0
goto end
:end
echo Done!
```

To go to the "okay" label if the value of the ERRORLEVEL environment variable is less than or equal to 1, type:

```
if %errorlevel% LEQ 1 goto okay
```

Additional references

[Command-Line Syntax Key](#)

[If](#)

[Goto](#)

inuse

4/13/2018 • 2 minutes to read • [Edit Online](#)

Inuse is deprecated, and is not guaranteed to be supported in future releases of Windows.

This tool is included in Windows Server 2003 . For more information see [Inuse](#).

ipconfig

4/13/2018 • 4 minutes to read • [Edit Online](#)

Displays all current TCP/IP network configuration values and refreshes Dynamic Host Configuration Protocol (DHCP) and Domain Name System (DNS) settings. Used without parameters, **ipconfig** displays Internet Protocol version 4 (IPv4) and IPv6 addresses, subnet mask, and default gateway for all adapters.

Syntax

```
ipconfig [/allcompartments] [/all] [/renew [<Adapter>]] [/release [<Adapter>]] [/renew6 [<Adapter>]] [/release6 [<Adapter>]] [/flushdns] [/displaydns] [/registerdns] [/showclassid <Adapter>] [/setclassid <Adapter> [<ClassID>]]
```

Parameters

| PARAMETER | DESCRIPTION |
|----------------------|---|
| /all | Displays the full TCP/IP configuration for all adapters. Adapters can represent physical interfaces, such as installed network adapters, or logical interfaces, such as dial-up connections. |
| /allcompartments | Displays the full TCP/IP configuration for all compartments. |
| /displaydns | Displays the contents of the DNS client resolver cache, which includes both entries preloaded from the local Hosts file and any recently obtained resource records for name queries resolved by the computer. The DNS Client service uses this information to resolve frequently queried names quickly, before querying its configured DNS servers. |
| /flushdns | Flushes and resets the contents of the DNS client resolver cache. During DNS troubleshooting, you can use this procedure to discard negative cache entries from the cache, as well as any other entries that have been added dynamically. |
| /registerdns | Initiates manual dynamic registration for the DNS names and IP addresses that are configured at a computer. You can use this parameter to troubleshoot a failed DNS name registration or resolve a dynamic update problem between a client and the DNS server without rebooting the client computer. The DNS settings in the advanced properties of the TCP/IP protocol determine which names are registered in DNS. |
| /release [<Adapter>] | Sends a DHCPRELEASE message to the DHCP server to release the current DHCP configuration and discard the IP address configuration for either all adapters (if an adapter is not specified) or for a specific adapter if the <i>Adapter</i> parameter is included. This parameter disables TCP/IP for adapters configured to obtain an IP address automatically. To specify an adapter name, type the adapter name that appears when you use ipconfig without parameters. |

| PARAMETER | DESCRIPTION |
|---------------------------|---|
| /release6 [<Adapter>] | Sends a DHCPRELEASE message to the DHCPv6 server to release the current DHCP configuration and discard the IPv6 address configuration for either all adapters (if an adapter is not specified) or for a specific adapter if the <i>Adapter</i> parameter is included. This parameter disables TCP/IP for adapters configured to obtain an IP address automatically. To specify an adapter name, type the adapter name that appears when you use ipconfig without parameters. |
| /renew [<Adapter>] | Renews DHCP configuration for all adapters (if an adapter is not specified) or for a specific adapter if the <i>Adapter</i> parameter is included. This parameter is available only on computers with adapters that are configured to obtain an IP address automatically. To specify an adapter name, type the adapter name that appears when you use ipconfig without parameters. |
| /renew6 [<Adapter>] | Renews DHCPv6 configuration for all adapters (if an adapter is not specified) or for a specific adapter if the <i>Adapter</i> parameter is included. This parameter is available only on computers with adapters that are configured to obtain an IPv6 address automatically. To specify an adapter name, type the adapter name that appears when you use ipconfig without parameters. |
| /setclassid <Adapter> [] | Configures the DHCP class ID for a specified adapter. To set the DHCP class ID for all adapters, use the asterisk (*) wildcard character in place of <i>Adapter</i> . This parameter is available only on computers with adapters that are configured to obtain an IP address automatically. If a DHCP class ID is not specified, the current class ID is removed. |
| /showclassid <Adapter> | Displays the DHCP class ID for a specified adapter. To see the DHCP class ID for all adapters, use the asterisk (*) wildcard character in place of <i>Adapter</i> . This parameter is available only on computers with adapters that are configured to obtain an IP address automatically. |
| /? | Displays Help at the command prompt. |

Remarks

- This command is most useful on computers that are configured to obtain an IP address automatically. This enables users to determine which TCP/IP configuration values have been configured by DHCP, Automatic Private IP Addressing (APIPA), or an alternate configuration.
- If the name you supply for *Adapter* contains any spaces, use quotation marks around the adapter name (example: "*Adapter Name*").
- For adapter names, **ipconfig** supports the use of the asterisk (*) wildcard character to specify either adapters with names that begin with a specified string or adapters with names that contain a specified string. For example, **Local*** matches all adapters that start with the string Local and ***Con*** matches all adapters that contain the string Con.

Examples

To display the basic TCP/IP configuration for all adapters, type:

```
ipconfig
```

To display the full TCP/IP configuration for all adapters, type:

```
ipconfig /all
```

To renew a DHCP-assigned IP address configuration for only the Local Area Connection adapter, type:

```
ipconfig /renew "Local Area Connection"
```

To flush the DNS resolver cache when troubleshooting DNS name resolution problems, type:

```
ipconfig /flushdns
```

To display the DHCP class ID for all adapters with names that start with Local, type:

```
ipconfig /showclassid Local*
```

To set the DHCP class ID for the Local Area Connection adapter to TEST, type:

```
ipconfig /setclassid "Local Area Connection" TEST
```

Additional references

- [Command-Line Syntax Key](#)

ipxroute

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Displays and modifies information about the routing tables used by the IPX protocol. Used without parameters, **ipxroute** displays the default settings for packets that are sent to unknown, broadcast, and multicast addresses.

Syntax

```
ipxroute servers [/type=X]
ipxroute ripout <Network>
ipxroute resolve {guid | name} {GUID | <AdapterName>}
ipxroute board= N [def] [gbr] [mbr] [remove=xxxxxxxxxxx]
ipxroute config
```

Parameters

| PARAMETER | DESCRIPTION |
|---|---|
| servers[/type=X] | Displays the Service Access Point (SAP) table for the specified server type. X must be an integer. For example, /type=4 displays all file servers. If you do not specify /type , ipxroute servers displays all types of servers, listing them by server name. |
| ripout Network | Discovers if <i>Network</i> is reachable by consulting the IPX stack's route table and sending out a rip request if necessary. <i>Network</i> is the IPX network segment number. |
| resolve{ GUID name} { GUID AdapterName} | Resolves the name of the GUID to its friendly name, or the friendly name to its GUID. |
| board= N | Specifies the network adapter for which to query or set parameters. |
| def | Sends packets to the ALL ROUTES broadcast. If a packet is transmitted to a unique Media Access Card (MAC) address that is not in the source routing table, ipxroute sends the packet to the SINGLE ROUTES broadcast by default. |
| gbr | Sends packets to the ALL ROUTES broadcast. If a packet is transmitted to the broadcast address (FFFFFFFFFFFF), ipxroute sends the packet to the SINGLE ROUTES broadcast by default. |
| mbr | Sends packets to the ALL ROUTES broadcast. If a packet is transmitted to a multicast address (C000xxxxxxx), ipxroute sends the packet to the SINGLE ROUTES broadcast by default. |

| PARAMETER | DESCRIPTION |
|----------------------|---|
| remove= xxxxxxxxxxxx | removes the given node address from the source routing table. |
| config | Displays information about all of the bindings for which IPX is configured. |
| /? | Displays help at the command prompt. |

Examples

To display the network segments that the workstation is attached to, the workstation node address, and frame type being used, type:

```
ipxroute config
```

additional references

- [Command-Line Syntax Key](#)

irftp

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Sends files over an infrared link.

Syntax

```
irftp [<Drive>:\] [[<path>] <FileName>] [/h]/s]
```

Parameters

| PARAMETER | DESCRIPTION |
|----------------|---|
| Drive: | Specifies the drive that contains the files that you want to send over an infrared link. |
| [path]FileName | Specifies the location and name of the file or set of files that you want to send over an infrared link. If you specify a set of files, you must specify the full path for each file. |
| /h | Specifies hidden mode. When hidden mode is used, the files are sent without displaying the Wireless Link dialog box. |
| /s | Opens the Wireless Link dialog box, so that you can select the file or set of files that you want to send without using the command line to specify the drive, path, and file names. |

Remarks

- Before using this command, verify that the devices that you want to communicate over an infrared link have infrared functionality enabled and working correctly, and that an infrared link is established between the devices.
- Used without parameters or used with **/s**, **irftp** opens the **Wireless Link** dialog box, where you can select the files that you want to send without using the command line.

Examples

Send Example.txt over the infrared link.

```
irftp c:\example.txt
```

additional references

- [Command-Line Syntax Key](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

compacts a Windows Internet Name Service (WINS) or Dynamic Host Configuration Protocol (DHCP) database. Microsoft recommends that you compact the WINS database whenever it approaches 30 MB.

Syntax

```
jetpack.EXE <database name> <temp database name>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| | Specifies the original database file. |
| | Specifies the temporary database file. |
| /? | Displays help at the command prompt. |

Examples

To compact the WINS database:

```
cd %SYSTEMROOT%\SYSTEM32\WINS
NET STOP WINS
jetpack WINS.MDB TMP.MDB
NET start WINS
```

To compact the DHCP database:

```
cd %SYSTEMROOT%\SYSTEM32\DHCP
NET STOP DHCPSErver
jetpack DHCP.MDB TMP.MDB
NET start DHCPSErver
```

In the examples above, **Tmp.mdb** is a temporary database that is used by jetpack.exe. **Wins.mdb** is the WINS database. **Dhcp.mdb** is the DHCP database. jetpack.exe compacts the WINS or DHCP database by doing the following:

1. Copies database information to a temporary database file called **Tmp.mdb**.
2. deletes the original database file, **Wins.mdb** or **Dhcp.mdb**.
3. renames the temporary database files to the original filename.

NOTE

During the compact process, jetpack.exe creates a temporary file with the name that is specified by the *temp database name* parameter. The temporary file is removed when the compact process is complete. Make sure you do not have a file already existing in WINS or DHCP folder with the same name as the one specified in the *temp database name* parameter.

additional references

- [Command-Line Syntax Key](#)

klist

4/13/2018 • 5 minutes to read • [Edit Online](#)

Displays a list of currently cached Kerberos tickets. This information applies to Windows Server 2012. For examples of how this command can be used, see [Examples](#).

Syntax

```
klist [-lh <LogonId.HighPart>] [-li <LogonId.LowPart>] tickets | tgt | purge | sessions | kcd_cache | get |  
add_bind | query_bind | purge_bind
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|---|
| -lh | Denotes the high part of the user's locally unique identifier (LUID), expressed in hexadecimal. If neither -lh or -li are present, the command defaults to the LUID of the user who is currently signed in. |
| -li | Denotes the low part of the user's locally unique identifier (LUID), expressed in hexadecimal. If neither -lh or -li are present, the command defaults to the LUID of the user who is currently signed in. |
| tickets | Lists the currently cached ticket-granting-tickets (TGTs), and service tickets of the specified logon session. This is the default option. |
| tgt | Displays the initial Kerberos TGT. |
| purge | Allows you to delete all the tickets of the specified logon session. |
| sessions | Displays a list of logon sessions on this computer. |
| kcd_cache | Displays the Kerberos constrained delegation cache information. |
| get | Allows you to request a ticket to the target computer specified by the service principal name (SPN). |
| add_bind | Allows you to specify a preferred domain controller for Kerberos authentication. |
| query_bind | Displays a list of cached preferred domain controllers for each domain that Kerberos has contacted. |
| purge_bind | Removes the cached preferred domain controllers for the domains specified. |

| PARAMETER | DESCRIPTION |
|------------|---|
| kdcoptions | Displays the Key Distribution Center (KDC) options specified in RFC 4120. |
| /? | Displays Help for this command. |

Remarks

Membership in **Domain Admins**, or equivalent, is the minimum required to run all the parameters of this command.

If no parameters are provided, Klist will retrieve all the tickets for the currently logged on user.

The parameters display the following information:

- **tickets**

Lists the currently cached tickets of services that you have authenticated to since logon. Displays the following attributes of all cached tickets:

- LogonID: The LUID
- Client: The concatenation of the client name and the domain name of the client
- Server: The concatenation of the service name and the domain name of the service
- KerbTicket Encryption Type: The encryption type that is used to encrypt the Kerberos ticket
- Ticket Flags: The Kerberos ticket flags
- Start Time: The time from which the ticket will be valid
- End Time: The time the ticket becomes no longer valid. When a ticket is past this time, it can no longer be used to authenticate to a service or be used for renewal
- Renew Time: The time that a new initial authentication is required
- Session Key Type: The encryption algorithm that is used for the session key

- **tgt**

Lists the initial Kerberos TGT and the following attributes of the currently cached ticket:

- LogonID: Identified in hexadecimal
- ServiceName: krbtgt
- TargetName <SPN>: krbtgt
- DomainName: Name of the domain that issues the TGT
- TargetDomainName: Domain that the TGT is issued to
- AltTargetDomainName: Domain that the TGT is issued to
- Ticket Flags: Address and target actions and type
- Session Key: Key length and encryption algorithm
- StartTime: Local computer time that the ticket was requested
- EndTime: Time the ticket becomes no longer valid. When a ticket is past this time, it can no longer be used to authenticate to a service.
- RenewUntil: Deadline for ticket renewal
- TimeSkew: Time difference with the Key Distribution Center (KDC)
- EncodedTicket: Encoded ticket

- **purge**

Allows you to delete a specific ticket. Purging tickets destroys all tickets that you have cached, so use this

attribute with caution. It might stop you from being able to authenticate to resources. If this happens, you will have to log off and log on again.

- LogonID: Identified in hexadecimal

- **sessions**

Allows you to list and display the information for all logon sessions on this computer.

- LogonID: If specified, displays the logon session only by the given value. If not specified, displays all the logon sessions on this computer.

- **kcd_cache**

Allows you to display the Kerberos constrained delegation cache information.

- LogonID: If specified, displays the cache information for the logon session by the given value. If not specified, displays the cache information for the current user's logon session.

- **get**

Allows you to request a ticket to the target that is specified by the SPN.

- LogonID: If specified, requests a ticket by using the logon session by the given value. If not specified, requests a ticket by using the current user's logon session.
- kdcoptions: Requests a ticket with the given KDC options

- **add_bind**

Allows you to specify a preferred domain controller for Kerberos authentication.

- **query_bind**

Allows you to display cached, preferred domain controllers for the domains.

- **purge_bind**

Allows you to remove cached, preferred domain controllers for the domains.

- **kdcoptions**

For the current list of options and their explanations, see [RFC 4120](#).

Other considerations

- Klist.exe is available in Windows Server 2012 and Windows 8, and it requires no special installation.

Examples

1. When you are diagnosing an Event ID 27 while processing a ticket-granting service (TGS) request for the target server, the account did not have a suitable key to generate a Kerberos ticket. You can use Klist to query the Kerberos ticket cache to determine if any tickets are missing, if the target server or account is in error, or if the encryption type is not supported.

```
klist
```

```
klist -li 0x3e7
```

2. When you diagnose errors and you want to know the specifics of each ticket-granting-ticket that is cached on the computer for a logon session, you can use Klist to display the TGT information.

```
klist tgt
```

3. If you are unable to establish a connection and diagnosis might take too long, you can purge the Kerberos ticket cache, log off, and then log back on.

```
klist purge
```

```
klist purge -li 0x3e7
```

4. When you want to diagnose a logon session for a user or a service, you can use the following command to find the LogonID that is used in other Klist commands.

```
klist sessions
```

5. When you want to diagnose Kerberos constrained delegation failure, you can use the following command to find the last error that was encountered.

```
klist kcd_cache
```

6. When you want to diagnose if a user or a service can get a ticket to a server, you can use this command to request a ticket for a specific SPN.

```
klist get host/%computername%
```

7. When diagnosing replication issues across domain controllers, you typically need the client computer to target a specific domain controller. In these cases, you can use the following command to target the client computer to that specific domain controller.

```
klist add_bind CONTOSO KDC.CONTOSO.COM
```

```
klist add_bind CONTOSO.COM KDC.CONTOSO.COM
```

8. To query what domain controllers this computer recently contacted, you can use the following command.

```
klist query_bind
```

9. When you want Kerberos to rediscover domain controllers, you can use the following command. This command can also be used to flush the cache before creating new domain controller bindings with klist add_bind.

```
klist purge_bind
```

Additional references

- [Command-Line Syntax Key](#)

ksetup

4/13/2018 • 3 minutes to read • [Edit Online](#)

Performs tasks that are related to setting up and maintaining Kerberos protocol and the Key Distribution Center (KDC) to support Kerberos realms, which are not also Windows domains. For examples of how this command can be used, see the Examples section in each of the related subtopics.

Syntax

```
ksetup
[/setrealm <DNSDomainName>]
[/mapuser <Principal> <Account>]
[/addkdc <RealmName> <KDCName>]
[/delkdc <RealmName> <KDCName>]
[/addkpasswd <RealmName> <KDCPasswordName>]
[/delkpasswd <RealmName> <KDCPasswordName>]
[/server <ServerName>]
[/setcomputerpassword <Password>]
[/removerealms <RealmName>]
[/domain <DomainName>]
[/changepassword <OldPassword> <NewPassword>]
[/listrealmflags]
[/setrealmflags <RealmName> [sendaddress] [tcpsupported] [delegate] [ncsupported] [rc4]]
[/addrealmflags <RealmName> [sendaddress] [tcpsupported] [delegate] [ncsupported] [rc4]]
[/delrealmflags [sendaddress] [tcpsupported] [delegate] [ncsupported] [rc4]]
[/dumpstate]
[/addhosttorealmmap] <HostName> <RealmName>]
[/delhosttorealmmap] <HostName> <RealmName>]
[/setenttypeattr] <DomainName> {DES-CBC-CRC | DES-CBC-MD5 | RC4-HMAC-MD5 | AES128-CTS-HMAC-SHA1-96 | AES256-CTS-HMAC-SHA1-96}
[/getenttypeattr] <DomainName>
[/addenttypeattr] <DomainName> {DES-CBC-CRC | DES-CBC-MD5 | RC4-HMAC-MD5 | AES128-CTS-HMAC-SHA1-96 | AES256-CTS-HMAC-SHA1-96}
[/delenttypeattr] <DomainName>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------------------------|---|
| Ksetup:setrealm | Makes this computer a member of a Kerberos realm. |
| Ksetup:mapuser | Maps a Kerberos principal to an account. |
| Ksetup:addkdc | Defines a KDC entry for the given realm. |
| Ksetup:delkdc | Deletes a KDC entry for the realm. |
| Ksetup:addkpasswd | Adds a Kpasswd server address for a realm. |
| Ksetup:delkpasswd | Deletes a Kpasswd server address for a realm. |
| Ksetup:server | Allows you to specify the name of a Windows computer on which to apply the changes. |

| PARAMETER | DESCRIPTION |
|--|--|
| Ksetup:setcomputerpassword | Sets the password for the computer's domain account (or host principal). |
| Ksetup:removerealm | Deletes all information for the specified realm from the registry. |
| Ksetup:domain | Allows you to specify a domain (if <DomainName> has not been set by using /domain). |
| Ksetup:changepassword | Allows you to use the Kpasswd to change the logged on user's password. |
| Ksetup:listrealmflags | Lists the available realm flags that ksetup can detect. |
| Ksetup:setrealmflags | Sets realm flags for a specific realm. |
| Ksetup:addrealmflags | Adds additional realm flags to a realm. |
| Ksetup:delrealmflags | Deletes realm flags from a realm. |
| Ksetup:dumpstate | Analyzes the Kerberos configuration on the given computer. Adds a host to realm mapping to the registry. |
| Ksetup:addhosttorealmmap | Adds a registry value to map the host to the Kerberos realm. |
| Ksetup:delhosttorealmmap | Deletes the registry value that mapped the host computer to the Kerberos realm. |
| Ksetup:setenctypeattr | Sets one or more encryption types trust attributes for the domain. |
| Ksetup:getenctypeattr | Gets the encryption types trust attribute for the domain. |
| Ksetup:addenctypeattr | Adds encryption types to the encryption types trust attribute for the domain. |
| Ksetup:delenctypeattr | Deletes the encryption types trust attribute for the domain. |
| /? | Displays Help at the command prompt. |

Remarks

Ksetup is used to change the computer settings for locating Kerberos realms. In non-Microsoft Kerberos-based implementations, this information is usually kept in the Krb5.conf file. In Windows Server operating systems, it is kept in the registry. You can use this tool to modify these settings. These settings are used by workstations to locate Kerberos realms and by domain controllers to locate Kerberos realms for cross-realm trust relationships.

Ksetup initializes registry keys that the Kerberos Security Support Provider (SSP) uses to locate a KDC for the Kerberos realm if the computer is running Windows Server 2003, Windows Server 2008, or Windows Server 2008 R2 and is not a member of a Windows domain. After configuration, the user of a client computer that is running the Windows operating system can log on to accounts in the Kerberos realm.

The Kerberos version 5 protocol is the default for network authentication on computers running Windows XP Professional, Windows Vista, and Windows 7. The Kerberos SSP searches the registry for the domain name of the user's realm and then resolves the name to an IP address by querying a DNS server. The Kerberos protocol can use DNS to locate KDCs by using only the realm name, but it must be specially configured to do so.

Additional references

- [Command-Line Syntax Key](#)

ksetup:setrealm

4/13/2018 • 2 minutes to read • [Edit Online](#)

Sets the name of a Kerberos realm. For examples of how this command can be used, see [Examples](#).

Syntax

```
ksetup /setrealm <DNSDomainName>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------|--|
| <DNSDomainName> | The DNS domain name can be in the form of a fully qualified domain name or simple domain name. |

Remarks

The DNS domain name parameter should be entered in uppercase letters. Otherwise, the **ksetup** command will ask for verification to continue.

Setting the Kerberos realm on a domain controller is not supported. Attempting to do so will cause a warning and a command failure.

Examples

Set the realm for this computer to a specific domain name to restrict access by a non-domain controller just to the CONTOSO Kerberos realm:

```
ksetup /setrealm CONTOSO
```

Additional references

- [Command-Line Syntax Key](#)
- [Ksetup](#)
- [Ksetup:removerealm](#)

ksetup:mapuser

4/13/2018 • 2 minutes to read • [Edit Online](#)

Maps the name of a Kerberos principal to an account. For examples of how this command can be used, see [Examples](#).

Syntax

```
ksetup /mapuser <Principal> <Account>
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------|---|
| <Principal> | The fully qualified domain name of any principal; for example, mike@corp.CONTOSO.COM. |
| <Account> | Any account or security group name that exists on this computer, such as Guest, Domain Users, or Administrator. |

Remarks

An account can be specifically identified, such as domain guests. Or you can use the wildcard character (*) to include all accounts.

If an account name is omitted, mapping is deleted for the specified principal.

The computer will only authenticate the principals of the given realm if they present valid Kerberos tickets.

Use **ksetup** without any parameters or arguments to see the current mapped settings and the default realm.

Whenever changes are made to the external Key Distribution Center (KDC) and the realm configuration, a restart of the computer where the setting was changed is required.

Examples

Map Mike Danseglio's account within the Kerberos realm CONTOSO to the guest account on this computer, granting him all the privileges of a member of the built-in Guest account without having to authenticate to this computer:

```
ksetup /mapuser mike@corp.CONTOSO.COM guest
```

Remove the mapping of Mike Danseglio's account to the guest account on this computer to prevent him from authenticating to this computer with his credentials from CONTOSO:

```
ksetup /mapuser mike@corp.CONTOSO.COM
```

Map Mike Danseglio's account within the CONTOSO Kerberos realm to any existing account on this computer. (if only the standard user and guest accounts are active on this computer, Mike's privileges will be set to those):

```
ksetup /mapuser mike@corp.CONTOSO.COM *
```

Map all accounts within the CONTOSO Kerberos realm to any existing account of the same name on this computer:

```
ksetup /mapuser * *
```

Additional references

- [Command-Line Syntax Key](#)
- [Ksetup](#)

ksetup:addkdc

4/13/2018 • 2 minutes to read • [Edit Online](#)

Adds a Key Distribution Center (KDC) address for the given Kerberos realm. For examples of how this command can be used, see [Examples](#).

Syntax

```
ksetup /addkdc <RealmName> [<KDCName>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------|---|
| <RealmName> | The realm name is stated as an uppercase DNS name, such as CORP.CONTOSO.COM, and it is listed as the default realm when ksetup is run. It is to this realm that you are attempting to add the other KDC. |
| <KDCName> | The KDC name is stated as a case insensitive fully qualified domain name, such as mitkdc.microsoft.com. If the KDC name is omitted, DNS will locate KDCs. |

Remarks

These mappings are stored in the registry under

HKEY_LOCAL_MACHINE\System\CurrentControlSet\Control\LSA\Kerberos\Domains. To deploy Kerberos realm configuration data to multiple computers, use the Security Configuration Template snap-in and policy distribution instead of using **ksetup** explicitly on individual computers.

The computer must be restarted before the new realm setting will be used.

To verify the default realm name for the computer, or to verify that this command worked as intended, run **ksetup** at the command prompt and verify the output for the added KDC.

Examples

Configure a non-Windows KDC server and the realm that the workstation should use:

```
ksetup /addkdc CORP.CONTOSO.COM mitkdc.contoso.com
```

Run the Ksetup tool at the command line of the same computer as in the preceding command to set the local computer account password to "p@sswrđ1%?". Then restart the computer.

```
Ksetup /setcomputerpassword p@sswrđ1%
```

Additional references

- [Ksetup](#)
- [Ksetup:setcomputerpassword](#)

- [Command-Line Syntax Key](#)

ksetup:delkdc

4/13/2018 • 2 minutes to read • [Edit Online](#)

Deletes instances of Key Distribution Center (KDC) names for the Kerberos realm. For examples of how this command can be used, see [Examples](#).

Syntax

```
ksetup /delkdc <RealmName> <KDCName>
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------|--|
| <RealmName> | The realm name is stated as an uppercase DNS name, such as CORP.CONTOSO.COM, and it is listed as the default realm when ksetup is run. It is to this realm from which you are attempting to delete the other KDC. |
| <KDCName> | The KDC name is stated as a case-insensitive, fully qualified domain name, such as mitkdc.contoso.com. |

Remarks

These mappings are stored in the registry in

HKEY_LOCAL_MACHINE\System\CurrentControlSet\Control\LSA\Kerberos\Domains. To remove realm configuration data from multiple computers, use the Security Configuration Template snap-in and policy distribution instead of using **ksetup** explicitly on individual computers.

On computers running Windows 2000 Server with Service Pack 1 (SP1) and earlier, the computer must be restarted before the changed realm setting configuration will be used.

To verify the default realm name for the computer, or to verify that this command worked as intended, run **ksetup** at the command prompt and verify that the KDC that was removed does not exist in the list.

Examples

The security requirements for this computer have changed so the link between the Windows realm and the non-Windows realm must be removed. First, determine which association to remove and produce the output of existing associations:

```
ksetup
```

Remove the association by using the following command:

```
Ksetup /delkdc CORP.CONTOSO.COM mitkdc.contoso.com
```

Additional references

- [Ksetup](#)

- [Command-Line Syntax Key](#)

ksetup:addkpasswd

4/13/2018 • 2 minutes to read • [Edit Online](#)

Adds a Kerberos password (Kpasswd) server address for a realm. For examples of how this command can be used, see [Examples](#).

Syntax

```
ksetup /addkpasswd <RealmName> [<KpasswdName>]
```

Parameters

If the Kerberos realm that the workstation will be authenticating to supports the Kerberos change password protocol, you can configure a client computer running the Windows operating system to use a Kerberos password server. This setting is configured on the realm side.

| PARAMETER | DESCRIPTION |
|---------------|--|
| <RealmName> | The realm name is stated as an uppercase DNS name, such as CORP.CONTOSO.COM, and is listed as the default realm or Realm= when ksetup is run. |
| <KpasswdName> | The KDC name that is to be used as the Kerberos password server is stated as a case insensitive fully qualified domain name, such as mitkdc.microsoft.com. If the KDC name is omitted, DNS might be used to locate KDCs. |

Remarks

If the Kerberos realm that the workstation will be authenticating to supports the Kerberos change password protocol, you can configure a client computer running the Windows operating system to use a Kerberos password server.

Run the command **ksetup** to verify the KDC name. If **kpasswd =** does not appear in the output, the mapping has not been configured.

You can add additional KDC names one at a time.

Examples

Configure the realm, CORP.CONTOSO.COM, so that it uses the non-Windows KDC server, mitkdc.contoso.com, as the password server:

```
ksetup /addkpasswd CORP.CONTOSO.COM mitkdc.contoso.com
```

This results in a non-Windows Kerberos password server that controls all the passwords for authentication between it and the realm.

Additional references

- [Ksetup](#)
- [Ksetup:delkpasswd](#)

- [Command-Line Syntax Key](#)

ksetup:delkpasswd

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

removes a Kerberos password server (Kpasswd) for a realm. For examples of how this command can be used, see [Examples](#).

Syntax

```
ksetup /delkpasswd <RealmName> <KpasswdName>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| | The realm name is stated as an uppercase DNS name, such as CORP.CONTOSO.COM, and is listed as the default realm or REALM= when ksetup is run. |
| | The KDC name to be used as the Kerberos password server is stated as a case-insensitive, fully qualified domain name, such as mitkdc.contoso.com. If the KDC name is omitted, DNS might be used to locate KDCs. |

Remarks

Run the command **ksetup** to verify the KDC name. If **kpasswd =** does not appear in the output, then the mapping has not been configured. Multiple mappings will be listed, if set.

Examples

verify the realm CORP.CONTOSO.COM uses the non-Windows KDC server mitkdc.contoso.com as the password server:

```
ksetup /delkpasswd CORP.CONTOSO.COM mitkdc.contoso.com
```

To verify the command worked as intended, run **ksetup** on the Windows computer to verify the realm CORP.CONTOSO.COM is not mapped to a Kerberos password server (the KDC name).

additional references

- [ksetup](#)
- [ksetup:delkpasswd](#)
- [Command-Line Syntax Key](#)

ksetup:server

4/13/2018 • 2 minutes to read • [Edit Online](#)

Allows you to specify a name for a computer running the Windows operating system so that the changes made by using **ksetup** will update the target computer. For examples of how this command can be used, see [Examples](#).

Syntax

```
ksetup /server <ServerName>
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------|---|
| <ServerName> | The full computer name on which the configuration will be effective, such as IPops897.corp.contoso.com. If an incomplete fully qualified domain computer name is specified, the command will fail. |

Remarks

There is no way to remove the targeted server name; you can only change it back to the local computer name, which is the default.

The target server name is stored in the registry in

HKEY_LOCAL_MACHINE\SYSTEM\ControlSet001\Control\LSA\Kerberos. It is not reported by using **ksetup**.

Examples

Make your **ksetup** configurations effective on the IPops897 computer that is connected on the Contoso domain:

```
ksetup /server IPops897.corp.contoso.com
```

Additional references

- [Ksetup](#)
- [Command-Line Syntax Key](#)

ksetup:setcomputerpassword

4/13/2018 • 2 minutes to read • [Edit Online](#)

Sets the password for the local computer. For examples of how this command can be used, see [Examples](#).

Syntax

```
ksetup /setcomputerpassword <Password>
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|---|
| <Password> | Uses the supplied password to set the computer account on the local computer. The password can only be set by using an account with administrative privileges. The password can be from 1 to 156 alphanumeric or special characters. |

Remarks

This command affects the computer account only.

You must restart the computer for the password change to take effect.

The computer account password is not displayed in the registry or as output from the **ksetup** command.

Examples

Change the computer account password on the local computer from IPops897 to IPop\$897!.

```
ksetup /setcomputerpassword IPop$897!
```

Additional references

- [Ksetup](#)
- [Command-Line Syntax Key](#)

ksetup:removerealm

4/13/2018 • 2 minutes to read • [Edit Online](#)

Deletes all information for the specified realm from the registry. For examples of how this command can be used, see [Examples](#).

Syntax

```
ksetup /removerealm <RealmName>
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------|---|
| <RealmName> | The realm name is stated as an uppercase DNS name, such as CORP.CONTOSO.COM, and it is listed as the default realm when ksetup is run. |

Remarks

The realm name is stored in two places in the registry: **HKEY_LOCAL_MACHINE\SYSTEM\ControlSet001** and **\CurrentControlSet\Control\Lsa\Kerberos**.

You cannot remove the default realm name from the domain controller because this will reset its DNS information, and removing it might make the domain controller unusable.

Examples

Mistakenly set the realm name by misspelling ".COM" on the local computer to CORP.CONTOSO.CON

```
ksetup /setrealm CORP.CONTOSO.CON
```

Remove that erroneous realm name from the local computer:

```
ksetup /removerealm CORP.CONTOSO.CON
```

Verify the removal by running **ksetup** and review the output.

Additional references

- [Ksetup](#)
- [Ksetup:setrealm](#)
- [Command-Line Syntax Key](#)

ksetup:domain

4/13/2018 • 2 minutes to read • [Edit Online](#)

Sets the domain name for all Kerberos operations. For examples of how this command can be used, see [Examples](#).

Syntax

```
ksetup /domain <DomainName>
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------|---|
| <DomainName> | Name of the domain to which you want to establish a connection. Use the fully qualified domain name or a simple form of the name, such as contoso.com or contoso. |

Remarks

None.

Examples

Establish a connection to a valid domain, such as Microsoft by using the /mapuser subcommand:

```
ksetup /mapuser principal@realm domain-user /domain domain-name
```

When the connection is successful, you will receive a new TGT or an existing TGT will be refreshed.

Additional references

- [Ksetup](#)
- [Command-Line Syntax Key](#)

ksetup:changepassword

4/13/2018 • 2 minutes to read • [Edit Online](#)

Uses the Key Distribution Center (KDC) password (kpasswd) value to change the password of the logged-on user. For examples of how this command can be used, see [Examples](#).

Syntax

```
ksetup /changepassword <OldPasswd> <NewPasswd>
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------|--|
| <OldPasswd> | States the logged-on user's existing password. |
| <NewPasswd> | States the logged on user's new password. |

Remarks

This command uses the KDC password (kpasswd) value to change the password of the logged-on user. The kpasswd, if set, is displayed in the output by running the **ksetup /dumpstate** command.

The user's new password must meet all the password requirements that are set on this computer.

If the user account is not found in the current domain, the system will ask you to supply the domain name where the user account resides.

If you want to force a password change at next logon, this command allows the use of the asterisk (*) so the user will be prompted for a new password.

The output of the command informs you of the success or failure status.

Examples

Change the password of a user who is currently logged on to this computer in this domain:

```
ksetup /changepassword Pa$$w0rd Pa$$w0rd
```

Change the password of a user who is currently logged on in the Contoso domain:

```
ksetup /domain CONTOSO /changepassword Pa$$w0rd Pa$$w0rd
```

Force the currently logged on user to change the password at the next logon:

```
ksetup /changepassword Pa$$w0rd *
```

Additional references

- [Command-Line Syntax Key](#)

ksetup:listrealmflags

4/13/2018 • 2 minutes to read • [Edit Online](#)

Lists the available realm flags that can be reported by **ksetup**. For examples of how this command can be used, see [Examples](#).

Syntax

```
ksetup /listrealmflags
```

Parameters

None

Remarks

The realm flags specify additional features of a non-Windows-based Kerberos realm. Computers that are running Windows Server 2003, Windows Server 2008, or Windows Server 2008 R2 can use a non-Windows-based Kerberos server to administer authentication instead of using a domain that is running a Windows Server operating system. These systems participate in a Kerberos realm instead of a Windows domain. This entry establishes the features of the realm. The following table describes each.

| VALUE | REALM FLAG | DESCRIPTION |
|-------|--------------|---|
| 0xF | All | All realm flags are set. |
| 0x00 | None | No realm flags are set, and no additional features are enabled. |
| 0x01 | SendAddress | The IP address will be included within the ticket-granting tickets. |
| 0x02 | TcpSupported | The Transmission Control Protocol (TCP) and the User Datagram Protocol (UDP) are supported in this realm. |
| 0x04 | Delegate | Everyone in this realm is trusted for delegation. |
| 0x08 | NcSupported | This realm supports name canonicalization, which allows for DNS and realm naming standards. |
| 0x80 | RC4 | This realm supports RC4 encryption to enable cross-realm trust, which allows for the use of TLS. |

Realm flags are stored in the registry in

HKEY_LOCAL_MACHINE\SYSTEM\CurrentControlSet\Control\Lsa\Kerberos\Domains*Realm-name*.

This entry does not exist in the registry by default. You can use the [ksetup:addrealmflags](#) command to populate the registry.

Examples

List the known realm flags on this computer:

```
ksetup /listrealmflags
```

Set the available realm flags that **Ksetup** does not know by typing either of the following commands at the command line:

```
ksetup /setrealmflags CORP.CONTOSO.COM sendaddress tcpsupported delete ncsupported
```

```
ksetup /setrealmflags CORP.CONTOSO.COM 0xF
```

Additional references

- [Ksetup:setrealmflags](#)
- [Ksetup:addrealmflags](#)
- [Ksetup:delrealmflags](#)
- [Ksetup](#)
- [Command-Line Syntax Key](#)

ksetup:setrealmflags

4/13/2018 • 2 minutes to read • [Edit Online](#)

Sets realm flags for the specified realm. For examples of how this command can be used, see [Examples](#).

Syntax

```
ksetup /setrealmflags <RealmName> [sendaddress] [tcpsupported] [delegate] [ncsupported] [rc4]
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------|---|
| <RealmName> | The realm name is stated as an uppercase DNS name, such as CORP.CONTOSO.COM. |
| Realm flag | Denotes one of the following flags: <ul style="list-style-type: none">- SendAddress- TcpSupported- Delegate- NcSupported- RC4 |

Remarks

The realm flags specify additional features of a Kerberos realm that is not based on the Windows Server operating system. Computers that are running Windows Server 2003, Windows Server 2008, or Windows Server 2008 R2 can use a Kerberos server to administer authentication instead of using a domain that is running a Windows Server operating system, and these systems participate in a Kerberos realm. This entry establishes the features of the realm. The following table describes each.

| VALUE | REALM FLAG | DESCRIPTION |
|-------|--------------|--|
| 0xF | All | All realm flags are set. |
| 0x00 | None | No realm flags are set, and no additional features are enabled. |
| 0x01 | SendAddress | The IP address will be included within the ticket-granting tickets. |
| 0x02 | TcpSupported | Both the Transmission Control Protocol (TCP) and the User Datagram Protocol (UDP) are supported in this realm. |
| 0x04 | Delegate | Everyone in this realm is trusted for delegation. |

| VALUE | REALM FLAG | DESCRIPTION |
|-------|-------------|--|
| 0x08 | NcSupported | This realm supports name canonicalization, which allows for DNS and Realm naming standards. |
| 0x80 | RC4 | This realm supports RC4 encryption to enable cross-realm trust, which allows for the use of TLS. |

Realm flags are stored in the registry under

HKEY_LOCAL_MACHINE\SYSTEM\CurrentControlSet\Control\Lsa\Kerberos\Domains\RealmName.

This entry does not exist in the registry by default. You can use the [Ksetup:addrealmflags](#) command to populate the registry.

You can see what realm flags are available and set by viewing the output of **ksetup**.

Examples

List the available and set realm flags for the realm CONTOSO:

```
ksetup
```

Set two flags that are not currently set:

```
ksetup /setrealmflags CONTOSO ncsupported delegate
```

Run the **ksetup** command to verify that the realm flag is set by viewing the output and looking for **Realm flags =**.

Additional references

- [Ksetup:listrealmflags](#)
- [Ksetup:addrealmflags](#)
- [Ksetup:delrealmflags](#)
- [Command-Line Syntax Key](#)

ksetup:addrealmflags

4/13/2018 • 2 minutes to read • [Edit Online](#)

Adds additional realm flags to the specified realm. For examples of how this command can be used, see [Examples](#).

Syntax

```
ksetup /addrealmflags <RealmName> [sendaddress] [tcpsupported] [delegate] [ncsupported] [rc4]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|--|
| Realm name | The realm name is stated as an uppercase DNS name, such as CORP.CONTOSO.COM. |

Remarks

The realm flags specify additional features of a Kerberos realm that is not based on the Windows Server operating system. Computers that are running Windows Server 2003, Windows Server 2008, or Windows Server 2008 R2 can use a Kerberos server to administer authentication instead of using a domain that is running a Windows Server operating system, and these systems participate in a Kerberos realm. This entry establishes the features of the realm. The following table describes each.

| VALUE | REALM FLAG | DESCRIPTION |
|-------|--------------|--|
| 0xF | All | All realm flags are set. |
| 0x00 | None | No realm flags are set, and no additional features are enabled. |
| 0x01 | SendAddress | The IP address will be included within the ticket-granting tickets. |
| 0x02 | TcpSupported | Both the Transmission Control Protocol (TCP) and the User Datagram Protocol (UDP) are supported in this realm. |
| 0x04 | Delegate | Everyone in this realm is trusted for delegation. |
| 0x08 | NcSupported | This realm supports name canonicalization, which allows for DNS and Realm naming standards. |
| 0x80 | RC4 | This realm supports RC4 encryption to enable cross-realm trust, which allows for the use of TLS. |

Realm flags are stored in the Registry under

HKEY_LOCAL_MACHINE\SYSTEM\CurrentControlSet\Control\Lsa\Kerberos\Domains\Realm-name.

This entry does not exist in the registry by default. You can use the [Ksetup:addrealmflags](#) command to populate the Registry.

You can see what realm flags are available and set by viewing the output of ksetup or ksetup /dumpstate.

Examples

List the available realm flags for the realm CONTOSO:

```
Ksetup /listrealmflags
```

Set two flags to the CONTOSO realm:

```
ksetup /setrealmflags CONTOSO ncsupported delegate
```

Add one more flag that is not currently in the set:

```
ksetup /addrealmflags CONTOSO SendAddress
```

Run the **ksetup** command to verify that the realm flag is set by viewing the output and looking for **Realm flags** =.

Additional references

- [Ksetup:listrealmflags](#)
- [Ksetup:setrealmflags](#)
- [Ksetup:delrealmflags](#)
- [Command-Line Syntax Key](#)

ksetup:delrealmflags

4/13/2018 • 2 minutes to read • [Edit Online](#)

Removes realm flags from the specified realm. For examples of how this command can be used, see [Examples](#).

Syntax

```
ksetup /delrealmflags <RealmName> [sendaddress] [tcpsupported] [delegate] [ncsupported] [rc4]
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------|--|
| <RealmName> | The realm name is stated as an uppercase DNS name, such as CORP.CONTOSO.COM, and is listed as the default realm when Ksetup is run. |

Remarks

The realm flags specify additional features of a Kerberos realm that is not based on the Windows Server operating system. Computers that are running Windows Server 2003, Windows Server 2008, or Windows Server 2008 R2 can use a Kerberos server to administer authentication instead of using a domain that is running a Windows Server operating system, and these systems participate in a Kerberos realm. This entry establishes the features of the realm. The following table describes each.

| VALUE | REALM FLAG | DESCRIPTION |
|-------|--------------|--|
| 0xF | All | All realm flags are set. |
| 0x00 | None | No realm flags are set, and no additional features are enabled. |
| 0x01 | SendAddress | The IP address will be included within the ticket-granting-tickets. |
| 0x02 | TcpSupported | Both the Transmission Control Protocol (TCP) and the User Datagram Protocol (UDP) are supported in this realm. |
| 0x04 | Delegate | Everyone in this realm is trusted for delegation. |
| 0x08 | NcSupported | This realm supports name canonicalization, which allows for DNS and Realm naming standards. |
| 0x80 | RC4 | This realm supports RC4 encryption to enable cross-realm trust, which allows for the use of TLS. |

Realm flags are stored in the registry in

HKEY_LOCAL_MACHINE\SYSTEM\CurrentControlSet\Control\Lsa\Kerberos\Domains\RealmName.

This entry does not exist in the registry by default. You can use the [Ksetup:addrealmflags](#) command to populate the Registry.

You can see what realm flags are available and set by viewing the output of **ksetup** or **ksetup /dumpstate**.

Examples

List the available realm flags for the realm CONTOSO:

```
Ksetup /listrealmflags
```

Remove two flags that are currently in the set:

```
ksetup /delrealmflags CONTOSO ncsupported delegate
```

Run the **ksetup** command to verify that the realm flag is set by viewing the output and looking for **Realm flags** =.

Additional references

- [Ksetup:listrealmflags](#)
- [Ksetup:setrealmflags](#)
- [Ksetup:addrealmflags](#)
- [Command-Line Syntax Key](#)

ksetup:dumpstate

4/13/2018 • 2 minutes to read • [Edit Online](#)

Displays the current state of realm settings for all realms that are defined on the computer. For examples of how this command can be used, see [Examples](#).

Syntax

```
ksetup /dumpstate
```

Parameters

None

Remarks

The output of this command includes the default realm (the domain that the computer is a member of) and all the realms that are defined on this computer. The following is included for each realm:

- All the Key Distribution Centers (KDCs) that are associated with this realm
- All the **set realm** flags for this realm
- The KDC password

This command does not display the domain name that is specified by DNS detection or by the command **ksetup /domain**.

This command does not display the computer password that is set by using the command **ksetup /setcomputerpassword**.

Ksetup produces the same output as **ksetup /dumpstate**.

Examples

Find most of the Kerberos realm configurations on a computer:

```
ksetup /dumpstate
```

Additional references

- [Ksetup](#)
- [Command-Line Syntax Key](#)

ksetup:addhosttorealmmmap

4/13/2018 • 2 minutes to read • [Edit Online](#)

Adds a service principal name (SPN) mapping between the stated host and the realm. For examples of how this command can be used, see [Examples](#).

Syntax

```
ksetup /addhosttorealmmmap <HostName> <RealmName>
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------|---|
| <HostName> | The host name is the computer name, and it can be stated as the computer's fully qualified domain name. |
| <RealmName> | The realm name is stated as an uppercase DNS name, such as CORP.CONTOSO.COM. |

Remarks

This command allows you to map a host or multiple hosts that are sharing the same DNS suffix to the realm.

The mapping is recorded in the registry in

HKEY_LOCAL_MACHINE\SYSTEM\CurrentControlSet\Lsa\Kerberos\HostToRealm.

Examples

As part of configuring the realm CONTOSO, map the host computer IPops897 to the realm:

```
ksetup /addhosttorealmmmap IPops897 CONTOSO
```

Verify in the registry that the mapping is as intended.

Additional references

- [Ksetup:delhosttorealmmmap](#)
- [Ksetup](#)
- [Command-Line Syntax Key](#)

ksetup:delhosttorealmmmap

4/13/2018 • 2 minutes to read • [Edit Online](#)

Removes a service principal name (SPN) mapping between the stated host and the realm. For examples of how this command can be used, see [Examples](#).

Syntax

```
ksetup /delhosttorealmmmap <HostName> <RealmName>
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------|---|
| <HostName> | The host name is the computer name, and it can be stated as the computer's fully qualified domain name. |
| <RealmName> | The realm name is stated as an uppercase DNS name, such as CORP.CONTOSO.COM. |

Remarks

When a host to realm (or multiple hosts to realm) mapping exists, this command removes that mapping.

The mapping is recorded in the registry in

HKEY_LOCAL_MACHINE\SYSTEM\CurrentControlSet\Lsa\Kerberos\HostToRealm. You should verify the mapping in the registry after using this command.

Examples

Altering the configuration of the realm CONTOSO, delete the mapping of the host computer IPops897 to the realm:

```
ksetup /delhosttorealmmmap IPops897 CONTOSO
```

After running this command, you can verify in the registry that the mapping is as intended.

Additional references

- [Ksetup:addhosttorealmmmap](#)
- [Ksetup](#)
- [Command-Line Syntax Key](#)

ksetup:setenctypeattr

4/13/2018 • 2 minutes to read • [Edit Online](#)

Sets the encryption type attribute for the domain. For examples of how this command can be used, see [Examples](#).

Syntax

```
ksetup /setenctypeattr <Domain name> {DES-CBC-CRC | DES-CBC-MD5 | RC4-HMAC-MD5 | AES128-CTS-HMAC-SHA1-96 | AES256-CTS-HMAC-SHA1-96}
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------|---|
| <DomainName> | Name of the domain to which you want to establish a connection. Use the fully qualified domain name or a simple form of the name, such as corp.contoso.com or contoso. |
| Encryption type | Must be one of the following supported encryption types: <ul style="list-style-type: none">- DES-CBC-CRC- DES-CBC-MD5- RC4-HMAC-MD5- AES128-CTS-HMAC-SHA1-96- AES256-CTS-HMAC-SHA1-96 |

Remarks

To view the encryption type for the Kerberos ticket-granting ticket (TGT) and the session key, run the **klist** command and view the output.

You can set or add multiple encryption types by separating the encryption types in the command with a space. However, you can only do so for one domain at a time.

If the command succeeds or fails, a status message is displayed.

To set the domain that you want to connect to and use, run the **ksetup /domain <DomainName>** command.

Examples

Determine the current encryption types that are set on this computer:

```
klist
```

Set the domain to corp.contoso.com:

```
ksetup /domain corp.contoso.com
```

Set the encryption type attribute to AES-256-CTS-HMAC-SHA1-96 for the domain corp.contoso.com:

```
ksetup /setenctypeattr corp.contoso.com AES-256-CTS-HMAC-SHA1-96
```

Verify that the encryption type attribute was set as intended for the domain:

```
ksetup /getenctypeattr corp.contoso.com
```

Additional references

- [Klist](#)
- [Ksetup:domain](#)
- [Ksetup:addenctypeattr](#)
- [Ksetup:getenctypeattr](#)
- [Ksetup:delenctypeattr](#)
- [Command-Line Syntax Key](#)

ksetup:getenctypeattr

4/13/2018 • 2 minutes to read • [Edit Online](#)

Retrieves the encryption type attribute for the domain. For examples of how this command can be used, see [Examples](#).

Syntax

```
ksetup /getenctypeattr <DomainName>
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------|--|
| <DomainName> | Name of the domain to which you want to establish a connection. Use the fully qualified domain name or a simple form of the name, such as corp.contoso.com or contoso. |

Remarks

To view the encryption type for the Kerberos ticket-granting ticket (TGT) and the session key, run the **klist** command and view the output.

If the command succeeds or fails, a status message is displayed upon successful or failed completion.

To set the domain that you want to connect to and use, run the **ksetup /domain <DomainName>** command.

Examples

Verify the encryption type attribute for the domain:

```
ksetup /getenctypeattr mit.contoso.com
```

Additional references

- [Klist](#)
- [Ksetup:domain](#)
- [Ksetup:addenctypeattr](#)
- [Ksetup:setenctypeattr](#)
- [Ksetup:delenctypeattr](#)
- [Command-Line Syntax Key](#)

ksetup:addentypeattr

4/13/2018 • 2 minutes to read • [Edit Online](#)

Adds the encryption type attribute to the list of possible types for the domain. For examples of how this command can be used, see [Examples](#).

Syntax

```
ksetup /addentypeattr <DomainName> {DES-CBC-CRC | DES-CBC-MD5 | RC4-HMAC-MD5 | AES128-CTS-HMAC-SHA1-96 | AES256-CTS-HMAC-SHA1-96}
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------|---|
| <DomainName> | Name of the domain to which you want to establish a connection. Use the fully qualified domain name or a simple form of the name, such as corp.contoso.com or contoso. |
| Encryption type | Must be one of the following supported encryption types: <ul style="list-style-type: none">- DES-CBC-CRC- DES-CBC-MD5- RC4-HMAC-MD5- AES128-CTS-HMAC-SHA1-96- AES256-CTS-HMAC-SHA1-96 |

Remarks

To view the encryption type for the Kerberos ticket-granting ticket (TGT) and the session key, run the **klist** command and view the output.

You can set or add multiple encryption types by separating the encryption types in the command with a space. However, you can only do so for one domain at a time.

If the command succeeds or fails, a status message is displayed.

To set the domain that you want to connect to and use, run the **ksetup /domain <DomainName>** command.

Examples

Determine the current encryption types that are set on this computer:

```
klist
```

Set the domain to corp.contoso.com:

```
ksetup /domain corp.contoso.com
```

Add the encryption type AES-256-CTS-HMAC-SHA1-96 to the list of possible types for the domain corp.contoso.com:

```
ksetup /addenctypeattr corp.contoso.com AES-256-CTS-HMAC-SHA1-96
```

Set the encryption type attribute to AES-256-CTS-HMAC-SHA1-96 for the domain corp.contoso.com:

```
ksetup /setenctypeattr corp.contoso.com AES-256-CTS-HMAC-SHA1-96
```

Verify that the encryption type attribute was set as intended for the domain:

```
ksetup /getenctypeattr corp.contoso.com
```

Additional references

- [Klist](#)
- [Ksetup:domain](#)
- [Ksetup:setenctypeattr](#)
- [Ksetup:getenctypeattr](#)
- [Ksetup:delenctypeattr](#)
- [Command-Line Syntax Key](#)

ksetup:delenctypeattr

4/13/2018 • 2 minutes to read • [Edit Online](#)

Removes the encryption type attribute for the domain. For examples of how this command can be used, see [Examples](#).

Syntax

```
ksetup /delenctypeattr <DomainName>
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------|--|
| <DomainName> | Name of the domain to which you want to establish a connection. Use the fully qualified domain name or a simple form of the name, such as corp.contoso.com or contoso. |

Remarks

To view the encryption type for the Kerberos ticket-granting ticket (TGT) and the session key, run the **klist** command and view the output.

A status message is displayed upon successful or failed completion.

To set the domain that you want to connect to and use, run the **ksetup /domain <DomainName>** command.

Examples

Determine the current encryption types that are set on this computer:

```
klist
```

Set the domain to mit.contoso.com:

```
ksetup /domain mit.contoso.com
```

Verify what the encryption type attribute is for the domain:

```
ksetup /getenctypeattr mit.contoso.com
```

Remove the set encryption type attribute for the domain mit.contoso.com:

```
ksetup /delenctypeattr mit.contoso.com
```

Additional references

- [Klist](#)
- [Ksetup:domain](#)

- [Ksetup:addenctypeattr](#)
- [Ksetup:setenctypeattr](#)
- [Ksetup:delenctypeattr](#)
- [Command-Line Syntax Key](#)

ktmutil

4/13/2018 • 2 minutes to read • [Edit Online](#)

Starts the Kernel Transaction Manager utility. If used without parameters, **ktmutil** displays available subcommands.

For examples of how to use this command, see [Examples](#).

Syntax

```
ktmutil list tms
ktmutil list transactions [{TmGuid}]
ktmutil resolve complete {TmGuid} {RmGuid} {EnGuid}
ktmutil resolve commit {TxGuid}
ktmutil resolve rollback {TxGuid}
ktmutil force commit {??Guid}
ktmutil force rollback {??Guid}
ktmutil forget
```

Parameters

Remarks

Examples

To force an Indoubt transaction with GUID 311a9209-03f4-11dc-918f-00188b8f707b to commit, type:

```
ktmutil force commit {311a9209-03f4-11dc-918f-00188b8f707b}
```

Additional references

[Command-Line Syntax Key](#)

ktpass

10/17/2017 • 4 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Configures the server principal name for the host or service in active directory Domain Services (AD DS) and generates a .keytab file that contains the shared secret key of the service. The .keytab file is based on the Massachusetts Institute of Technology (MIT) implementation of the Kerberos authentication protocol. The ktpass command-line tool allows non-Windows services that support Kerberos authentication to use the interoperability features provided by the Kerberos Key Distribution Center (KDC) service. This topic applies to the operating system versions designated in the **Applies To** list at the beginning of the topic.

Syntax

```
ktpass
[/out <FileName>]
[/princ <PrincipalName>]
[/mapuser <UserAccount>]
[/mapop {add|set}] [{-|+}desonly] [/in <FileName>]
[/pass {Password}*|{-|+}rndpass}]
[/minpass]
[/maxpass]
[/crypto {DES-CBC-CRC|DES-CBC-MD5|RC4-HMAC-NT|AES256-SHA1|AES128-SHA1|All}]
[/itercount]
[/ptype {KRB5_NT_PRINCIPAL|KRB5_NT_SRV_INST|KRB5_NT_SRV_HST}]
[/kvno <KeyversionNum>]
[/answer {-|+}]
[/target]
[/rawsalt] [{-|+}dumpsalt] [{-|+}setupn] [{-|+}setpass <Password>] [/?|/h|/help]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| /out | Specifies the name of the Kerberos version 5 .keytab file to generate. Note: This is the .keytab file that you transfer to a computer that is not running the Windows operating system, and then replace or merge with your existing .keytab file, /Etc/Krb5.keytab. |
| /princ | Specifies the principal name in the form host/computer.contoso.com@CONTOSO.COM. Warning: This parameter is case sensitive. See remarks for more information. |
| /mapuser | Maps the name of the Kerberos principal, which is specified by the princ parameter, to the specified domain account. |

| PARAMETER | DESCRIPTION |
|---|--|
| /mapop {add set} | <p>Specifies how the mapping attribute is set.</p> <ul style="list-style-type: none"> - add adds the value of the specified local user name. This is the default. - Set sets the value for Data Encryption Standard (DES)-only encryption for the specified local user name. |
| {- +}desonly | <p>DES-only encryption is set by default.</p> <ul style="list-style-type: none"> - + Sets an account for DES-only encryption. - - Releases restriction on an account for DES-only encryption. <p>IMPORTANT: Beginning with Windows 7 and Windows Server 2008 R2, Windows does not support DES by default.</p> |
| /in | Specifies the .keytab file to read from a host computer that is not running the Windows operating system. |
| /pass {Password * {- +}rndpass} | Specifies a password for the principal user name that is specified by the princ parameter. Use "*" to prompt for a password. |
| /minpass | Sets the minimum length of the random password to 15 characters. |
| /maxpass | Sets the maximum length of the random password to 256 characters. |
| /crypto {DES-CBC-CRC DES-CBC-MD5 RC4-HMAC-NT AES256-SHA1 AES128-SHA1 All} | <p>Specifies the keys that are generated in the keytab file:</p> <ul style="list-style-type: none"> - DES-CBC-CRC is used for compatibility. - DES-CBC-MD5 adheres more closely to the MIT implementation and is used for compatibility. - RC4-HMAC-NT employs 128-bit encryption. - AES256-SHA1 employs AES256-CTS-HMAC-SHA1-96 encryption. - AES128-SHA1 employs AES128-CTS-HMAC-SHA1-96 encryption. - All states that all supported cryptographic types can be used. Note: The default settings are based on older MIT versions. Therefore, <code>/crypto</code> should always be specified. |
| /itercount | Specifies the iteration count that is used for AES encryption. The default is that itercount is ignored for non-AES encryption and set at 4,096 for AES encryption. |
| /ptype {KRB5_NT_PRINCIPAL KRB5_NT_SRV_INST KRB5_NT_SRV_HST} | <p>Specifies the principal type.</p> <ul style="list-style-type: none"> - KRB5_NT_PRINCIPAL is the general principal type (recommended). - KRB5_NT_SRV_INST is the user service instance. - KRB5_NT_SRV_HST is the host service instance. |
| /kvno | Specifies the key version number. The default value is 1. |

| PARAMETER | DESCRIPTION |
|---------------|--|
| /answer {- +} | Sets the background answer mode: - Answers reset password prompts automatically with NO. + Answers reset password prompts automatically with YES. |
| /target | Sets which domain controller to use. The default is for the domain controller to be detected, based on the principal name. If the domain controller name does not resolve, a dialog box will prompt for a valid domain controller. |
| /rawsalt | forces ktpass to use the rawsalt algorithm when generating the key. This parameter is not needed. |
| {- +}dumpsalt | The output of this parameter shows the MIT salt algorithm that is being used to generate the key. |
| {- +}setupn | Sets the user principal name (UPN) in addition to the service principal name (SPN). The default is to set both in the .keytab file. |
| {- +}setpass | Sets the user's password when supplied. If rndpass is used, a random password is generated instead. |
| /?/h/help | Displays command-line help for ktpass. |

remarks

Services running on systems that are not running the Windows operating system can be configured with service instance accounts in active directory Domain Services. This allows any Kerberos client to authenticate to services that are not running the Windows operating system by using Windows KDCs.

The /princ parameter is not evaluated by ktpass and is used as provided. There is no check to see if the parameter matches the exact case of the **userPrincipalName** attribute value when generating the Keytab file. Case sensitive Kerberos distributions using this Keytab file might have problems when there is no exact case match and could fail during pre-authentication. Check and retrieve the correct **userPrincipalName** attribute value from a LDifDE export file. For example:

```
ldifde /f keytab_user.ldf /d "CN=Keytab User,OU=UserAccounts,DC=contoso,DC=corp,DC=microsoft,DC=com" /p base /l samaccountname,userprincipalname
```

Examples

The following example illustrates how to create a Kerberos .keytab file, machine.keytab, in the current directory for the user Sample1. (You will merge this file with the Krb5.keytab file on a host computer that is not running the Windows operating system.) The Kerberos .keytab file will be created for all supported encryption types for the general principal type.

To generate a .keytab file for a host computer that is not running the Windows operating system, use the following steps to map the principal to the account and set the host principal password:

1. Use the active directory User and computers snap-in to create a user account for a service on a computer that is not running the Windows operating system. For example, create an account with the name Sample1.
2. Use ktpass to set up an identity mapping for the user account by typing the following at a command prompt:

```
ktpass /princ host/Sample1.contoso.com@CONTOSO.COM /mapuser Sample1 /pass MyPas$w0rd /out Sample1.keytab  
/crypto all /ptype KRB5_NT_PRINCIPAL /mapop set
```

NOTE

You cannot map multiple service instances to the same user account.

3. Merge the .keytab file with the /Etc/Krb5.keytab file on a host computer that is not running the Windows operating system.

additional references

[Command-Line Syntax Key](#)

label

4/13/2018 • 2 minutes to read • [Edit Online](#)

Creates, changes, or deletes the volume label (that is, the name) of a disk. If used without parameters, the **label** command changes the current volume label or deletes the existing label.

For examples of how to use this command, see [Examples](#).

Syntax

```
label [/mp] [<Volume>] [<Label>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| /mp | Specifies that the volume should be treated as a mount point or volume name. |
| <Volume> | Specifies a drive letter (followed by a colon), mount point, or volume name. If a volume name is specified, the /mp parameter is unnecessary. |
| <Label> | Specifies the label for the volume. |
| /? | Displays help at the command prompt. |

Remarks

- Windows displays the volume label and serial number (if it has one) as part of the directory listing.
- An NTFS volume label can be up to 32 characters in length, including spaces. NTFS volume labels retain and display the case that was used when the label was created.
- If you do not specify a value for the **Label** parameter, the **label** command displays output in the following format:

```
Volume in drive C: xxxxxxxxxxxx Volume Serial Number is xxxx-xxxx Volume label (32 characters, ENTER for none)?
```

You can type a new volume label or press ENTER to keep the current label. If you press ENTER and the volume currently has a label, the **label** command prompts you with the following message:

```
Delete current volume label (Y/N)?
```

Press Y to delete the label, or press N to keep the label.

Examples

To label a disk in drive A that contains sales information for July, type:

```
label a:sales-july
```

To delete the current label for drive C, follow these steps:

1. At the command prompt, type:

```
Label
```

Output similar to the following should be displayed:

```
Volume in drive C: is Main Disk Volume Serial Number is 6789-ABCD Volume label (32 characters, ENTER for none)?
```

2. Press ENTER. The following prompt should be displayed:

```
Delete current volume label (Y/N)?
```

3. Press Y to delete the current label.

Additional references

[Command-Line Syntax Key](#)

lodctr

4/13/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Allows you to register or save performance counter name and registry settings in a file and designate trusted services.

Syntax

```
lodctr <filename> [/s:<filename>] [/r:<filename>] [/t:<servicename>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| | Registers the Performance counter name settings and Explain text provided in initialization file FileName. |
| /s: | Saves Performance counter registry settings and Explain text to file . |
| /r | Restores counter registry settings and Explain text from current registry settings and cached performance files related to the registry. This option is available only in the Windows Server 2003 operating system. |
| /r: | Restores Performance counter registry settings and Explain text from file . Warning: if you use the lodctr /r command, you will overwrite all Performance counter registry settings and Explain text, replacing them with the configuration defined in the file specified. |
| /t: | Indicates that service is trusted. |
| /? | Displays help at the command prompt. |

Remarks

If the information that you supply contains spaces, use quotation marks around the text (for example, "").

Examples

To save the current Performance registry settings and counter Explain text to file **perf backup1.txt**:

```
lodctr /s:"perf backup1.txt"
```

additional references

- [Command-Line Syntax Key](#)

logman

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

logman creates and manages Event Trace Session and Performance logs and supports many functions of Performance Monitor from the command line.

Syntax

```
logman [create | query | start | stop | delete | update | import | export | /?] [options]
```

Actions

| ACTION | DESCRIPTION |
|--|---|
| logman create | create a counter, trace, configuration data collector, or API. |
| logman query | query data collector properties. |
| logman start stop | start or stop data collection. |
| logman delete | delete an existing data collector. |
| logman update | Update the properties of an existing data collector. |
| logman import export | import a data collector set from an XML file or export a data collector set to an XML file. |

logman create

4/13/2018 • 2 minutes to read • [Edit Online](#)

Syntax

```
logman create <counter | trace | alert | cfg | api> <[-n] <name>> [options]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------------------------------|--|
| logman create counter | Create a counter data collector. |
| logman create trace | Create a trace data collector. |
| logman create alert | Create an alert data collector. |
| logman create cfg | Create a configuration data collector. |
| logman create api | Create an API tracing data collector. |

Additional references

logman query

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

query data collector or data collector set properties.

Syntax

```
logman query [providers|"Data Collector Set name"] [options]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| /? | Displays context-sensitive help. |
| -s | Perform the command on the specified remote computer. |
| -config | Specifies the settings file containing command options. |
| [-n] | Name of the target object. |
| -ets | Send commands to Event Trace Sessions directly without saving or scheduling. |

Examples

The following command lists all Data Collector Sets configured on the target system.

```
logman query
```

The following command lists the data collectors contained in the Data Collector Set named perf_log.

```
logman query "perf_log"
```

The following command lists all available providers of data collectors on the target system.

```
logman query providers
```

additional references

[logman](#)

logman start | stop

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

start a data collector and set the begin time to manual, or stop a data collector set and set the end time to manual.

Syntax

```
logman start <[-n] <name>> [options]
logman stop <[-n] <name>> [options]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| -? | Displays context-sensitive help. |
| -s | Perform the command on the specified remote computer. |
| -config | Specifies the settings file containing command options. |
| [-n] | Name of the target object. |
| -ets | Send commands to Event Trace Sessions directly without saving or scheduling. |
| -as | Perform the requested operation asynchronously. |

Examples

The following command starts the data collector perf_log on the remote computer server_1.

```
logman start perf_log -s server_1
```

additional references

[logman](#)

logman delete

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

delete an existing data collector.

Syntax

```
logman delete <[-n] <name>> [options]
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------------------|--|
| /? | Displays context-sensitive help. |
| -s | Perform the command on the specified remote computer. |
| -config | Specifies the settings file containing command options. |
| [-n] | Name of the target data collector. |
| -ets | Send commands to Event Trace Sessions directly without saving or scheduling. |
| -[-]u <user [password]> | Specifies the user to Run As. Entering a * for the password produces a prompt for the password. The password is not displayed when you type it at the password prompt. |

Examples

The following command deletes the data collector perf_log.

```
logman delete perf_log
```

additional references

[logman](#)

logman update

4/13/2018 • 2 minutes to read • [Edit Online](#)

Update an existing data collector.

Syntax

```
logman update <counter | trace | alert | cfg | api> <[-n] <name>> [options]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------------------------------|--|
| logman update counter | Update a counter data collector. |
| logman update trace | Update a trace data collector. |
| logman update alert | Update an alert data collector. |
| logman update cfg | Update a configuration data collector. |
| logman update api | Update an API tracing data collector. |

Additional references

[Logman](#)

logman import | export

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

import a Data Collector Set from an XML file, or export a Data Collector Set to an XML file.

Syntax

```
logman import <[-n] <name>> <-xml <name>> [options]
logman export <[-n] <name>> <-xml <name>> [options]
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------------------|--|
| -? | Displays context-sensitive help. |
| -s | Perform the command on the specified remote computer. |
| -config | Specifies the settings file containing command options. |
| [-n] | Name of the target object. |
| -xml | Name of the XML file to import or export. |
| -ets | Send commands to Event Trace Sessions directly without saving or scheduling. |
| -[-]u <user [password]> | User to Run As. Entering a * for the password produces a prompt for the password. The password is not displayed when you type it at the password prompt. |
| -y | Answer yes to all questions without prompting. |

Examples

The following command imports the XML file c:\windows\perf_log.xml from the computer server_1 as a data collector set called perf_log.

```
logman import perf_log -s server_1 -xml "c:\windows\perf_log.xml"
```

additional references

[logman](#)

logoff

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Logs off a user from a session on a Remote Desktop Session Host (rd Session Host) server and deletes the session from the server. for examples of how to use this command, see [Examples](#).

NOTE

In Windows Server 2008 R2, Terminal Services was renamed Remote Desktop Services. To find out what's new in the latest version, see [What's New in Remote Desktop Services in Windows Server 2012](#) in the Windows Server TechNet Library.

Syntax

```
logoff [<SessionName> | <SessionID>] [/server:<ServerName>] [/v]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| | Specifies the name of the session. |
| | Specifies the numeric ID which identifies the session to the server. |
| /server: | Specifies the rd Session Host server that contains the session whose user you want to log off. If unspecified, the server on which you are currently active is used. |
| /v | Displays information about the actions being performed. |
| /? | Displays help at the command prompt. |

Remarks

- You can always log off from the session to which you are currently logged on. You must, however, have Full Control permission to log off users from other sessions.
- Logging off a user from a session without warning can result in loss of data at the user's session. You should send a message to the user by using the **msg** command to warn the user before taking this action.
- if <SessionID> or <SessionName> is not specified, **logoff** logs off the user from the current session. If you specify <SessionName>, it must be an active one.
- When you log off a user, all processes end and the session is deleted from the server.
- You cannot log off a user from the console session. ## Examples
- To log off a user from the current session, type: `logoff`

- To log off a user from a session by using the session's ID, for example session 12, type: `logoff 12`
- To log off a user from a session by using the name of the session and server, for example session TERM04 on Server1, type: `logoff TERM04 /server:Server1`

additional references

- [Command-Line Syntax Key](#)
- [Remote Desktop Services \(Terminal Services\) Command Reference](#)

lpq

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Displays the status of a print queue on a computer running Line printer Daemon (LPD).

Syntax

```
lpq -S <ServerName> -P <printerName> [-l]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| -S | Specifies (by name or IP address) the computer or printer sharing device that hosts the LPD print queue with a status that you want to display. Required. |
| -P | Specifies (by name) the printer for the print queue with a status that you want to display. Required. |
| -l | Specifies that you want to display details about the status of the print queue. |
| /? | Displays help at the command prompt. |

Remarks

The **-S** and **-P** parameters are case sensitive and must be typed in upper-case letters.

Examples

This example shows how to display the status of the Laserprinter1 printer queue on an LPD host at 10.0.0.45:

```
lpq -S 10.0.0.45 -P Laserprinter1
```

additional references

[Command-Line Syntax Key](#)
[print Command Reference](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Sends a file to a computer or printer sharing device running the Line printer Daemon (LPD) service in preparation for printing.

Syntax

```
lpr [-S <ServerName>] -P <printerName> [-C <BannerContent>] [-J <JobName>] [-o | "-o l"] [-x] [-d] <filename>
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------|---|
| -S | Specifies (by name or IP address) the computer or printer sharing device that hosts the LPD print queue with a status that you want to display. Required. |
| -P | Specifies (by name) the printer for the print queue with a status that you want to display. Required. |
| -C | Specifies the content to print on the banner page of the print job. If you do not include this parameter, the name of the computer from which the print job was sent appears on the banner page. |
| -J | Specifies the print job name that will be printed on the banner page. If you do not include this parameter, the name of the file being printed appears on the banner page. |
| [-o "-o l"] | Specifies the type of file that you want to print. The parameter -o specifies that you want to print a text file. The parameter "-o l" specifies that you want to print a binary file (for example, a PostScript file). |
| -d | Specifies that the data file must be sent before the control file. Use this parameter if your printer requires the data file to be sent first. For more information, see your printer documentation. |
| -x | Specifies that the lpr command must be compatible with the Sun Microsystems operating system (referred to as SunOS) for releases up to and including 4.1.4_u1. |
| | Specifies (by name) the file to be printed. Required. |
| /? | Displays help at the command prompt. |

Remarks

- To find the name of the printer, open the printers folder.
- The **-S**, **-P**, **-C**, and **-J** parameters are case sensitive and must be typed in upper-case letters.

Examples

This example shows how to print the "Document.txt" text file to the Laserprinter1 printer queue on an LPD host at 10.0.0.45:

```
lpr -S 10.0.0.45 -P Laserprinter1 -o Document.txt
```

This example shows how to print the "PostScript_file.ps" Adobe PostScript file to the Laserprinter1 printer queue on an LPD host at 10.0.0.45:

```
lpr -S 10.0.0.45 -P Laserprinter1 "-o 1" PostScript_file.ps
```

additional references

- [Command-Line Syntax Key](#)
- [print Command Reference](#)

macfile

7/11/2018 • 6 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Manages File Server for Macintosh servers, volumes, directories, and files. You can automate administrative tasks by including a series of commands in batch files and starting them manually or at predetermined times.

- [To modify directories in Macintosh-accessible volumes](#)
- [To join a Macintosh file's data and resource forks](#)
- [To change the logon message and limit sessions](#)
- [To add, change, or remove Macintosh-accessible volumes](#)

To modify directories in Macintosh-accessible volumes

Syntax

```
macfile directory[/server:\\<computerName>] /path:<directory> [/owner:<OwnerName>] [/group:<GroupName>] [/permissions:<Permissions>]
```

Parameters

- /server:\\ Specifies the server on which to change a directory. If omitted, the operation is performed on the local computer.
- /path: Required. Specifies the path to the directory that you want to change. The directory must exist. **macfile directory** does not create directories.
- /owner: changes the owner of the directory. If omitted, the owner remains unchanged.
- /group: Specifies or changes the Macintosh primary group that is associated with the directory. If omitted, the primary group remains unchanged.
- /permissions: Sets permissions on the directory for the owner, primary group, and world (everyone). An 11-digit number is used to set permissions. The number 1 grants permission and 0 revokes permission (for example, 11111011000). If omitted, permissions remain unchanged. The position of the digit determines which permission is set, as described in the following table.

| POSITION | SETS PERMISSION FOR |
|----------|---------------------|
| First | OwnerSeeFiles |
| Second | OwnerSeeFolders |
| Third | OwnerMakechanges |
| Fourth | GroupSeeFiles |
| Fifth | GroupSeeFolders |
| Sixth | GroupMakechanges |

| POSITION | SETS PERMISSION FOR |
|----------|--|
| Seventh | WorldSeeFiles |
| Eighth | WorldSeeFolders |
| Ninth | WorldMakechanges |
| Tenth | The directory cannot be renamed, moved, or deleted. |
| Eleventh | The changes apply to the current directory and all subdirectories. |

- `/?` Displays help at the command prompt.

Remarks

- if the information that you supply contains spaces or special characters, use quotation marks around the text (for example, "*computer Name*").
- Use **macfiledirectory** to make an existing directory in a Macintosh-accessible volume available to Macintosh users. The **macfiledirectory** command does not create directories. Use File Manager, the command prompt, or the **macintosh new folder** command to create a directory in a Macintosh-accessible volume before you use the **macfile directory** command. ### Examples The following example changes the permissions of the subdirectory May sales, in the Macintosh-accessible volume Statistics, on the E drive of the local server. The example assigns See Files, See Folders, and Make changes permissions to the owner and See Files and See Folders permissions to all other users, while preventing the directory from being renamed, moved, or deleted.

```
macfile directory /path:"e:\statistics\may sales" /permissions:11111011000
```

To join a Macintosh file's data and resource forks

Syntax

```
macfile forkize[/server:\\<computerName>] [/creator:<CreatorName>] [/type:<typeName>] [/datafork:<Filepath>]
[/resourcefork:<Filepath>] /targetfile:<Filepath>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------------------|---|
| <code>/server:\\</code> | Specifies the server on which to join files. If omitted, the operation is performed on the local computer. |
| <code>/creator:</code> | Specifies the creator of the file. The Macintosh finder uses the /creator command-line option to determine the application that created the file. |
| <code>/type:</code> | Specifies the type of file. The Macintosh finder uses the /type command-line option to determine the file type within the application that created the file. |
| <code>/datafork:</code> | Specifies the location of the data fork that is to be joined. You can specify a remote path. |
| <code>/resourcefork:</code> | Specifies the location of the resource fork that is to be joined. You can specify a remote path. |

| PARAMETER | DESCRIPTION |
|--------------|--|
| /targetfile: | Required. Specifies the location of the file that is created by joining a data fork and a resource fork, or specifies the location of the file whose type or creator you are changing. The file must be on the specified server. |
| /? | Displays help at the command prompt. |

Remarks

- if the information that you supply contains spaces or special characters, use quotation marks around the text (for example, "*computer Name*").

Examples

To create the file treeapp on the Macintosh-accessible volume D:\Release, using the resource fork C:\Cross\Mac\Appcode, and to make this new file appear to Macintosh clients as an application (Macintosh applications use the type APPL) with the creator (signature) set to MAGNOLIA, type:

```
macfile forkize /resourcefork:c:\cross\mac\appcode /type:APPL /creator:MAGNOLIA /targetfile:D:\Release\treeapp
```

To change the file creator to Microsoft Word 5.1, for the file WOrd.txt in the directory D:\Word documents\Group files, on the server \\SERverA, type:

```
macfile forkize /server:\\servera /creator:MSWD /type:TEXT /targetfile:"d:\Word documents\Group files\Word.txt"
```

To change the logon message and limit sessions

Syntax

```
macfile server [/server:\\<computerName>] [/maxsessions:{Number | unlimited}] [/loginmessage:<Message>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------------------------|---|
| /server:\\ | Specifies the server on which to change parameters. If omitted, the operation is performed on the local computer. |
| /maxsessions:{Number unlimited} | Specifies the maximum number of users who can simultaneously use File and print Servers for Macintosh. If omitted, the maxsessions setting for the server remains unchanged. |
| /loginmessage: | changes the message Macintosh users see when logging on to the File Server for Macintosh server. The maximum number of characters for the logon message is 199. If omitted, the loginmessage message for the server remains unchanged. To remove an existing logon message, include the /loginmessage parameter, but leave the <i>Message</i> variable blank. |
| /? | Displays help at the command prompt. |

Remarks

- if the information that you supply contains spaces or special characters, use quotation marks around the text (for example, "*computer Name*").

Examples

To change the number of File and print Server for Macintosh sessions that are permitted on the local server from the current setting to five sessions, and to add the logon message "Log off from Server for Macintosh when you are finished.", type:

```
macfile server /maxsessions:5 /loginmessage:"Log off from Server for Macintosh when you are finished."
```

To add, change, or remove Macintosh-accessible volumes

Syntax

```
macfile volume {/add|/set} [/server:\\<computerName>] /name:<volumeName>/path:<directory>[/readonly:{true | false}] [/guestsallowed:{true | false}] [/password:<Password>] [/maxusers:{<Number>>|unlimited}]
macfile volume /remove[/server:\\<computerName>] /name:<volumeName>
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------------------------|---|
| {/add /set} | Required when you are adding or changing a Macintosh-accessible volume. adds or changes the specified volume. |
| /server:\\ | Specifies the server on which to add, change, or remove a volume. If omitted, the operation is performed on the local computer. |
| /name: | Required. Specifies the volume name to be added, changed, or removed. |
| /path: | Required and valid only when you are adding a volume. Specifies the path to the root directory of the volume to be added. |
| /readonly:{true false} | Specifies whether users can change files in the volume. type true to specify that users cannot change files in the volume. type false to specify that users can change files in the volume. If omitted when adding a volume, changes to files are allowed. If omitted when changing a volume, the readonly setting for the volume remains unchanged. |
| /guestsallowed:{true false} | Specifies whether users who log on as guests can use the volume. type true to specify that guests can use the volume. type false to specify that guests cannot use the volume. If omitted when adding a volume, guests can use the volume. If omitted when changing a volume, the guestsallowed setting for the volume remains unchanged. |
| /password: | Specifies a password that will be required to access the volume. If omitted when adding a volume, no password is created. If omitted when changing a volume, the password remains unchanged. |

| PARAMETER | DESCRIPTION |
|-------------------------|---|
| /maxusers:{> unlimited} | Specifies the maximum number of users who can simultaneously use the files on the volume. If omitted when adding a volume, an unlimited number of users can use the volume. If omitted when changing a volume, the maxusers value remains unchanged. |
| /remove | Required when you are removing a Macintosh-accessible volume. removes the specified volume. |
| /? | Displays help at the command prompt. |

Remarks

- if the information that you supply contains spaces or special characters, use quotation marks around the text (for example, "*computer Name*").

Examples

To create a volume called US Marketing Statistics on the local server, using the Stats directory in the E drive, and to specify that the volume cannot be accessed by guests, type:

```
macfile volume /add /name:"US Marketing Statistics" /guestsallowed:false /path:e:\Stats
```

To change the volume created above to be read-only and to require a password, and to set the number of maximum users to five, type:

```
macfile volume /set /name:"US Marketing Statistics" /readonly:true /password:saturn /maxusers:5
```

To add a volume called Landscape Design, on the server \\Magnolia, using the trees directory in the E drive, and to specify that the volume can be accessed by guests, type:

```
macfile volume /add /server:\\Magnolia /name:"Landscape Design" /path:e:\trees
```

To remove the volume called Sales Reports on the local server, type:

```
macfile volume /remove /name:"Sales Reports"
```

additional references

- [Command-Line Syntax Key](#)

makecab

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Package existing files into a cabinet (.cab) file.

Syntax

```
makecab [/v[n]] [/d var=<value> ...] [/l <dir>] <source> [<destination>]
makecab [/v[<n>]] [/d var=<value> ...] /f <directives_file> [...]
```

Parameters

| PARAMETER | DESCRIPTION |
|----------------------|---|
| | File to compress. |
| | File name to give compressed file. If omitted, the last character of the source file name is replaced with an underscore (_) and used as the destination. |
| /f <directives_file> | A file with makecab directives (may be repeated). |
| /d var= | Defines variable with specified value. |
| /l | Location to place destination (default is current directory). |
| /v[] | Set debugging verbosity level (0=none,...,3=full). |
| /? | Displays help at the command prompt. |

Remarks

- Refer to [Microsoft Cabinet format](#) on MSDN for information on directive_file.

additional references

- [Command-Line Syntax Key](#)

manage-bde

4/13/2018 • 2 minutes to read • [Edit Online](#)

Used to turn on or turn off BitLocker, specify unlock mechanisms, update recovery methods, and unlock BitLocker-protected data drives. This command-line tool can be used in place of the **BitLocker Drive Encryption** Control Panel item. For examples of how this command can be used, see [Examples](#).

Syntax

```
manage-bde [-status] [-on] [-off] [-pause] [-resume] [-lock] [-unlock] [-autounlock] [-protectors] [-tpm] [-SetIdentifier] [-ForceRecovery] [-changepassword] [-changePIN] [-changekey] [-KeyPackage] [-upgrade] [-WipeFreeSpace] [{-?|/?}] [{-help|-h}]
```

Parameters

| PARAMETER | DESCRIPTION |
|---|---|
| Manage-bde: status | Provides information about all drives on the computer, whether or not they are BitLocker-protected. |
| Manage-bde: on | Encrypts the drive and turns on BitLocker. |
| Manage-bde: off | Decrypts the drive and turns off BitLocker. All key protectors are removed when decryption is complete. |
| Manage-bde: pause | Pauses encryption or decryption. |
| Manage-bde: resume | Resumes encryption or decryption. |
| Manage-bde: lock | Prevents access to BitLocker-protected data. |
| Manage-bde: unlock | Allows access to BitLocker-protected data with a recovery password or a recovery key. |
| Manage-bde: autounlock | Manages automatic unlocking of data drives. |
| Manage-bde: protectors | Manages protection methods for the encryption key. |
| Manage-bde: tpm | Configures the computer's Trusted Platform Module (TPM). This command is not supported on computers running Windows 8 or win8_server_2 . To manage the TPM on these computers, use either the TPM Management MMC snap-in or the TPM Management cmdlets for Windows PowerShell. |
| Manage-bde: setidentifier | Sets the drive identifier field on the drive to the value specified in the Provide the unique identifiers for your organization Group Policy setting. |

| PARAMETER | DESCRIPTION |
|--|---|
| Manage-bde: ForceRecovery | Forces a BitLocker-protected drive into recovery mode on restart. This command deletes all TPM-related key protectors from the drive. When the computer restarts, only a recovery password or recovery key can be used to unlock the drive. |
| Manage-bde: changepassword | Modifies the password for a data drive. |
| Manage-bde: changepin | Modifies the PIN for an operating system drive. |
| Manage-bde: changekey | Modifies the startup key for an operating system drive. |
| Manage-bde: KeyPackage | Generates a key package for a drive. |
| Manage-bde: upgrade | Upgrades the BitLocker version. |
| Manage-bde: WipeFreeSpace | Wipes the free space on a drive. |
| -? or /? | Displays brief Help at the command prompt. |
| -help or -h | Displays complete Help at the command prompt. |

Examples

The following example displays the drives on the computer and identifies whether or not they are BitLocker-protected and the current encryption status.

```
manage-bde -status
```

The following example illustrates enabling BitLocker on drive C with the option of a recovery password. The recovery password will be generated by BitLocker and displayed on the screen so that you can record it.

```
manage-bde -on C: -recoverypassword
```

The following example illustrates unlocking a BitLocker-protected drive by using a recovery password.

```
manage-bde -unlock E: -recoverypassword 111111-222222-333333-444444-555555-666666-777777-888888
```

Additional references

- [Command-Line Syntax Key](#)
- [Enabling BitLocker by Using the Command Line](#)

manage-bde: status

4/13/2018 • 2 minutes to read • [Edit Online](#)

Provides the following information about all drives on the computer; whether or not they are BitLocker-protected:

- Size
- BitLocker version
- Conversion status
- Percentage encrypted
- Encryption method
- Protection status
- Lock status
- Identification field
- Key protectors

For examples of how this command can be used, see [Examples](#).

Syntax

```
manage-bde -status [<Drive>] [-protectionaserrorlevel] [-computername <Name>] [{-?|/?}] [{-help|-h}]
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------------------|--|
| <Drive> | Represents a drive letter followed by a colon. |
| -protectionaserrorlevel | Causes the Manage-bde command-line tool to send the return code of 0 when the volume is protected and 1 when the volume is unprotected; most commonly used for batch scripts to determine if a drive is BitLocker-protected. You can also use -p as an abbreviated version of this command. |
| -computername | Specifies that Manage-bde.exe will be used to modify BitLocker protection on a different computer. You can also use -cn as an abbreviated version of this command. |
| <Name> | Represents the name of the computer on which to modify BitLocker protection. Accepted values include the computer's NetBIOS name and the computer's IP address. |
| -? or /? | Displays brief Help at the command prompt. |
| -help or -h | Displays complete Help at the command prompt. |

Examples

The following example illustrates using the **-status** command to display the status of drive C.

```
manage-bde -status C:
```

Additional references

- [Command-Line Syntax Key](#)
- [Manage-bde](#)

manage-bde: on

4/13/2018 • 4 minutes to read • [Edit Online](#)

Encrypts the drive and turns on BitLocker. For examples of how this command can be used, see [Examples](#).

Syntax

```
manage-bde -on <Drive> {[-recoveryPassword <NumericalPassword>][[-recoverykey <PathToExternalDirectory>][[-startupkey <PathToExternalKeyDirectory>][[-certificate][[-tpmandpin][[-tpmandpinandstartupkey <PathToExternalKeyDirectory>][[-tpmandstartupkey <PathToExternalKeyDirectory>][[-password][[-ADAccountOrGroup <Domain\Account>]]]]]]]] [-UsedSpaceOnly][[-encryptionmethod {aes128_diffuser|aes256_diffuser|aes128|aes256}]] [-skiphardwaretest] [-discoveryvolumetype <FileSystemType>] [-ForceEncryptionType <type>] [-RemoveVolumeShadowCopies][[-computername <Name>]] [-?|/?]} [-help|-h]}
```

Parameters

| PARAMETER | DESCRIPTION |
|------------------------------|--|
| <Drive> | Represents a drive letter followed by a colon. |
| -recoverypassword | Adds a numerical password protector. You can also use -rp as an abbreviated version of this command. |
| <NumericalPassword> | Represents the recovery password. |
| -recoverykey | Adds an external key protector for recovery. You can also use -rk as an abbreviated version of this command. |
| <PathToExternalDirectory> | Represents the directory path to the recovery key. |
| -startupkey | Adds an external key protector for startup. You can also use -sk as an abbreviated version of this command. |
| <PathToExternalKeyDirectory> | Represents the directory path to the startup key. |
| -certificate | Adds a public key protector for a data drive. You can also use -cert as an abbreviated version of this command. |
| -tpmandpin | Adds a Trusted Platform Module (TPM) and personal identification number (PIN) protector for the operating system drive. You can also use -tp as an abbreviated version of this command. |
| -tpmandstartupkey | Adds a TPM and startup key protector for the operating system drive. You can also use -tsk as an abbreviated version of this command. |
| -tpmandpinandstartupkey | Adds a TPM, PIN, and startup key protector for the operating system drive. You can also use -tpsk as an abbreviated version of this command. |

| PARAMETER | DESCRIPTION |
|---------------------------|---|
| -password | Adds a password key protector for the data drive. You can also use -pw as an abbreviated version of this command. |
| -ADAccountOrGroup | Adds a SID-based identity protector for the volume. The volume will automatically unlock if the user or computer has the proper credentials. When specifying a computer account, append a \$ to the computer name and specify -service to indicate that the unlock should happen in the content of the BitLocker server instead of the user. You can also use -sid as an abbreviated version of this command. |
| -UsedSpaceOnly | Sets the encryption mode to Used Space Only encryption. The sections of the volume containing used space will be encrypted but the free space will not. If this option is not specified, all used space and free space on the volume will be encrypted.. You can also use -used as an abbreviated version of this command. |
| -encryptionMethod | Configures the encryption algorithm and key size. You can also use -em as an abbreviated version of this command. |
| -skiphardwaretest | Begins encryption without a hardware test. You can also use -s as an abbreviated version of this command. |
| -discoveryvolumetype | Specifies the file system to use for the discovery data drive. The discovery data drive is a hidden drive added to a FAT-formatted, BitLocker-protected removable data drive that contains the BitLocker To Go Reader so that Windows Vista or Windows XP operating systems can be used to view BitLocker-protected drives. |
| -ForceEncryptionType | Forces BitLocker to use either software or hardware encryption. You can specify either Hardware or Software as the encryption type. If the hardware parameter is selected, but the drive does not support hardware encryption, manage-bde returns an error. If Group Policy settings forbids the specified encryption type, manage-bde returns an error. You can also use -fet as an abbreviated version of this command. |
| -RemoveVolumeShadowCopies | Force deletion of Volume Shadow Copies for the volume. You will not be able to restore this volume using previous system restore points after running this command. You can also use -rvsc as an abbreviated version of this command. |
| <FileSystemType> | Specifies which file systems can be used with discovery data drives: FAT32, default, or none. |
| -computername | Specifies that Manage-bde is being used to modify BitLocker protection on a different computer. You can also use -cn as an abbreviated version of this command. |
| <Name> | Represents the name of the computer on which to modify BitLocker protection. Accepted values include the computer's NetBIOS name and the computer's IP address. |

| PARAMETER | DESCRIPTION |
|-------------|---|
| -? or /? | Displays brief Help at the command prompt. |
| -help or -h | Displays complete Help at the command prompt. |

Examples

The following example illustrates using the **-on** command to turn on BitLocker for drive C and add a recovery password to the drive.

```
manage-bde -on C: -recoverypassword
```

The following example illustrates using the **-on** command to turn on BitLocker for drive C, add a recovery password to the drive, and save a recovery key to drive E.

```
manage-bde -on C: -recoverykey E:\ -recoverypassword
```

The following example illustrates using the **-on** command to turn on BitLocker for drive C by using an external key protector (such as a USB key) to unlock the operating system drive. This method is required if you are using BitLocker with computers that do not have a TPM.

```
manage-bde -on C: -startupkey E:\
```

The following example illustrates using the **-on** command to turn on BitLocker for data drive E and add a password key protector. Manage-bde will prompt you to enter the password after this command has been entered.

```
manage-bde -on E: -pw
```

The following example illustrates using the **-on** command to turn on BitLocker for operating system drive C and use hardware-based encryption.

```
manage-bde -on C: -fet Hardware
```

Additional references

- [Command-Line Syntax Key](#)
- [Manage-bde](#)

manage-bde: off

4/13/2018 • 2 minutes to read • [Edit Online](#)

Decrypts the drive and turns off BitLocker. All key protectors are removed when decryption is complete. For examples of how this command can be used, see [Examples](#).

Syntax

```
manage-bde -off [<Volume>] [-computername <Name>] [{-?|/?}] [{-help|-h}]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------|---|
| <Volume> | A drive letter followed by a colon, a volume GUID path, or a mounted volume. |
| -computername | Specifies that Manage-bde.exe will be used to modify BitLocker protection on a different computer. You can also use -cn as an abbreviated version of this command. |
| <Name> | Represents the name of the computer on which to modify BitLocker protection. Accepted values include the computer's NetBIOS name and the computer's IP address. |
| -? or /? | Displays brief Help at the command prompt. |
| -help or -h | Displays complete Help at the command prompt. |

Examples

The following example illustrates using the **-off** command to turn off BitLocker on drive C.

```
manage-bde -off C:
```

Additional references

- [Command-Line Syntax Key](#)
- [Manage-bde](#)

manage-bde: pause

4/13/2018 • 2 minutes to read • [Edit Online](#)

Pauses BitLocker encryption or decryption. For examples of how this command can be used, see [Examples](#).

Syntax

```
manage-bde -pause <Volume> [-computername <Name>] [{-?|/?}] [{-help|-h}]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------|---|
| <Volume> | A drive letter followed by a colon, a volume GUID path, or a mounted volume. |
| -computername | Specifies that Manage-bde.exe will be used to modify BitLocker protection on a different computer. You can also use -cn as an abbreviated version of this command. |
| <Name> | Represents the name of the computer on which to modify BitLocker protection. Accepted values include the computer's NetBIOS name and the computer's IP address. |
| -? or /? | Displays brief Help at the command prompt. |
| -help or -h | Displays complete Help at the command prompt. |

Examples

The following example illustrates using the **-pause** command to pause BitLocker encryption on drive C.

```
manage-bde -pause C:
```

Additional references

- [Command-Line Syntax Key](#)
- [Manage-bde](#)

manage-bde: resume

4/13/2018 • 2 minutes to read • [Edit Online](#)

Resumes BitLocker encryption or decryption after it has been paused. For examples of how this command can be used, see [Examples](#).

Syntax

```
manage-bde -resume [<Drive>] [-computername <Name>] [{-?|/?}] [{-help|-h}]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------|---|
| <Drive> | Represents a drive letter followed by a colon. |
| -computername | Specifies that Manage-bde.exe will be used to modify BitLocker protection on a different computer. You can also use -cn as an abbreviated version of this command. |
| <Name> | Represents the name of the computer on which to modify BitLocker protection. Accepted values include the computer's NetBIOS name and the computer's IP address. |
| -? or /? | Displays brief Help at the command prompt. |
| -help or -h | Displays complete Help at the command prompt. |

Examples

The following example illustrates using the **-resume** command to resume BitLocker encryption on drive C.

```
manage-bde -resume C:
```

Additional references

- [Command-Line Syntax Key](#)
- [Manage-bde](#)

manage-bde: lock

4/13/2018 • 2 minutes to read • [Edit Online](#)

Locks a BitLocker-protected drive to prevent access to it unless the unlock key is provided. For examples of how this command can be used, see [Examples](#).

Syntax

```
manage-bde -lock [<Drive>] [-computername <Name>] [{-?|/?}] [{-help|-h}]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------|---|
| <Drive> | Represents a drive letter followed by a colon. |
| -computername | Specifies that Manage-bde.exe will be used to modify BitLocker protection on a different computer. You can also use -cn as an abbreviated version of this command. |
| <Name> | Represents the name of the computer on which to modify BitLocker protection. Accepted values include the computer's NetBIOS name and the computer's IP address. |
| -? or /? | Displays brief Help at the command prompt. |
| -help or -h | Displays complete Help at the command prompt. |

Examples

The following example illustrates using the **-lock** command to lock data drive D.

```
manage-bde -lock D:
```

Additional references

- [Command-Line Syntax Key](#)
- [Manage-bde](#)

manage-bde: unlock

4/13/2018 • 2 minutes to read • [Edit Online](#)

Unlocks a BitLocker-protected drive by using a recovery password or a recovery key. For examples of how this command can be used, see [Examples](#).

Syntax

```
manage-bde -unlock {-recoverypassword <Password>|-recoverykey <PathToExternalKeyFile>} <Drive> [-certificate  
{-cf PathToCertificateFile | -ct CertificateThumbprint} {-pin}] [-password] [-computername <Name>] [{-?/?}]  
[{-help|-h}]
```

Parameters

| PARAMETER | VALUE | DESCRIPTION |
|-------------------|-----------------------------|---|
| -recoverypassword | | Specifies that a recovery password will be used to unlock the drive. Abbreviation: -rp |
| | <Password> | Represents the recovery password that can be used to unlock the drive. |
| -recoverykey | | Specifies that an external recovery key file will be used to unlock the drive. Abbreviation: -rk |
| | <PathToExternalKeyFile> | Represents the external recovery key file that can be used to unlock the drive. |
| | <Drive> | Represents a drive letter followed by a colon. |
| -certificate | | The local user certificate for a BitLocker certificate to unlock the volume is located in the local user certificate store. Abbreviation: -cert |
| | <-cf PathToCertificateFile> | Path to the certificate file |
| | <-ct CertificateThumbprint> | Certificate thumbprint which may optionally include the PIN (-pin). |
| -password | | Presents a prompt for the password to unlock the volume. Abbreviation: -pw |
| -computername | | Specifies that Manage-bde.exe will be used to modify BitLocker protection on a different computer. Abbreviation: -cn |

| PARAMETER | VALUE | DESCRIPTION |
|-------------|--------|---|
| | <Name> | Represents the name of the computer on which to modify BitLocker protection. Accepted values include the computer's NetBIOS name and the computer's IP address. |
| -? or /? | | Displays brief Help at the command prompt. |
| -help or -h | | Displays complete Help at the command prompt. |

Examples

The following example illustrates using the **-unlock** command to unlock drive E with a recovery key file that has been saved to a backup folder on another drive.

```
manage-bde -unlock E: -recoverykey "F:\Backupkeys\recoverykey.bek"
```

Additional references

- [Command-Line Syntax Key](#)
- [Manage-bde](#)

manage-bde: autounlock

4/13/2018 • 2 minutes to read • [Edit Online](#)

Manages the automatic unlocking of BitLocker-protected data drives. For examples of how this command can be used, see [Examples](#).

Syntax

```
manage-bde -autounlock [{-enable|-disable|-clearallkeys}] <Drive> [-computername <Name>] [{-?|/?}] [{-help|-h}]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------|---|
| -enable | Enables automatic unlocking for a data drive. |
| -disable | Disables automatic unlocking for a data drive. |
| -clearallkeys | Removes all stored external keys on the operating system drive. |
| <Drive> | Represents a drive letter followed by a colon. |
| -computername | Specifies that Manage-bde.exe will be used to modify BitLocker protection on a different computer. You can also use -cn as an abbreviated version of this command. |
| <Name> | Represents the name of the computer on which to modify BitLocker protection. Accepted values include the computer's NetBIOS name and the computer's IP address. |
| -? or /? | Displays brief Help at the command prompt. |
| -help or -h | Displays complete Help at the command prompt. |

Examples

The following example illustrates using the **-autounlock** command to enable automatic unlocking of data drive E.

```
manage-bde -autounlock -enable E:
```

Additional references

- [Command-Line Syntax Key](#)
- [Manage-bde](#)

manage-bde: protectors

8/8/2018 • 6 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016

Manages the protection methods used for the BitLocker encryption key. For examples of how this command can be used, see [Examples](#).

Syntax

```
manage-bde -protectors [{-get|-add|-delete|-disable|-enable|-adbackup|-aadbackup}] <Drive> [-computername <Name>] [{-?|/?}] [{-help|-h}]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|--|
| -get | Displays all the key protection methods enabled on the drive and provides their type and identifier (ID). |
| -add | adds key protection methods as specified by using additional add parameters . |
| -delete | deletes key protection methods used by BitLocker. All key protectors will be removed from a drive unless the optional -delete parameters are used to specify which protectors to delete. When the last protector on a drive is deleted, BitLocker protection of the drive is disabled to ensure that access to data is not lost inadvertently. |
| -disable | Disables protection, which will allow anyone to access encrypted data by making the encryption key available unsecured on drive. No key protectors are removed. Protection will be resumed the next time Windows is booted unless the optional -disable parameters are used to specify the reboot count. |
| -enable | Enables protection by removing the unsecured encryption key from the drive. All configured key protectors on the drive will be enforced. |
| -adbackup | Backs up all recovery information for the drive specified to Active Directory Domain Services (AD DS). To back up only a single recovery key to AD DS, append the -id parameter and specify the ID of a specific recovery key to back up. |
| -aadbackup | Backs up all recovery information for the drive specified to Azure Active Directory (Azure AD). To back up only a single recovery key to Azure AD, append the -id parameter and specify the ID of a specific recovery key to back up. |
| | Represents a drive letter followed by a colon. |

| PARAMETER | DESCRIPTION |
|---------------|---|
| -computername | Specifies that manage-bde.exe will be used to modify BitLocker protection on a different computer. You can also use -cn as an abbreviated version of this command. |
| | Represents the name of the computer on which to modify BitLocker protection. Accepted values include the computer's NetBIOS name and the computer's IP address. |
| -? or /? | Displays brief help at the command prompt. |
| -help or -h | Displays complete help at the command prompt. |

-add syntax and parameters

```
manage-bde protectors add [<Drive>] [-forceupgrade] [-recoverypassword <NumericalPassword>] [-recoverykey
<pathToExternalKeydirectory>]
[-startupkey <pathToExternalKeydirectory>] [-certificate {-cf <pathToCertificateFile>|-ct
<CertificateThumbprint>}] [-tpm] [-tpmandpin]
[-tpmandstartupkey <pathToExternalKeydirectory>] [-tpmandpinandstartupkey <pathToExternalKeydirectory>] [-
password][-adaccountorgroup <securityidentifier> [-computername <Name>]
[{-?|/?}] [{-help|-h}]
```

| PARAMETER | DESCRIPTION |
|-------------------|--|
| | Represents a drive letter followed by a colon. |
| -recoverypassword | adds a numerical password protector. You can also use -rp as an abbreviated version of this command. |
| | Represents the recovery password. |
| -recoverykey | adds an external key protector for recovery. You can also use -rk as an abbreviated version of this command. |
| | Represents the directory path to the recovery key. |
| -startupkey | adds an external key protector for startup. You can also use -sk as an abbreviated version of this command. |
| | Represents the directory path to the startup key. |
| -certificate | adds a public key protector for a data drive. You can also use -cert as an abbreviated version of this command. |
| -cf | Specifies that a certificate file will be used to provide the public key certificate. |
| | Represents the directory path to the certificate file. |
| -ct | Specifies that a certificate thumbprint will be used to identify the public key certificate |

| PARAMETER | DESCRIPTION |
|-------------------------|---|
| | Specifies the value of the thumbprint property of the certificate you want to use. For example, a certificate thumbprint value of "a9 09 50 2d d8 2a e4 14 33 e6 f8 38 86 b0 0d 42 77 a3 2a 7b" should be specified as "a909502dd82ae41433e6f83886b00d4277a32a7b." |
| -tpmandpin | adds a Trusted Platform Module (TPM) and personal identification number (PIN) protector for the operating system drive. You can also use -tp as an abbreviated version of this command. |
| -tpmandstartupkey | adds a TPM and startup key protector for the operating system drive. You can also use -tsk as an abbreviated version of this command. |
| -tpmandpinandstartupkey | adds a TPM, PIN, and startup key protector for the operating system drive. You can also use -tpsk as an abbreviated version of this command. |
| -password | adds a password key protector for the data drive. You can also use -pw as an abbreviated version of this command. |
| -adaccountorgroup | adds a security identifier(SID)-based identity protector for the volume. You can also use -sid as an abbreviated version of this command. IMPORTANT: By default, you cannot add an ADAccountOrGroup protector remotely using either WMI or manage-bde. if your deployment requires the ability to add this protector remotely you must enable constrained delegation. |
| -computername | Specifies that manage-bde is being used to modify BitLocker protection on a different computer. You can also use -cn as an abbreviated version of this command. |
| | Represents the name of the computer on which to modify BitLocker protection. Accepted values include the computer's NetBIOS name and the computer's IP address. |

-delete syntax and parameters

```
manage-bde protectors delete <Drive> [-type
{recoverypassword|externalkey|certificate|tpm|tpmandstartupkey|tpmandpin|tpmandpinandstartupkey|Password|Identity}]
[-id <KeyProtectorID>] [-computername <Name>] [{-?|/?}] [{-help|-h}]
```

| PARAMETER | DESCRIPTION |
|------------------|---|
| | Represents a drive letter followed by a colon. |
| -type | Identifies the key protector to delete. You can also use -t as an abbreviated version of this command. |
| recoverypassword | Specifies that any recovery password key protectors should be deleted. |

| PARAMETER | DESCRIPTION |
|------------------------|---|
| externalkey | Specifies that any external key protectors associated with the drive should be deleted. |
| certificate | Specifies that any certificate key protectors associated with the drive should be deleted. |
| tpm | Specifies that any TPM-only key protectors associated with the drive should be deleted. |
| tpmandstartupkey | Specifies that any TPM and startup key based key protectors associated with the drive should be deleted. |
| tpmandpin | Specifies that any TPM and PIN based key protectors associated with the drive should be deleted. |
| tpmandpinandstartupkey | Specifies that any TPM, PIN, and startup key based key protectors associated with the drive should be deleted. |
| password | Specifies that any password key protectors associated with the drive should be deleted. |
| identity | Specifies that any identity key protectors associated with the drive should be deleted. |
| -id | Identifies the key protector to delete by using the key identifier. This parameter is an alternative option to the -type parameter. |
| | Identifies an individual key protector on the drive to delete. Key protector IDs can be displayed by using the manage-bde -protectors -get command. |
| -computername | Specifies that manage-bde.exe will be used to modify BitLocker protection on a different computer. You can also use -cn as an abbreviated version of this command. |
| | Represents the name of the computer on which to modify BitLocker protection. Accepted values include the computer's NetBIOS name and the computer's IP address. |
| -? or /? | Displays brief help at the command prompt. |
| -help or -h | Displays complete help at the command prompt. |

-disable syntax and parameters

```
manage-bde protectors disable <Drive> [-RebootCount <integer 0 - 15>] [-computername <Name>] [{-?|/?}] [{-help|-h}]
```

| PARAMETER | DESCRIPTION |
|-----------|--|
| | Represents a drive letter followed by a colon. |

| PARAMETER | DESCRIPTION |
|---------------|---|
| RebootCount | Starting with Windows 8, specifies that protection of the operating system volume has been suspended and will resume after Windows has been restarted the number of times specified in the RebootCount parameter. Specify 0 to suspend protection indefinitely. If this parameter is not specified BitLocker protection will automatically resume when Windows is restarted. You can also use -rc as an abbreviated version of this command. |
| -computername | Specifies that manage-bde.exe will be used to modify BitLocker protection on a different computer. You can also use -cn as an abbreviated version of this command. |
| | Represents the name of the computer on which to modify BitLocker protection. Accepted values include the computer's NetBIOS name and the computer's IP address. |
| -? or /? | Displays brief help at the command prompt. |
| -help or -h | Displays complete help at the command prompt. |

Examples

The following example illustrates using the **-protectors** command to add a certificate key protector identified by a certificate file to drive E.

```
manage-bde protectors add E: -certificate cf "c:\File Folder\Filename.cer"
```

The following example illustrates using the **-protectors** command to add an **adaccountorgroup** key protector identified by domain and user name to drive E.

```
manage-bde protectors add E: -sid DOMAIN\user
```

The following example illustrates using the **** protectors**** command to disable protection until the computer has rebooted 3 times.

```
manage-bde protectors disable C: -rc 3
```

The following example illustrates using the **-protectors** command to delete all TPM and startup key based key protectors on drive C.

```
manage-bde protectors delete C: -type tpmandstartupkey
```

The following example illustrates using the **-protectors** command to back up all recovery information for drive C to AD DS.

```
manage-bde protectors adbackup C:
```

additional references

- [Command-Line Syntax Key](#)
- [manage-bde](#)

manage-bde: tpm

10/24/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

IMPORTANT

This command is not supported for use on computers running Windows 8, Windows Server 2012 or later operating systems. For those computers, you can use the [TPM Management cmdlets for Windows PowerShell](#). If you are using this command on a computer running Windows 7 or Windows Server 2008, you can still configure the computer's Trusted Platform Module (TPM) using this command. For examples of how this command can be used, see [Examples](#).

Syntax

```
manage-bde -tpm [-turnon] [-takeownership <OwnerPassword>] [-computername <Name>] [{-?|/?}] [{-help|-h}]
```

Parameters

| PARAMETER | DESCRIPTION |
|----------------|---|
| -turnon | Enables and activates the TPM, allowing the TPM owner password to be set. You can also use -t as an abbreviated version of this command. |
| -takeownership | Takes ownership of the TPM by setting an owner password. You can also use -o as an abbreviated version of this command. |
| | Represents the owner password that you specify for the TPM. |
| -computername | Specifies that manage-bde.exe will be used to modify BitLocker protection on a different computer. You can also use -cn as an abbreviated version of this command. |
| | Represents the name of the computer on which to modify BitLocker protection. Accepted values include the computer's NetBIOS name and the computer's IP address. |
| -? or /? | Displays brief help at the command prompt. |
| -help or -h | Displays complete help at the command prompt. |

Examples

The following example illustrates using the **-tpm** command to turn on the TPM.

```
manage-bde tpm -turnon
```

The following example illustrates using the **** tpm**** command to take ownership of the TPM and set the owner password to OwnerP@ss.

```
manage-bde tpm takeownership 0wnerP@ss
```

additional references

- [Command-Line Syntax Key](#)
- [manage-bde](#)

manage-bde: setidentifier

4/13/2018 • 2 minutes to read • [Edit Online](#)

Sets the drive identifier field on the drive to the value specified in the **Provide the unique identifiers for your organization** Group Policy setting. For examples of how this command can be used, see [Examples](#).

Syntax

```
manage-bde -setidentifier <Drive> [-computername <Name>] [{-?|/?}] [{-help|-h}]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------|---|
| <Drive> | Represents a drive letter followed by a colon. |
| -computername | Specifies that Manage-bde.exe will be used to modify BitLocker protection on a different computer. You can also use -cn as an abbreviated version of this command. |
| <Name> | Represents the name of the computer on which to modify BitLocker protection. Accepted values include the computer's NetBIOS name and the computer's IP address. |
| -? or /? | Displays brief Help at the command prompt. |
| -help or -h | Displays complete Help at the command prompt. |

Examples

The following example illustrates using the **-setidentifier** command to set BitLocker drive identifier field for C.

```
manage-bde -setidentifier C:
```

Additional references

- [Command-Line Syntax Key](#)
- [Manage-bde](#)
- [Using Data Recovery Agents with BitLocker](#)

manage-bde: forcerecovery

4/13/2018 • 2 minutes to read • [Edit Online](#)

Forces a BitLocker-protected drive into recovery mode on restart. This command deletes all Trusted Platform Module (TPM)-related key protectors from the drive. When the computer restarts, only a recovery password or recovery key can be used to unlock the drive. For examples of how this command can be used, see [Examples](#).

Syntax

```
manage-bde -forcerecovery <Drive> [-computername <Name>] [{-?|/?}] [{-help|-h}]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------|---|
| <Drive> | Represents a drive letter followed by a colon. |
| -computername | Specifies that Manage-bde.exe will be used to modify BitLocker protection on a different computer. You can also use -cn as an abbreviated version of this command. |
| <Name> | Represents the name of the computer on which to modify BitLocker protection. Accepted values include the computer's NetBIOS name and the computer's IP address. |
| -? or /? | Displays brief Help at the command prompt. |
| -help or -h | Displays complete Help at the command prompt. |

Examples

The following example illustrates using the **-forcerecovery** command to cause BitLocker to start in recovery mode on drive C.

```
manage-bde -forcerecovery C:
```

Additional references

- [Command-Line Syntax Key](#)
- [Manage-bde](#)

manage-bde: changepassword

4/13/2018 • 2 minutes to read • [Edit Online](#)

Modifies the password for a data drive. The user is prompted for a new password. For examples of how this command can be used, see [Examples](#).

Syntax

```
manage-bde -changepassword [<Drive>] [-computername <Name>] [{-?|/?}] [{-help|-h}]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------|---|
| <Drive> | Represents a drive letter followed by a colon. |
| -computername | Specifies that Manage-bde.exe will be used to modify BitLocker protection on a different computer. You can also use -cn as an abbreviated version of this command. |
| <Name> | Represents the name of the computer on which to modify BitLocker protection. Accepted values include the computer's NetBIOS name and the computer's IP address. |
| -? or /? | Displays brief Help at the command prompt. |
| -help or -h | Displays complete Help at the command prompt. |

Examples

The following example illustrates using the **-changepassword** command to change the password used to unlock BitLocker on data drive D.

```
manage-bde -changepassword D:
```

Additional references

- [Command-Line Syntax Key](#)
- [Manage-bde](#)

manage-bde: changepin

4/13/2018 • 2 minutes to read • [Edit Online](#)

Modifies the PIN for an operating system drive. The user is prompted to enter a new PIN. For examples of how this command can be used, see [Examples](#).

Syntax

```
manage-bde -changepin [<Drive>] [-computername <Name>] [{-?|/?}] [{-help|-h}]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------|---|
| <Drive> | Represents a drive letter followed by a colon. |
| -computername | Specifies that Manage-bde.exe will be used to modify BitLocker protection on a different computer. You can also use -cn as an abbreviated version of this command. |
| <Name> | Represents the name of the computer on which to modify BitLocker protection. Accepted values include the computer's NetBIOS name and the computer's IP address. |
| -? or /? | Displays brief Help at the command prompt. |
| -help or -h | Displays complete Help at the command prompt. |

Examples

The following example illustrates using the **-changepin** command to change the PIN used with BitLocker on drive C.

```
manage-bde -changepin C:
```

Additional references

- [Command-Line Syntax Key](#)
- [Manage-bde](#)

manage-bde: changekey

4/13/2018 • 2 minutes to read • [Edit Online](#)

Modifies the startup key for an operating system drive. For examples of how this command can be used, see [Examples](#).

Syntax

```
manage-bde -changekey [<Drive>] [<PathToExternalKeyDirectory>] [-computername <Name>] [{-?|/?}] [{-help|-h}]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------------------------|---|
| <Drive> | Represents a drive letter followed by a colon. |
| <PathToExternalKeyDirectory> | Represents the directory location to save the external startup key file that can be used to unlock the drive. |
| -computername | Specifies that Manage-bde.exe will be used to modify BitLocker protection on a different computer. You can also use -cn as an abbreviated version of this command. |
| <Name> | Represents the name of the computer on which to modify BitLocker protection. Accepted values include the computer's NetBIOS name and the computer's IP address. |
| -? or /? | Displays brief Help at the command prompt. |
| -help or -h | Displays complete Help at the command prompt. |

Examples

The following example illustrates using the **-changekey** command to create a new startup key on drive E to use with BitLocker encryption on drive C.

```
manage-bde -changekey C: E:\
```

Additional references

- [Command-Line Syntax Key](#)
- [Manage-bde](#)

manage-bde: KeyPackage

4/13/2018 • 2 minutes to read • [Edit Online](#)

Generates a key package for a drive. The key package can be used in conjunction with the repair tool to repair corrupted drives. For examples of how this command can be used, see [Examples](#).

Syntax

```
manage-bde -KeyPackage [<Drive>] [-ID <KeyProtectoryID>] [-path <PathToExternalKeyDirectory>] [-computername <Name>] [{-?|/?}] [{-help|-h}]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------|---|
| <Drive> | Represents a drive letter followed by a colon. |
| -ID | Create a key package using the key protector with the identifier specified by this ID value. |
| -path | Location in which to save the key package created. |
| -computername | Specifies that Manage-bde.exe will be used to modify BitLocker protection on a different computer. You can also use -cn as an abbreviated version of this command. |
| <Name> | Represents the name of the computer on which to modify BitLocker protection. Accepted values include the computer's NetBIOS name and the computer's IP address. |
| -? or /? | Displays brief Help at the command prompt. |
| -help or -h | Displays complete Help at the command prompt. |

Examples

The following example illustrates using the **-KeyPackage** command to create a key package for drive C based off the key protector identified by the GUID and to save the key package to F:\Folder.

```
manage-bde -KeyPackage C: -id {84E151C1...7A62067A512} -path "f:\Folder"
```

TIP

Use **manage-bde -protectors -get** along with the drive letter that you want to create a key package for to get a list of available GUIDs to use as the ID value.

Additional references

- [Command-Line Syntax Key](#)

- [Manage-bde](#)

manage-bde: upgrade

4/13/2018 • 2 minutes to read • [Edit Online](#)

Upgrades the BitLocker version. For examples of how this command can be used, see [Examples](#).

Syntax

```
manage-bde -upgrade [<Drive>] [-computername <Name>] [{-?|/?}] [{-help|-h}]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------|---|
| <Drive> | Represents a drive letter followed by a colon. |
| -computername | Specifies that Manage-bde.exe will be used to modify BitLocker protection on a different computer. You can also use -cn as an abbreviated version of this command. |
| <Name> | Represents the name of the computer on which to modify BitLocker protection. Accepted values include the computer's NetBIOS name and the computer's IP address. |
| -? or /? | Displays brief Help at the command prompt. |
| -help or -h | Displays complete Help at the command prompt. |

Examples

The following example illustrates using the **-upgrade** command to upgrade BitLocker encryption on drive C.

```
manage-bde -upgrade C:
```

Additional references

- [Command-Line Syntax Key](#)
- [Manage-bde](#)
- [Upgrading a BitLocker-Protected Computer from Windows Vista to Windows 7](#)

manage-bde: WipeFreeSpace

4/13/2018 • 2 minutes to read • [Edit Online](#)

Wipes the free space on the volume removing any data fragments that may have existed in the space. Running this command on a volume that was encrypted using the "Used Space Only" encryption method provides the same level of protection as the "Full Volume Encryption" encryption method. For examples of how this command can be used, see [Examples](#).

Syntax

```
manage-bde -WipeFreeSpace|-w [<Drive>] [-Cancel] [-computername <Name>] [{-?|/?}] [{-help|-h}]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------|---|
| <Drive> | Represents a drive letter followed by a colon, a volume GUID path, or a mounted volume. |
| -Cancel | Cancels a wipe of free space that is in process. |
| -computername | Specifies that Manage-bde.exe will be used to modify BitLocker protection on a different computer. You can also use -cn as an abbreviated version of this command. |
| <Name> | Represents the name of the computer on which to modify BitLocker protection. Accepted values include the computer's NetBIOS name and the computer's IP address. |
| -? or /? | Displays brief Help at the command prompt. |
| -help or -h | Displays complete Help at the command prompt. |

Examples

The following example illustrates using the **-w** command to create wipe the free space on drive C.

```
manage-bde -w C:
```

The following example illustrates using the **-w** command with the **-cancel** parameter to cancel the wipe the free space on drive C.

```
manage-bde -w -Cancel C:
```

Additional references

- [Command-Line Syntax Key](#)
- [Manage-bde](#)

mapadmin

4/13/2018 • 6 minutes to read • [Edit Online](#)

You can use **Mapadmin** to manage User Name Mapping for Microsoft Services for Network File System.

Syntax

```
mapadmin [<computer>] [-u <user> [-p <password>]]
mapadmin [<computer>] [-u <user> [-p <password>]] {start | stop}
mapadmin [<computer>] [-u <user> [-p <password>]] config <option[...]>
mapadmin [<computer>] [-u <user> [-p <password>]] add -wu <WindowsUser> -uu <UNIXUser> [-setprimary]
mapadmin [<computer>] [-u <user> [-p <password>]] add -wg <WindowsGroup> -ug <UNIXGroup> [-setprimary]
mapadmin [<computer>] [-u <user> [-p <password>]] setprimary -wu <WindowsUser> [-uu <UNIXUser>]
mapadmin [<computer>] [-u <user> [-p <password>]] setprimary -wg <WindowsGroup> [-ug <UNIXGroup>]
mapadmin [<computer>] [-u <user> [-p <password>]] delete <option[...]>
mapadmin [<computer>] [-u <user> [-p <password>]] list <option[...]>
mapadmin [<computer>] [-u <user> [-p <password>]] backup <filename>
mapadmin [<computer>] [-u <user> [-p <password>]] restore <filename>
mapadmin [<computer>] [-u <user> [-p <password>]] adddomainmap -d <WindowsDomain> {-y <<NISdomain>> | -f <path>}
mapadmin [<computer>] [-u <user> [-p <password>]] removedomainmap -d <WindowsDomain> -y <<NISdomain>>
mapadmin [<computer>] [-u <user> [-p <password>]] removedomainmap -all
mapadmin [<computer>] [-u <user> [-p <password>]] listdomainmaps
```

Description

The **mapadmin** command-line utility administers User Name Mapping on the local or remote computer running Microsoft Services for Network File System. If you are logged on with an account that does not have administrative credentials, you can specify a user name and password of an account that does.

In addition to specific command arguments, **mapadmin** accepts the following arguments and options:

<computer> Specifies the remote computer running the User Name Mapping service that you want to administer. You can specify the computer using a Windows Internet Name Service (WINS) name or a Domain Name System (DNS) name, or by Internet Protocol (IP) address.

-u <user> Specifies the user name of the user whose credentials are to be used. It might be necessary to add the domain name to the user name in the form *domain\user name*.

-p <password> Specifies the password of the user. If you specify the **-u** option but omit the **-p** option, you are prompted for the user's password. The specific action that **mapadmin** performs depends on the command argument you specify:

Parameters

start

starts the User Name Mapping service.

stop

Stops the User Name Mapping service.

config

Specifies general settings for User Name Mapping. The following options are available with this command

argument: **-r <ddd>:<hh>:<mm>** - Specifies the refresh interval for updating from the Windows and NIS databases in days, hours, and minutes. The minimum interval is 5 minutes. **-i {yes | no}** - Turns simple mapping on (**yes**) or off (**no**). By default, simple mapping is on. **add** - creates a new mapping for a user or group. The following options are available with this command argument:

| OPTION | DEFINITION |
|-------------|---|
| -wu <name> | Specifies the name of the Windows user for which a new mapping is being created. |
| -uu <name> | Specifies the name of the UNIX user for which a new mapping is being created. |
| -wg <group> | Specifies the name of the Windows group for which a new mapping is being created. |
| -ug <group> | Specifies the name of the UNIX group for which a new mapping is being created. |
| -setprimary | Specifies that the new mapping is the primary mapping. |

setprimary - Specifies which mapping is the primary mapping for a UNIX user or group with multiple mappings. The following options are available with this command argument:

| OPTION | DEFINITION |
|-------------|--|
| -wu <name> | Specifies the Windows user of the primary mapping. If more than one mapping for the user exists, use the -uu option to specify the primary mapping. |
| -uu <name> | Specifies the UNIX user of the primary mapping. |
| -wg <group> | Specifies the Windows group of the primary mapping. If more than one mapping for the group exists, use the -ug option to specify the primary mapping. |
| -ug <group> | Specifies the UNIX group of the primary mapping. |

delete - removes the mapping for a user or group. The following options are available for this command argument:

| OPTION | DEFINITION |
|-------------|--|
| -wu <user> | The Windows user for which the mapping will be deleted, specified as <WindowsDomain>\<User Name>. You must specify either the -wu or the -uu option, or both. If you specify both options, the particular mapping identified by the two options will be deleted. If you specify only the -wu option, all mappings for the specified user will be deleted. |
| -wg <group> | The Windows group for which the mapping will be deleted, specified as <WindowsDomain>\<groupname>. You must specify either the -wg or the -ug option, or both. If you specify both options, the particular mapping identified by the two options will be deleted. If you specify only the -wg option, all mappings for the specified group will be deleted. |

| OPTION | DEFINITION |
|-------------|---|
| -uu <user> | The UNIX user for whom the mapping will be deleted, specified as <User Name>. You must specify either the -wu or the -uu option, or both. If you specify both options, the particular mapping identified by the two options will be deleted. If you specify only the -uu option, all mappings for the specified user will be deleted. |
| -ug <group> | The UNIX group for which the mapping will be deleted, specified as <groupname>. You must specify either the -wg or the -ug option, or both. If you specify both options, the particular mapping identified by the two options will be deleted. If you specify only the -ug option, all mappings for the specified group will be deleted. |

list - Displays information about user and group mappings. The following options are available with this command argument:

| OPTION | DEFINITION |
|-------------|---|
| -all | lists both simple and advanced mappings for users and groups. |
| -simple | lists all simple mapped users and groups. |
| -advanced | lists all advanced mapped users and groups. Maps are listed in the order in which they are evaluated. Primary maps, marked with an asterisk (*), are listed first, followed by secondary maps, which are marked with a carat (^). |
| -wu <name> | lists the mapping for a specified Windows user. |
| -wg <group> | lists the mapping for a Windows group. |
| -uu <name> | lists the mapping for a UNIX user. |
| -ug <group> | lists the mapping for a UNIX group. |

backup - Saves User Name Mapping configuration and mapping data to the file specified by <filename>. **restore** - replaces configuration and mapping data with data from the file (specified by <filename>) that was created using the **backup** command argument. **adddomainmap** - adds a simple map between a Windows domain and an NIS domain or password and group files. The following options are available for this command argument:

| OPTION | DEFINITION |
|--------------------|---|
| -d <WindowsDomain> | Specifies the Windows domain to be mapped. |
| -y <NISdomain> | Specifies the NIS domain to be mapped. -n <nisServer> Specifies the NIS server for the NIS domain specified with the -y option. |

| OPTION | DEFINITION |
|-----------|---|
| -f <path> | Specifies the fully qualified path of directory containing the password and group files to be mapped. The files must be located on the computer being managed, and you cannot use mapadmin to manage a remote computer to set up maps based on password and group files. |

removedomainmap - removes a simple map between a Windows domain and an NIS domain. The following options and argument are available for this command argument:

| OPTION | DEFINITION |
|--------------------|---|
| -d <WindowsDomain> | Specifies the Windows domain of the map to be removed. |
| -y <NISdomain> | Specifies the NIS domain of the map to be removed. |
| -all | Specifies that all simple maps between Windows and NIS domains are to be removed. This will also remove any simple map between a Windows domain and password and group files. |

listdomainmaps - lists the Windows domains that are mapped to NIS domains or password and group files.

Notes

- if you do not specify a command argument, **mapadmin** displays the current settings for User Name Mapping.
- for all options that specify a user or group name, the following formats can be used:
- for Windows users, use the form <domain>\<user name>, \\<computer>\<user name>, \<computer>\<user name>, or <computer>\<user name>
- for Windows groups, use the form <domain>\<<groupname>>, \\<computer>\<<groupname>>, \<computer>\<<groupname>>, or <computer>\<<groupname>>
- for UNIX users, use the form <NISdomain>\<user name>, <user name>@<NISdomain>, user <name>@PCNFS, or PCNFS\<user name>
- for UNIX groups, use the form <NISdomain>\<groupname>, <groupname>@<NISdomain>, <groupname>@PCNFS, or PCNFS\<groupname>

additional references

[Command-Line Syntax Key](#)

Md

4/13/2018 • 2 minutes to read • [Edit Online](#)

Creates a directory or subdirectory.

NOTE

This command is the same as the **mkdir** command.

For examples of how to use this command, see [Examples](#).

Syntax

```
md [<Drive>:]<Path>  
mkdir [<Drive>:]<Path>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| <Drive>: | Specifies the drive on which you want to create the new directory. |
| <Path> | Required. Specifies the name and location of the new directory. The maximum length of any single path is determined by the file system. |
| /? | Displays help at the command prompt. |

Remarks

Command extensions, which are enabled by default, allow you to use a single **md** command to create intermediate directories in a specified path.

Examples

To create a directory named Directory1 within the current directory, type:

```
md Directory1
```

To create the directory tree Taxes\Property\Current within the root directory, with command extensions enabled, type:

```
md \Taxes\Property\Current
```

To create the directory tree Taxes\Property\Current within the root directory as in the previous example, but with command extensions disabled, type the following sequence of commands:

```
md \Taxes  
cd \Taxes  
md Property  
cd Property  
md Current
```

Additional references

[Command-Line Syntax Key](#)

[Cmd](#)

mkdir

4/13/2018 • 2 minutes to read • [Edit Online](#)

This command is the same as the **md** command. See [Md](#) for syntax and parameters.

mklink

10/17/2017 • 2 minutes to read • [Edit Online](#)

Creates a symbolic link.

For examples of how to use this command, see [Examples](#).

Syntax

```
mklink [[/d] | [/h] | [/j]] <Link> <Target>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| /d | Creates a directory symbolic link. By default, mklink creates a file symbolic link. |
| /h | Creates a hard link instead of a symbolic link. |
| /j | Creates a Directory Junction. |
| <Link> | Specifies the name of the symbolic link that is being created. |
| <Target> | Specifies the path (relative or absolute) that the new symbolic link refers to. |
| /? | Displays help at the command prompt. |

Examples

To create a symbolic link named MyDocs from the root directory to the \Users\User1\Documents directory, type:

```
mklink /d \MyDocs \Users\User1\Documents
```

mmc

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Using mmc command-line options, you can open a specific **mmc** console, open **mmc** in author mode, or specify that the 32-bit or 64-bit version of **mmc** is opened.

Syntax

```
mmc <path>\<filename>.msc [/a] [/64] [/32]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| \.msc | starts mmc and opens a saved console. You need to specify the complete path and file name for the saved console file. If you do not specify a console file, mmc opens a new console. |
| /a | Opens a saved console in author mode. Used to make changes to saved consoles. |
| /64 | Opens the 64-bit version of mmc (mmc64). Use this option only if you are running a Microsoft 64-bit operating system and want to use a 64-bit snap-in. |
| /32 | Opens the 32-bit version of mmc (mmc32). When running a Microsoft 64-bit operating system, you can run 32-bit snap-ins by opening mmc with this command-line option when you have 32-bit only snap-ins. |
| /? | Displays help at the command prompt. |

Remarks

- Using the **\.msc** command-line option You can use environment variables to create command lines or shortcuts that do not depend on the explicit location of console files. For instance, if the path to a console file is in the system folder (for example, **mmc c:\winnt\system32\console_name.msc**), you can use the expandable data string **%systemroot%** to specify the location (**mmc%systemroot%\system32\console_name.msc**). This may be useful if you are delegating tasks to people in your organization who are working on different computers.
- Using the **/a** command-line option When consoles are opened with this option, they are opened in author mode, regardless of their default mode. This does not permanently change the default mode setting for files; when you omit this option, mmc opens console files according to their default mode settings.
- After you open **mmc** or a console file in author mode, you can open any existing console by clicking **Open** on the **Console** menu.
- You can use the command line to create shortcuts for opening **mmc** and saved consoles. A command-line command works with the **Run** command on the **start** menu, in any command-prompt window, in shortcuts, or

in any batch file or program that calls the command. ## additional references

- [Command-Line Syntax Key](#)

mode

4/13/2018 • 5 minutes to read • [Edit Online](#)

Displays system status, changes system settings, or reconfigures ports or devices. If used without parameters, **mode** displays all the controllable attributes of the console and the available COM devices.

You can use **mode** to perform the following tasks—each task uses a different syntax:

- [To configure a serial communications port](#)
- [To display the status of all devices or of a single device](#)
- [To redirect output from a parallel port to a serial communications port](#)
- [To select, refresh, or display the numbers of the code pages for the console](#)
- [To change the size of the command prompt screen buffer](#)
- [To set the keyboard typematic rate](#)

To configure a serial communications port

Syntax

```
mode com<M>[:] [baud=<B>] [parity=<P>] [data=<D>] [stop=<S>] [to={on|off}] [xon={on|off}] [odsr={on|off}]  
[octs={on|off}] [dtr={on|off|hs}] [rts={on|off|hs|tg}] [idsr={on|off}]
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------|--|
| Com<M>[:] | Specifies the number of the async Prncnfg.vbshronous communications port. |
| baud= | <p>Specifies the transmission rate in bits per second. The following table lists valid abbreviations for <i>B</i> and their related rates.</p> <ul style="list-style-type: none">- 11 = 110 baud- 15 = 150 baud- 30 = 300 baud- 60 = 600 baud- 12 = 1200 baud- 24 = 2400 baud- 48 = 4800 baud- 96 = 9600 baud- 19 = 19,200 baud |
| parity= <P> | <p>Specifies how the system uses the parity bit to check for transmission errors. The following table lists valid values for <i>P</i>. The default value is e. Not all computers support the values m and s.</p> <ul style="list-style-type: none">- n = none- e = even- o = odd- m = mark- s = space |

| PARAMETER | DESCRIPTION |
|-----------|--|
| data= <D> | Specifies the number of data bits in a character. Valid values for d are in the range 5 through 8. The default value is 7. Not all computers support the values 5 and 6. |
| stop= <S> | Specifies the number of stop bits that define the end of a character: 1, 1.5, or 2. If the baud rate is 110, the default value is 2. Otherwise, the default value is 1. Not all computers support the value 1.5. |
| to={on | off} |
| xon={on | off} |
| odsr={on | off} |
| octs={on | off} |
| dtr={on | off |
| rts={on | off |
| idsr={on | off} |
| /? | Displays help at the command prompt. |

To display the status of all devices or of a single device

Syntax

```
mode [<Device>] [/status]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| <Device> | Specifies the name of the device for which you want to display the status. |
| /status | Requests the status of any redirected parallel printers. You can abbreviate the /status command-line option as /sta . |
| /? | Displays help at the command prompt. |

Remarks

If used without parameters, **mode** displays the status of all devices that are installed on your system.

To redirect output from a parallel port to a serial communications port

Syntax

```
mode lpt<N>[:]=com<M>[:]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| lpt<N>[:] | Required. Specifies the parallel port. Valid values for <i>N</i> are in the range 1 through 3. |
| com<M>[:] | Required. Specifies the serial port. Valid values for <i>M</i> are in the range 1 through 4. |
| /? | Displays help at the command prompt. |

Remarks

You must be a member of the Administrators group to redirect printing.

Examples

To set up your system so that it sends parallel printer output to a serial printer, you must use the **mode** command twice. The first time, use **mode** to configure the serial port. The second time, use **mode** to redirect parallel printer output to the serial port you specified in the first **mode** command.

For example, if your serial printer operates at 4800 baud with even parity, and it is connected to the COM1 port (the first serial connection on your computer), type:

```
mode com1 48,e,,b
mode lpt1=com1
```

If you redirect parallel printer output from LPT1 to COM1, but then you decide that you want to print a file by using LPT1, type the following command before you print the file:

```
mode lpt1
```

This command prevents the redirection the file from LPT1 to COM1.

To select, refresh, or display the numbers of the code pages for the console

Syntax

```
mode <Device> codepage select=<YYY>
mode <Device> codepage [/status]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------------|--|
| <Device> | Required. Specifies the device for which you want to select a code page. CON is the only valid name for a device. |
| codepage select= | Required. Specifies which code page to use with the specified device. You can abbreviate codepage select as cp sel . |

| PARAMETER | DESCRIPTION |
|-----------|---|
| <YYY> | Required. Specifies the number of the code page to select. The following list shows each code page that is supported and its country/region or language. 437: United States 850: Multilingual (Latin I) 852: Slavic (Latin II) 855: Cyrillic (Russian) 857: Turkish 860: Portuguese 861: Icelandic 863: Canadian-French 865: Nordic 866: Russian 869: Modern Greek |
| codepage | Required. Displays the numbers of the code pages (if any) that are selected for the specified device. |
| /status | Displays the numbers of the current code pages selected for the specified device. You can abbreviate /status to /sta . Whether or not you specify /status, mode codepage displays the numbers of the code pages that are selected for the specified device. |
| /? | Displays help at the command prompt. |

To change the size of the command prompt screen buffer

Syntax

```
mode con[:] [cols=<C>] [lines=<N>]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|---|
| con[:] | Required. Indicates that the change applies to the Command Prompt window. |
| cols= <C> | Specifies the number of columns in the command prompt screen buffer. |
| lines= <N> | Specifies the number of lines in the command prompt screen buffer. |
| /? | Displays help at the command prompt. |

To set the keyboard typematic rate

Syntax

```
mode con[:] [rate=<R> delay=<D>]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|--|
| con[:] | Required. Refers to the keyboard. |
| rate= <R> | Specifies the rate at which a character is repeated on the screen when you hold down a key. |
| delay= <D> | Specifies the amount of time that will elapse after you press and hold down a key before the character output repeats. |
| /? | Displays help at the command prompt. |

Remarks

- The typematic rate is the rate at which a character repeats when you hold down the key for that character. The typematic rate has two components, the rate and the delay. Some keyboards do not recognize this command.
- Using **rate**=*R*

Valid values are in the range 1 through 32. These values are equal to approximately 2 to 30 characters per second. The default value is 20 for IBM AT-compatible keyboards, and 21 for IBM PS/2-compatible keyboards. If you set the rate, you must also set the delay.

- Using **delay**=*D*

Valid values for *D* are 1, 2, 3, and 4 (representing 0.25, 0.50, 0.75, and 1 second). The default value is 2. If you set the delay, you must also set the rate.

Additional references

[Command-Line Syntax Key](#)

more

4/13/2018 • 2 minutes to read • [Edit Online](#)

Displays one screen of output at a time.

For examples of how to use this command, see [Examples](#).

Syntax

```
<Command> | more [/c] [/p] [/s] [/t<N>] [+<N>]
more [[/c] [/p] [/s] [/t<N>] [+<N>]] < [<Drive>:][<Path><FileName>
more [/c] [/p] [/s] [/t<N>] [+<N>] [<Files>]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------|---|
| <Command> | Specifies a command for which you want to display the output. |
| /c | Clears the screen before displaying a page. |
| /p | Expands form-feed characters. |
| /s | Displays multiple blank lines as a single blank line. |
| /t<N> | Displays tabs as the number of spaces specified by <i>N</i> . |
| + <N> | Displays the first file beginning at the line specified by <i>N</i> . |
| [<Drive>:] [] | Specifies the location and name of a file to display. |
| <Files> | Specifies a list of files to display. Separate file names with a space. |
| /? | Displays help at the command prompt. |

Remarks

- The following subcommands are accepted at the **more** prompt (`-- More --`).
|Key|Action| |---|-----| |SPACEBAR|Displays the next page.| |ENTER|Displays the next line.| |f|Displays the next file.| |q|Quits the **more** command.| |=|Shows the line number.| |p <N>|Displays the next *N* lines.| |s <N>|Skips the next *N* lines.| |?|Shows the commands that are available at the **more** prompt.|
- When using the redirection character (<), you must specify a file name as the source. When using the pipe (|), you can use such commands as **dir**, **sort**, and **type**.
- The **more** command, with different parameters, is available from the Recovery Console.

Examples

To view the first screen of information of a file named `Clients.new`, type one of the following commands:

```
more < clients.new  
type clients.new | more
```

The **more** command displays the first screen of information from `Clients.new`, and then displays the following prompt:

```
-- More --
```

You can then press the SPACEBAR to see the next screen of information.

To clear the screen and remove all extra blank lines before displaying the file `Clients.new`, type one of the following commands:

```
more /c /s < clients.new  
type clients.new | more /c /s
```

The **more** command displays the first screen of information from `Clients.new`, and then displays the following prompt:

```
-- More --
```

Using more subcommands

The following examples can be used at the **more** prompt (`-- More --`).

- To display the file one line at a time, press ENTER at the **more** prompt.
- To display the next screen, press the SPACEBAR at the **more** prompt.
- To display the next file listed on the command line, type **f** at the **more** prompt.
- To show the available commands, type **?** at the **more** prompt.
- To quit **more**, type **q** at the **more** prompt.
- To display the current line number, type **=** at the **more** prompt. The current line number is added to the **more** prompt as follows:

```
-- More [Line: 24] --
```

- To display a specific number of lines, type **p** at the **more** prompt. **More** prompts you for the number of lines to display as follows:

```
-- More -- Lines:
```

Type the number of lines to display, and then press ENTER. **More** displays the specified number of lines.

- To skip a specific number of lines, type **s** at the **more** prompt. **More** prompts you for the number of lines to skip as follows:

```
-- More -- Lines:
```

Type the number of lines to skip, and then press ENTER. **More** skips the specified number of lines and displays the next screen of information.

Additional references

[Command-Line Syntax Key](#)

mount

5/14/2018 • 2 minutes to read • [Edit Online](#)

You can use **mount** to mount Network File System (NFS) network shares.

Syntax

```
mount [-o <Option>[...]] [-u:<UserName>] [-p:{<Password> | *}] {\\<ComputerName>\<ShareName> |  
<ComputerName>:<ShareName>} {<DeviceName> | *}
```

Description

The **mount** command-line utility mounts the file system identified by *ShareName* exported by the NFS server identified by *ComputerName* and associates it with the drive letter specified by *DeviceName* or, if an asterisk (*) is used, by the first available driver letter. Users can then access the exported file system as though it were a drive on the local computer. When used without options or arguments, **mount** displays information about all mounted NFS file systems.

The **mount** utility is available only if Client for NFS is installed.

The following options and arguments can be used with the **mount** utility.

| TERM | DEFINITION |
|------------------------|--|
| -o rsize= <buffersize> | Sets the size in kilobytes of the read buffer. Acceptable values are 1, 2, 4, 8, 16, and 32; the default is 32 KB. |
| -o wsize= <buffersize> | Sets the size in kilobytes of the write buffer. Acceptable values are 1, 2, 4, 8, 16, and 32; the default is 32 KB. |
| -o timeout= <seconds> | Sets the time-out value in seconds for a remote procedure call (RPC). Acceptable values are 0.8, 0.9, and any integer in the range 1-60; the default is 0.8. |
| -o retry= <number> | Sets the number of retries for a soft mount. Acceptable values are integers in the range 1-10; the default is 1. |
| -o mtype={soft | hard} |
| -o anon | Mounts as an anonymous user. |
| -o nolock | Disables locking (default is enabled). |
| -o casesensitive | Forces file lookups on the server to be case sensitive. |

| TERM | DEFINITION |
|-----------------------|---|
| -o fileaccess= <mode> | <p>Specifies the default permission mode of new files created on the NFS share. Specify <i>mode</i> as a three-digit number in the form <i>ogw</i>, where <i>o</i>, <i>g</i>, and <i>w</i> are each a digit representing the access granted the file's owner, group, and the world, respectively. The digits must be in the range 0-7 with the following meaning:</p> <ul style="list-style-type: none"> - 0: No access - 1: x (execute access) - 2: w (write access) - 3: wx - 4: r (read access) - 5: rx - 6: rw - 7: rwx |
| -o lang={euc-jp | euc-tw |
| -u:<UserName> | <p>Specifies the user name to use for mounting the share. If <i>username</i> is not preceded by a backslash (\), it is treated as a UNIX user name.</p> |
| -p:<Password> | <p>The password to use for mounting the share. If you use an asterisk (*), you will be prompted for the password.</p> |

NOTE

mountvol

4/13/2018 • 2 minutes to read • [Edit Online](#)

Creates, deletes, or lists a volume mount point.

For examples of how to use this command see [Examples](#).

Syntax

```
mountvol [<Drive>:]<Path VolumeName>
mountvol [<Drive>:]<Path> /d
mountvol [<Drive>:]<Path> /l
mountvol [<Drive>:]<Path> /p
mountvol /r
mountvol [/n | /e]
mountvol <Drive>: /s
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------|---|
| [<Drive>:] | Specifies the existing NTFS directory where the mount point will reside. |
| <VolumeName> | Specifies the volume name that is the target of the mount point. The volume name uses the following syntax, where <i>GUID</i> is a globally unique identifier: <code>\\\\\\?\\Volume\\{GUID}\\</code> The brackets { } are required. |
| /d | Removes the volume mount point from the specified folder. |
| /l | Lists the mounted volume name for the specified folder. |
| /p | Removes the volume mount point from the specified directory, dismounts the basic volume, and takes the basic volume offline, making it unmountable. If other processes are using the volume, mountvol closes any open handles before dismounting the volume. |
| /r | Removes volume mount point directories and registry settings for volumes that are no longer in the system, preventing them from being automatically mounted and given their former volume mount point(s) when added back to the system. |
| /n | Disables automatic mounting of new basic volumes. New volumes are not mounted automatically when added to the system. |
| /e | Re-enables automatic mounting of new basic volumes. |

| PARAMETER | DESCRIPTION |
|-----------|--|
| /s | Mounts the EFI system partition on the specified drive. Available on Itanium-based computers only. |
| /? | Displays help at the command prompt. |

Remarks

- **Mountvol** allows you to link volumes without requiring a drive letter.
- Volumes that are dismounted by using **/p** are listed in the volumes list as "NOT MOUNTED UNTIL A VOLUME MOUNT POINT IS CREATED." If the volume has more than one mount point, use **/d** to remove the additional mount points before using **/p**. You can make the basic volume mountable again by assigning a volume mount point.
- If you need to expand your volume space without reformatting or replacing a hard drive, you can add a mount path to another volume. The benefit of using one volume with several mount paths is that you can access all local volumes by using a single drive letter (such as `C:`). You do not need to remember which volume corresponds to which drive letter—although you can still mount local volumes and assign them drive letters.

Examples

To create a mount point, type:

```
mountvol \sysmount \\?\Volume\{2eca078d-5cbc-43d3-aff8-7e8511f60d0e}\
```

Additional references

[Command-Line Syntax Key](#)

move

4/13/2018 • 2 minutes to read • [Edit Online](#)

Moves one or more files from one directory to another directory.

For examples of how to use this command, see [Examples](#).

Syntax

```
move [{/y | /-y}] [<Source>] [<Target>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| /y | Suppresses prompting to confirm that you want to overwrite an existing destination file. |
| /-y | Causes prompting to confirm that you want to overwrite an existing destination file. |
| <Source> | Specifies the path and name of the file or files to move. If you want to move or rename a directory, <i>Source</i> should be the current directory path and name. |
| <Target> | Specifies the path and name to move files to. If you want to move or rename a directory, <i>Target</i> should be the desired directory path and name. |
| /? | Displays help at the command prompt. |

Remarks

- The **/y** command-line option might be preset in the COPYCMD environment variable. You can override this with **/-y** on the command line. The default is to prompt before overwriting files unless the **copy** command is run from within a batch script.
- Moving encrypted files to a volume that does not support Encrypting File System (EFS) results in an error. Decrypt the files first or move the files to a volume that supports EFS.

Examples

To move all files with the .xls extension from the \Data directory to the \Second_Q\Reports directory, type:

```
move \data\*.xls \second_q\reports\
```

Additional references

[Command-Line Syntax Key](#)

mqbkup

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Backs up MSMQ message files and registry settings to a storage device and restores previously-stored messages and settings.

Both the backup and the restore operation will stop the local MSMQ service. If the MSMQ service was started beforehand, the utility will attempt to restart the MSMQ service at the end of the backup or the restore operation. If the service was already stopped before running the utility, no attempt to restart the service is made.

Before using the MSMQ Message Backup/Restore utility you must close all local applications that are using MSMQ.

Syntax

```
mqbkup {/b | /r} <folder path_to_storage_device>
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------------------------|--|
| /b | Specifies backup operation |
| /r | Specifies restore operation |
| <folder path_to_storage_device> | Specifies the path where the MSMQ message files and registry settings are stored |
| /? | Displays help at the command prompt. |

Examples

To backup all MSMQ message files and registry settings and store them in the *Msmqbkup* folder on your C: drive.

```
mqbkup /b c:\msmqbkup
```

if the specified folder does not exist, the utility will automatically create one. If you choose to specify an existing folder, this folder must be empty. If you specify a non-empty folder, the utility will delete every file and subfolder contained within it. In this case, you will be prompted to give permission to delete existing files and subfolders. You can use the **/y** parameter to indicate that you agree beforehand to the deletion of all existing files and subfolders in the specified folder.

To delete all files and subfolders in the *Oldbkup* folder on your C: drive and store MSMQ message files and registry settings in this folder.

```
mqbkup /b /y c:\oldbkup
```

To restore MSMQ messages and registry settings:

```
mqbkup /r c:\msmqbkup
```

The locations of folders used to store MSMQ message files are stored in the registry. Thus, the utility will restore MSMQ message files to the folders specified in the registry and not to the storage folders used before the restore operation. If the folders specified in the registry do not exist, the restore operation will automatically create them. If folders directories do exist and are not empty, the utility will prompt you for permission to delete the current contents of these folders.

additional references

- [Command-Line Syntax Key](#)

mqsvc

4/13/2018 • 2 minutes to read • [Edit Online](#)

Message Queuing technology enables applications running at different times to communicate across heterogeneous networks and systems that may be temporarily offline. Message Queuing provides guaranteed message delivery, efficient routing, security, and priority-based messaging. It can be used to implement solutions for both asynchronous and synchronous messaging scenarios. For more information about this command, see [Message Queuing \(MSMQ\)](#) on MSDN.

Syntax

```
Mqsvc.exe
```

Parameters

None

Additional references

- [Command-Line Syntax Key](#)

mqtgsvc

4/13/2018 • 2 minutes to read • [Edit Online](#)

Monitors a queue for incoming messages and performs an action, in the form of an executable file or COM component, when the rules of a trigger are evaluated as true. For examples of how the Message Queuing Triggers service can be used, see [Message Queuing Triggers](#) on MSDN.

Syntax

```
Mqtgsvc.exe
```

Parameters

None

Additional references

- [Command-Line Syntax Key](#)

msdt

4/13/2018 • 2 minutes to read • [Edit Online](#)

Invokes a troubleshooting pack at the command line or as part of an automated script, and enables additional options without user input.

Syntax

```
msdt </id <name> | /path <name> | /cab < name>> <</parameter> [options] ... <parameter> [options]>>
```

Parameters

The following table includes the parameters and options supported by msdt.exe.

| PARAMETER | DESCRIPTION |
|--------------------|--|
| /id <package name> | Specifies which diagnostic package to run. For a list of available packages, see the Troubleshooting Pack ID in the "Available troubleshooting packs" section later in this topic. |
| /path <directory> | .diagpkg file |
| /dci <passkey> | Prepopulates the passkey field in msdt. This parameter is only used when a support provider has supplied a passkey. |
| /dt <directory> | Displays the troubleshooting history in the specified directory. Diagnostic results are stored in the user's %LOCALAPPDATA%\Diagnostics or %LOCALAPPDATA%\ElevatedDiagnostics directories. |
| /af <answer file> | Specifies an answer file in XML format that contains responses to one or more diagnostic interactions. |

msg

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Sends a message to a user on a Remote Desktop Session Host (rd Session Host) server. for examples of how to use this command, see [Examples](#).

NOTE

In Windows Server 2008 R2, Terminal Services was renamed Remote Desktop Services. To find out what's new in the latest version, see [What's New in Remote Desktop Services in Windows Server 2012](#) in the Windows Server TechNet Library.

Syntax

```
msg {<UserName> | <SessionName> | <SessionID> | @<FileName> | *} [/server:<ServerName>] [/time:<Seconds>] [/v] [/w] [<Message>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| | Specifies the name of the user that you want to receive the message. |
| | Specifies the name of the session that you want to receive the message. |
| | Specifies the numeric ID of the session whose user you want to receive a message. |
| @ | Identifies a file containing a list of user names, session names, and session IDs that you want to receive the message. |
| * | Sends the message to all user names on the system. |
| /server: | Specifies the rd Session Host server whose session or user you want to receive the message. If unspecified, /server uses the server to which you are currently logged on. |
| /time: | Specifies the amount of time that the message you sent is displayed on the user's screen. After the time limit is reached, the message disappears. If no time limit is set, the message remains on the user's screen until the user sees the message and clicks OK . |
| /v | Displays information about the actions being performed. |

| PARAMETER | DESCRIPTION |
|-----------|---|
| /w | Waits for an acknowledgment from the user that the message has been received. Use this parameter with /time: <Seconds> to avoid a possible long delay if the user does not immediately respond. Using this parameter with /v is also helpful. |
| | Specifies the text of the message that you want to send. If no message is specified, you will be prompted to enter a message. To send a message that is contained in a file, type the less than (<) symbol followed by the file name. |
| /? | Displays help at the command prompt. |

Remarks

- if you do not specify a user or a session, **msg** displays an error message. When specifying a session, it must be an active one.
- The user must have Message special access permission to send a message.

Examples

- To send the message entitled "Let's meet at 1PM today" to all sessions for User1, type:

```
msg User1 Let's meet at 1PM today
```

- To send the same message to session modeM02, type: `msg modem02 Let's meet at 1PM today`

- To send the message to session 12, type: `msg 12 Let's meet at 1PM today`

- To send the message to all sessions contained in the file USERlist, type: `msg @userlist Let's meet at 1PM today`

- To send the message to all users who are logged on, type: `msg * Let's meet at 1PM today`

- To send the message to all users, with an acknowledgment time-out (for example, 10 seconds), type:

```
msg * /time:10 Let's meet at 1PM today
```

additional references

- [Command-Line Syntax Key](#)
- [Remote Desktop Services \(Terminal Services\) Command Reference](#)

msiexec

4/13/2018 • 2 minutes to read • [Edit Online](#)

Msiexec

Provides the means to install, modify, and perform operations on Windows Installer from the command line.

For the syntax and examples of how to use this command, see [Msiexec](#).

msinfo32

11/6/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Opens the System Information tool to display a comprehensive view of the hardware, system components, and software environment on the local computer.

Syntax

```
msinfo32 [/pch] [/nfo <path>] [/report <path>] [/computer <computerName>] [/showcategories] [/category <CategoryID>] [/categories {+<CategoryID>(<CategoryID>)+all(-<CategoryID>)}]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------|--|
| | <p>Specifies the file to be opened in the format <i>C:\Folder1\File1.XXX</i>, where <i>C</i> is the drive letter, <i>Folder1</i> is the folder, <i>File1</i> is the file name, and <i>XXX</i> is the file name extension.</p> <p>This file can be an .nfo, .xml, .txt, or .cab file.</p> |
| | <p>Specifies the name of the target or local computer. This can be a UNC name, an IP address, or a full computer name.</p> |
| | <p>Specifies the ID of the category item. You can obtain the category ID by using /showcategories.</p> |
| /pch | <p>Displays the System History view in the System Information tool.</p> |
| /nfo | <p>Saves the exported file as an .nfo file. If the file name that is specified in <i>path</i> does not end in an .nfo extension, the .nfo extension is automatically appended to the file name.</p> |
| /report | <p>Saves the file in <i>path</i> as a text file. The file name is saved exactly as it appears in <i>path</i>. The .txt extension is not appended to the file unless it is specified in <i>path</i>.</p> |
| /computer | <p>starts the System Information tool for the specified remote computer. You must have the appropriate permissions to access the remote computer.</p> |
| /showcategories | <p>starts the System Information tool with all available category IDs displayed, rather than displaying the friendly or localized names. For example, the Software Environment category is displayed as the SWEnv category.</p> |

| PARAMETER | DESCRIPTION |
|-------------|---|
| /category | starts System Information with the specified category selected. Use /showcategories to display a list of available category IDs. |
| /categories | starts System Information with only the specified category or categories displayed. It also limits the output to the selected category or categories. Use /showcategories to display a list of available category IDs. |
| /? | Displays help at the command prompt. |

Remarks

Some System Information categories contain large amounts of data. You can use the **start /wait** command to optimize reporting performance for these categories. For more information, see [System Information](#).

Examples

To list the available category IDs, type:

```
msinfo32 /showcategories
```

To start the System Information tool with all available information displayed, except Loaded Modules, type:

```
msinfo32 /categories +all -loadedmodules
```

To display only System Summary information and create an .nfo file called syssum.nfo that contains information in the System Summary category, type:

```
msinfo32 /nfo syssum.nfo /categories +systemsummary
```

To display resource conflict information and create an .nfo file called conflicts.nfo that contains information about resource conflicts, type:

```
msinfo32 /nfo conflicts.nfo /categories +componentsproblemdevices+resourcesconflicts+resourcesforcedhardware
```

additional references

- [Command-Line Syntax Key](#)

mstsc

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

creates connections to Remote Desktop Session Host (rd Session Host) servers or other remote computers, edits an existing Remote Desktop Connection (.rdp) configuration file, and migrates legacy connection files that were created with Client Connection Manager to new .rdp connection files. for examples of how to use this command, see [Examples](#).

NOTE

In Windows Server 2008 R2, Terminal Services was renamed Remote Desktop Services. To find out what's new in the latest version, see [What's New in Remote Desktop Services in Windows Server 2012](#) in the Windows Server TechNet Library.

Syntax

```
mstsc.exe [<Connection File>] [/v:<Server>[:<Port>]] [/admin] [/f] [/w:<Width> /h:<Height>] [/public] [/span]
mstsc.exe /edit <Connection File>
mstsc.exe /migrate
```

Parameters

| PARAMETER | DESCRIPTION |
|----------------|---|
| | Specifies the name of an .rdp file for the connection. |
| /v:<Server>[:] | Specifies the remote computer and, optionally, the port number to which you want to connect. |
| /admin | Connects you to a session for administering the server. |
| /f | starts Remote Desktop Connection in full-screen mode. |
| /w: | Specifies the width of the Remote Desktop window. |
| /h: | Specifies the height of the Remote Desktop window. |
| /public | Runs Remote Desktop in public mode. In public mode, passwords and bitmaps are not cached. |
| /span | Matches the Remote Desktop width and height with the local virtual desktop, spanning across multiple monitors if necessary. |
| /edit | Opens the specified .rdp file for editing. |

| PARAMETER | DESCRIPTION |
|-----------|---|
| /migrate | Migrates legacy connection files that were created with Client Connection Manager to new .rdp connection files. |
| /? | Displays help at the command prompt. |

Remarks

- Default.rdp is stored for each user as a hidden file in the user's Documents folder. User created .rdp files are saved by default in the user's Documents folder but can be saved anywhere.
- To span across monitors, the monitors must use the same resolution and must be aligned horizontally (that is, side by side). There is currently no support for spanning multiple monitors vertically on the client system.

Examples

- To connect to a session in full-screen mode, type: `mstsc /f`
- To open a file called filename.rdp for editing, type: `mstsc /edit filename.rdp`

additional references

- [Command-Line Syntax Key](#)
- [Remote Desktop Services \(Terminal Services\) Command Reference](#)

nbtstat

10/18/2018 • 4 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Displays NetBIOS over TCP/IP (NetBT) protocol statistics, NetBIOS name tables for both the local computer and remote computers, and the NetBIOS name cache. **nbtstat** allows a refresh of the NetBIOS name cache and the names registered with Windows Internet Name Service (WINS). Used without parameters, **nbtstat** displays help.

Syntax

```
nbtstat [/a <remoteName>] [/A <IPaddress>] [/c] [/n] [/r] [/R] [/RR] [/s] [/S] [<Interval>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| /a | Displays the NetBIOS name table of a remote computer, where <i>remoteName</i> is the NetBIOS computer name of the remote computer. The NetBIOS name table is the list of NetBIOS names that corresponds to NetBIOS applications running on that computer. |
| /A | Displays the NetBIOS name table of a remote computer, specified by the IP address (in dotted decimal notation) of the remote computer. |
| /c | Displays the contents of the NetBIOS name cache, the table of NetBIOS names and their resolved IP addresses. |
| /n | Displays the NetBIOS name table of the local computer. The status of registered indicates that the name is registered either by broadcast or with a WINS server. |
| /r | Displays NetBIOS name resolution statistics. On a computer running Windows XP or Windows Server 2003 that is configured to use WINS, this parameter returns the number of names that have been resolved and registered using broadcast and WINS. |
| /R | Purges the contents of the NetBIOS name cache and then reloads the #PRE-tagged entries from the Lmhosts file. |
| /RR | Releases and then refreshes NetBIOS names for the local computer that is registered with WINS servers. |
| /s | Displays NetBIOS client and server sessions, attempting to convert the destination IP address to a name. |

| PARAMETER | DESCRIPTION |
|-----------|---|
| /S | Displays NetBIOS client and server sessions, listing the remote computers by destination IP address only. |
| | Redisplays selected statistics, pausing the number of seconds specified in <i>Interval</i> between each display. Press CTRL+C to stop redisplaying statistics. If this parameter is omitted, nbtstat prints the current configuration information only once. |
| /? | Displays help at the command prompt. |

Remarks

- **nbtstat** command-line parameters are case-sensitive.
- The following table describes the column headings that are generated by **nbtstat**:

| HEADING | DESCRIPTION |
|-------------|---|
| Input | The number of bytes received. |
| Output | The number of bytes sent. |
| In/Out | Whether the connection is from the computer (outbound) or from another computer to the local computer (inbound). |
| Life | The remaining time that a name table cache entry will live before it is purged. |
| Local Name | The local NetBIOS name associated with the connection. |
| remote Host | The name or IP address associated with the remote computer. |
| <03> | The last byte of a NetBIOS name converted to hexadecimal. Each NetBIOS name is 16 characters long. This last byte often has special significance because the same name might be present several times on a computer, differing only in the last byte. For example, <20> is a space in ASCII text. |
| type | The type of name. A name can either be a unique name or a group name. |
| Status | Whether the NetBIOS service on the remote computer is running (registered) or a duplicate computer name has registered the same service (Conflict). |
| State | The state of NetBIOS connections. |

- The following table describes the possible NetBIOS connection states:

| STATE | DESCRIPTION |
|---------------|---|
| Connected | A session has been established. |
| associated | A connection endpoint has been created and associated with an IP address. |
| listening | This endpoint is available for an inbound connection. |
| Idle | This endpoint has been opened but cannot receive connections. |
| Connecting | A session is in the connecting phase and the name-to-IP address mapping of the destination is being resolved. |
| Accepting | An inbound session is currently being accepted and will be connected shortly. |
| Reconnecting | A session is trying to reconnect (it failed to connect on the first attempt). |
| Outbound | A session is in the connecting phase and the TCP connection is currently being created. |
| Inbound | An inbound session is in the connecting phase. |
| Disconnecting | A session is in the process of disconnecting. |
| Disconnected | The local computer has issued a disconnect and it is waiting for confirmation from the remote system. |

- This command is available only if the Internet Protocol (TCP/IP) protocol is installed as a component in the properties of a network adapter in Network Connections.

Examples

To display the NetBIOS name table of the remote computer with the NetBIOS computer name of CORP07, type:

```
nbtstat /a CORP07
```

To display the NetBIOS name table of the remote computer assigned the IP address of 10.0.0.99, type:

```
nbtstat /A 10.0.0.99
```

To display the NetBIOS name table of the local computer, type:

```
nbtstat /n
```

To display the contents of the local computer NetBIOS name cache, type:

```
nbtstat /c
```

To purge the NetBIOS name cache and reload the #PRE-tagged entries in the local Lmhosts file, type:

```
nbtstat /R
```

To release the NetBIOS names registered with the WINS server and re-register them, type:

```
nbtstat /RR
```

To display NetBIOS session statistics by IP address every five seconds, type:

```
nbtstat /S 5
```

additional references

- [Command-Line Syntax Key](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Installs the Windows Preinstallation Environment (WinPE), a lightweight version of Windows used to deploy workstations.

Syntax

```
netcfg [/v] [/e] [/winpe] [/l ] /c /i
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| /v | Run in verbose (detailed) mode |
| /e | Use servicing environment variables during install and uninstall |
| /winpe | Installs TCP/IP, NetBIOS and Microsoft Client for Windows preinstallation environment |
| /l | Provides the location of INF |
| /c | Provides the class of the component to be installed; protocol, Service, or client |
| /i | Provides the component ID |
| /s | Provides the type of components to show \ta = adapters, n = net components |
| /? | Displays help at the command prompt. |

Examples

To install the protocol *example* using c:\oemdir\example.inf:

```
netcfg /l c:\oemdir\example.inf /c p /i example
```

To install the *MS_Server* service:

```
netcfg /c s /i MS_Server
```

To install TCP/IP, NetBIOS and Microsoft Client for Windows preinstallation environment

```
netcfg /v /winpe
```

To display if component *MS_IPX* is installed:

```
netcfg /q MS_IPX
```

To uninstall component *MS_IPX*:

```
netcfg /u MS_IPX
```

To show all installed net components:

```
netcfg /s n
```

To shows binding paths containing *MS_TCPIP*:

```
netcfg /b ms_tcpip
```

additional references

- [Command-Line Syntax Key](#)

netsh

4/13/2018 • 2 minutes to read • [Edit Online](#)

Netsh is a command-line scripting utility that allows you to, either locally or remotely, display or modify the network configuration of a currently running computer.

There are functional differences between **netsh** commands on Windows Server® 2003, Windows Server® 2008, and Windows Server® 2008 R2:

- For more information about **netsh** commands on Windows Server 2003, see [Netsh](#).
- For more information about **netsh** commands for Windows Server 2008 and Windows Server 2008 R2, see [Netsh Technical Reference](#).

netstat

7/11/2018 • 3 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Displays active TCP connections, ports on which the computer is listening, Ethernet statistics, the IP routing table, IPv4 statistics (for the IP, ICMP, TCP, and UDP protocols), and IPv6 statistics (for the IPv6, ICMPv6, TCP over IPv6, and UDP over IPv6 protocols). Used without parameters, **netstat** displays active TCP connections.

Syntax

```
netstat [-a] [-e] [-n] [-o] [-p <Protocol>] [-r] [-s] [<Interval>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| -a | Displays all active TCP connections and the TCP and UDP ports on which the computer is listening. |
| -e | Displays Ethernet statistics, such as the number of bytes and packets sent and received. This parameter can be combined with -s . |
| -n | Displays active TCP connections, however, addresses and port numbers are expressed numerically and no attempt is made to determine names. |
| -o | Displays active TCP connections and includes the process ID (PID) for each connection. You can find the application based on the PID on the Processes tab in Windows Task Manager. This parameter can be combined with -a , -n , and -p . |
| -p | Shows connections for the protocol specified by <i>Protocol</i> . In this case, the <i>Protocol</i> can be tcp, udp, tcpv6, or udpv6. If this parameter is used with -s to display statistics by protocol, <i>Protocol</i> can be tcp, udp, icmp, ip, tcpv6, udpv6, icmpv6, or ipv6. |
| -s | Displays statistics by protocol. By default, statistics are shown for the TCP, UDP, ICMP, and IP protocols. If the IPv6 protocol is installed, statistics are shown for the TCP over IPv6, UDP over IPv6, ICMPv6, and IPv6 protocols. The -p parameter can be used to specify a set of protocols. |
| -r | Displays the contents of the IP routing table. This is equivalent to the route print command. |

| PARAMETER | DESCRIPTION |
|-----------|--|
| | Redisplays the selected information every <i>Interval</i> seconds. Press CTRL+C to stop the redisplay. If this parameter is omitted, netstat prints the selected information only once. |
| /? | Displays help at the command prompt. |

Remarks

- Parameters used with this command must be prefixed with a hyphen (-) rather than a slash (/).
- netstat** provides statistics for the following:
 - Proto The name of the protocol (TCP or UDP).
 - Local address The IP address of the local computer and the port number being used. The name of the local computer that corresponds to the IP address and the name of the port is shown unless the **-n** parameter is specified. If the port is not yet established, the port number is shown as an asterisk (*).
 - foreign address The IP address and port number of the remote computer to which the socket is connected. The names that corresponds to the IP address and the port are shown unless the **-n** parameter is specified. If the port is not yet established, the port number is shown as an asterisk (*).
 - (state) Indicates the state of a TCP connection. The possible states are as follows: CLOSE_WAIT CLOSED ESTABLISHED FIN_WAIT_1 FIN_WAIT_2 LAST_ACK listEN SYN_RECEIVED SYN_SEND timeD_WAIT for more information about the states of a TCP connection, see Rfc 793.
- This command is available only if the Internet Protocol (TCP/IP) protocol is installed as a component in the properties of a network adapter in Network Connections.

Examples

To display both the Ethernet statistics and the statistics for all protocols, type:

```
netstat -e -s
```

To display the statistics for only the TCP and UDP protocols, type:

```
netstat -s -p tcp udp
```

To display active TCP connections and the process IDs every 5 seconds, type:

```
netstat -o 5
```

To display active TCP connections and the process IDs using numerical form, type:

```
netstat -n -o
```

additional references

- [Command-Line Syntax Key](#)

Net print

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Displays information about a specified printer queue or a specified print job, or controls a specified print job. for examples of how to use this command, see the [Examples](#) section of this document.

NOTE

This command has been deprecated in Windows 7 and Windows Server 2008 R2 . However, you can perform many of the same tasks using prnjobs, Windows Management Instrumentation (WMI), or Windows PowerShell cmdlets. For more information, see [prnjobs](#), [Windows Management Instrumentation](#) (<https://go.microsoft.com/fwlink/?LinkID=29991>), [Windows PowerShell](#) (<https://go.microsoft.com/fwlink/?LinkID=128426>), and the [TechNet Script Center Gallery](#) (<https://go.microsoft.com/fwlink/?LinkId=164635>).

Syntax

```
Net print {\\<computerName>\<Sharename> |  
\\<computerName> <JobNumber> [/hold | /release | /delete]} [help]
```

Parameters

| PARAMETERS | DESCRIPTION |
|------------------------------|---|
| \\ | Specifies (by name) the computer and print queue about which you want to display information. |
| \\ | Specifies (by name) the computer that hosts the print job you want to control. If you do not specify a computer, the local computer is assumed. Requires the parameter. |
| | Specifies the number of the print job you want to control. This number is assigned by the computer that hosts the print queue where the print job is sent. After a computer assigns a number to a print job, that number is not assigned to any other print jobs in any queue hosted by that computer. Required when using the \\ parameter. |
| [/hold /release /delete] | <p>Specifies the action to take with the print job.</p> <ul style="list-style-type: none">- The /hold parameter delays the job, allowing other print jobs to bypass it until it is released.- The /release parameter releases a print job that has been delayed.- The /delete parameter removes a print job from a print queue. <p>if you specify a job number, but do not specify any action, information about the print job is displayed.</p> |
| help | Displays help for the Net print command. |

Remarks

- **Net print ** displays information about print jobs in a shared printer queue. The following is an example of a report for all print jobs in a queue for a shared printer named LASER:

```
printers at \\PRODUCTION Name Job # Size Status ----- LASER Queue 3 jobs
*printer active* USER1 84 93844 printing USER2 85 12555 Waiting USER3 86 10222 Waiting
```

- The following is an example of a report for a print job:

```
Job # 35 Status Waiting Size 3096 remark Submitting user USER2 Notify USER2 Job data type Job parameters
additional info
```

Examples This example shows how to list the contents of the Dotmatrix print queue on the \\Production computer:

```
Net print \\Production\Dotmatrix
```

This example shows how to display information about job number 35 on the

\\Production computer:

```
Net print \\Production 35
```

This example shows how to delay job number 263 on the

\\Production computer:

```
Net print \\Production 263 /hold
```

This example shows how to release job number 263 on

the \\Production computer:

```
Net print \\Production 263 /release
```

additional references [Command-Line Syntax](#)

[Key print Command Reference](#)

nfsadmin

10/17/2017 • 8 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

You can use **nfsadmin** to manage Server for NFS and Client for NFS.

Syntax

```
nfsadmin server [ computerName ] [ -u UserName [ -p Password ] ] -l  
nfsadmin server [ computerName ] [ -u UserName [ -p Password ] ] -r { client | all }  
nfsadmin server [ computerName ] [ -u UserName [ -p Password ] ] { start | stop }  
nfsadmin server [ computerName ] [ -u UserName [ -p Password ] ] config Option [...]   
nfsadmin server [ computerName ] [ -u UserName [ -p Password ] ] creategroup Name  
nfsadmin server [ computerName ] [ -u UserName [ -p Password ] ] listgroups  
nfsadmin server [ computerName ] [ -u UserName [ -p Password ] ] deletegroup Name  
nfsadmin server [ computerName ] [ -u UserName [ -p Password ] ] renamegroup OldName NewName  
nfsadmin server [ computerName ] [ -u UserName [ -p Password ] ] addmembers Name Host [...]   
nfsadmin server [ computerName ] [ -u UserName [ -p Password ] ] listmembers  
nfsadmin server [ computerName ] [ -u UserName [ -p Password ] ] deletemembers Group Host [...]   
nfsadmin client [ computerName ] [ -u UserName [ -p Password ] ] { start | stop }  
nfsadmin client [ computerName ] [ -u UserName [ -p Password ] ] config Option [...]
```

Description

The **nfsadmin** command-line utility administers Server for NFS or Client for NFS on the local or remote computer running Microsoft Services for Network File System (NFS). If you are logged on with an account that does not have the required privileges, you can specify a user name and password of an account that does. The action performed by **nfsadmin** depends on the command arguments you supply.

In addition to service-specific command arguments and options, **nfsadmin** accepts the following:

computerName

Specifies the remote computer you want to administer. You can specify the computer using a Windows Internet Name Service (WINS) name or a Domain Name System (DNS) name, or by Internet Protocol (IP) address.

-u *UserName*

Specifies the user name of the user whose credentials are to be used. It might be necessary to add the domain name to the user name in the form *domain\UserName*

-p *Password*

Specifies the password of the user specified using the **-u** option. If you specify the **-u** option but omit the **-p** option,

you are prompted for the user's password.

Administering Server for NFS

Use the **nfsadmin server** command to administer Server for NFS. The specific action that **nfsadmin server** takes depends on the command option or argument you specify:

-l

Lists all locks held by clients.

-r {client | all}

Releases the locks held by a *client* or, if **all** is specified, by all clients.

start

Starts the Server for NFS service.

stop

Stops the Server for NFS service.

config

Specifies general settings for Server for NFS. You must supply at least one of the following options with the **config** command argument:

mapsvr=server

Sets *server* as the User Name Mapping server for Server for NFS. Although this option continues to be supported for compatibility with previous versions, you should use the **sfuadmin** utility instead.

auditlocation={eventlog | file | both | none}

Specifies whether events will be audited and where the events will be recorded. One of the following arguments is required.

eventlog

Specifies that audited events will be recorded only in the Event Viewer application log.

file

Specifies that audited events will be recorded only in the file specified by **config fname**.

both

Specifies that audited events will be recorded in the Event Viewer application log as well as the file specified by **config fname**.

none

Specifies that events will not be audited.

fname=file

Sets the file specified by *file* as the audit file. The default is %sfudir%\log\nfssvr.log

fsize==size

Sets *size* as the maximum size in megabytes of the audit file. The default maximum size is 7 MB.

audit=[+|-]mount [+|-]read [+|-]write [+|-]create [+|-]delete [+|-]locking [+|-]all

Specifies the events to be logged. To start logging an event, type a plus sign (+) before the event name; to stop logging an event, type a minus sign (-) before the event name. If the sign is omitted, the plus sign is assumed. Do not use **all** with any other event name.

lockperiod=seconds

Specifies the number of seconds that Server for NFS will wait to reclaim locks after a connection to Server for NFS has been lost and then reestablished or after the Server for NFS service has been restarted.

Portmapprotocol={TCP | UDP | TCP+UDP}

Specifies which transport protocols Portmap supports. The default setting is **TCP+UDP**.

mountprotocol={TCP | UDP | TCP+UDP}

Specifies which transport protocols mount supports. The default setting is **TCP+UDP**.

nfsprotocol={TCP | UDP | TCP+UDP}

Specifies which transport protocols Network File System (NFS) supports. The default setting is **TCP+UDP**.

nlmprotocol={TCP | UDP | TCP+UDP}

Specifies which transport protocols Network Lock Manager (NLM) supports. The default setting is **TCP+UDP**.

nsmprotocol={TCP | UDP | TCP+UDP}

Specifies which transport protocols Network Status Manager (NSM) supports. The default setting is **TCP+UDP**.

enableV3={yes | no}

Specifies whether NFS version 3 protocols will be supported. The default setting is **yes**.

renewauth={yes | no}

Specifies whether client connections will be required to be reauthenticated after the period specified by **config renewauthinterval**. The default setting is **no**.

renewauthinterval=seconds

Specifies the number of seconds that elapse before a client is forced to be reauthenticated if **config renewauth** is set to **yes**. The default value is 600 seconds.

dircache=size

Specifies the size in kilobytes of the directory cache. The number specified as *size* must be a multiple of 4 between 4 and 128. The default directory-cache size is 128 KB.

translationfile=[file]

Specifies a file containing mapping information for replacing characters in the names of files when moving them from Windows-based to UNIX-based file systems. If *file* is not specified, then file name character translation is disabled. If the value of **translationfile** is changed, you must restart the server for the change to take effect.

dotfileshidden={yes | no}

Specifies whether files that are created with names beginning with a period (.) will be marked as hidden in the Windows file system and consequently hidden from NFS clients. The default setting is **no**.

casesensitivelookups={yes | no}

Specifies whether directory lookups will be case sensitive (requiring exact matching of character case).

You also need to disable Windows kernel case-insensitivity in order for Server for NFS to support case-sensitive file names. You can disable Windows kernel case-insensitivity by clearing the following registry key to 0:

HKLM\SYSTEM\CurrentControlSet\Control\Session Manager\kernel

DWORD obcaseinsensitive

IMPORTANT

This section applies only to Windows Server 2008 R2, Windows Server 2008, and Windows Server 2003. This section does not apply to Windows Server 2012 R2 or Windows Server 2012.

ntfscase={lower | upper | preserve}

Specifies whether the case of characters in the names of files in the NTFS file system will be returned in lowercase, uppercase, or in the form stored in the directory. The default setting is **preserve**. This setting cannot be changed if **casesensitivelookups** is set to **yes**.

creategroup name

creates a new client group, giving it the specified *name*.

listgroups

Displays the names of all client groups.

deletegroup *name*

removes the client group specified by *name*.

renamegroup *OldName NewName*

changes the name of the client group specified by *OldName* to *NewName*

addmembers *Name Host[...]*

adds *Host* to the client group specified by *Name*.

listmembers *Name*

lists the host computers in the client group specified by *Name*.

deletemembers *Group Host[...]*

removes the client specified by *Host* from the client group specified by *Group*.

if you do not specify a command option or argument, **nfsadmin server** displays the current Server for NFS configuration settings.

Administering Client for NFS

Use the **nfsadmin client** command to administer Client for NFS. The specific action that **nfsadmin client** takes depends on the command argument you specify:

start

starts the Client for NFS service.

stop

Stops the Client for NFS service.

config

Specifies general settings for Client for NFS. You must supply at least one of the following options with the **config** command argument:

fileaccess=mode

- Specifies the default permission mode for files created on Network File System (NFS) servers. The *mode* argument consists of a three digits from 0 to 7 (inclusive) representing the default permissions granted the user, group, and others (respectively). The digits translate to UNIX-style permissions as follows: 0=none, 1=x, 2=w, 3=wx, 4=r, 5=rx, 6=rw, and 7=rwx. For example, **fileaccess=750** gives rwx permission to the owner, rx permission to the group, and no access permission to others.

mapsvr=server

Sets *server* as the User Name Mapping server for Client for NFS. Although this option continues to be supported for compatibility with previous versions, you should use the **sfuadmin** utility instead.

mtype={hard | soft}

Specifies the default mount type. For a hard mount, Client for NFS continues to retry a failed RPC until it succeeds. For a soft mount, Client for NFS returns failure to the calling application after retrying the call the number of times specified by the **retry** option.

retry=number

Specifies the number of times to try to make a connection for a soft mount. This value must be from 1 to 10, inclusive. The default is 1.

timeout=seconds

Specifies the number of seconds to wait for a connection (remote procedure call). This value must be 0.8, 0.9, or an integer from 1 to 60, inclusive. The default is 0.8.

Protocol={TCP | UDP | TCP+UDP}

Specifies which transport protocols the client supports. The default setting is **TCP+UDP**

rsize=size

Specifies the size, in kilobytes, of the read buffer. This value can be 0.5, 1, 2, 4, 8, 16, or 32. The default is 32.

wsiz=*size*

Specifies the size, in kilobytes, of the write buffer. This value can be 0.5, 1, 2, 4, 8, 16, or 32. The default is 32.

perf=default

Restores the following performance settings to default values:

- **mtype**
- **retry**
- **timeout**
- **rsize**
- **wsiz**

fileaccess=*mode*

Specifies the default permission mode for files created on Network File System (NFS) servers. The *mode* argument consists of a three digits from 0 to 7 (inclusive) representing the default permissions granted the user, group, and others (respectively). The digits translate to UNIX-style permissions as follows: 0=none, 1=x, 2=w, 3=wx, 4=r, 5=rx, 6=rw, and 7=rwx. For example, **fileaccess=750** gives rwx permission to the owner, rx permission to the group, and no access permission to others.

if you do not specify a command option or argument, **nfsadmin client** displays the current Client for NFS configuration settings.

nfsshare

4/13/2018 • 2 minutes to read • [Edit Online](#)

You can use **nfsshare** to control Network File System (NFS) shares.

Syntax

```
nfsshare <ShareName>=<Drive:Path> [-o <Option=value>...]  
nfsshare {<ShareName> | <Drive>:<Path> | * } /delete
```

Description

Without arguments, the **nfsshare** command-line utility lists all Network File System (NFS) shares exported by Server for NFS. With *ShareName* as the only argument, **nfsshare** lists the properties of the NFS share identified by *ShareName*. When *ShareName* and *Drive:Path* are provided, **nfsshare** exports the folder identified by *Drive:Path* as *ShareName*. When the **/delete** option is used, the specified folder is no longer made available to NFS clients.

Options

The **nfsshare** command accepts the following options and arguments:

| TERM | DEFINITION |
|------------------------|--|
| -o anon={yes | no} |
| -o rw[= <Host> [:]...] | Provides read-write access to the shared directory by the hosts or client groups specified by <i>Host</i> . Separate host and group names with a colon (:). If <i>Host</i> is not specified, all hosts and client groups (except those specified with the ro option) have read-write access. If neither the ro nor the rw option is set, all clients have read-write access to the shared directory. |
| -o ro[= <Host> [:]...] | Provides read-only access to the shared directory by the hosts or client groups specified by <i>Host</i> . Separate host and group names with a colon (:). If <i>Host</i> is not specified, all clients (except those specified with the rw option) have read-only access. If the ro option is set for one or more clients, but the rw option is not set, only the clients specified with the ro option have access to the shared directory. |
| -o encoding={big5 | euc-jp |
| -o anongid= <gid> | Specifies that anonymous (unmapped) users will access the share directory using <i>gid</i> as their group identifier (GID). The default is -2. The anonymous GID will be used when reporting the owner of a file owned by an unmapped user, even if anonymous access is disabled. |

| TERM | DEFINITION |
|-------------------------|---|
| -o anonuid= <uid> | Specifies that anonymous (unmapped) users will access the share directory using <i>uid</i> as their user identifier (UID). The default is -2. The anonymous UID will be used when reporting the owner of a file owned by an unmapped user, even if anonymous access is disabled. |
| -o root[= <Host>[:]...] | Provides root access to the shared directory by the hosts or client groups specified by <i>Host</i> . Separate host and group names with a colon (:). If <i>Host</i> is not specified, all clients have root access. If the root option is not set, no clients have root access to the shared directory. |
| /delete | If <i>ShareName</i> or <i>Drive:Path</i> is specified, deletes the specified share. If * is specified, deletes all NFS shares. |

NOTE

To view the complete syntax for this command, at a command prompt, type:

> **nfsshare /?**

#

[Services for Network File System Command Reference](#) See Also

nfsstat

4/13/2018 • 2 minutes to read • [Edit Online](#)

You can use **nfsstat** to display or reset counts of calls made to Server for NFS.

Syntax

```
nfsstat [-z]
```

Description

When used without the **-z** option, the **nfsstat** command-line utility displays the number of NFS V2, NFS V3, and Mount V3 calls made to the server since the counters were set to 0, either when the service started or when the counters were reset using **nfsstat -z**.

nlbmgr

4/13/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Using Network Load Balancing Manager, you can configure and manage your Network Load Balancing clusters and all cluster hosts from a single computer, and you can also replicate the cluster configuration to other hosts. You can start Network Load Balancing Manager from the command-line using the command **nlbmgr.exe**, which is installed in the **systemroot\System32** folder.

Syntax

```
nlbmgr [/help] [/noping] [/hostlist <filename>] [/autorefresh <interval>]
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------|---|
| /help | Displays help at the command prompt. |
| /noping | Prevents Network Load Balancing Manager from pinging the hosts prior to trying to contact them through Windows Management Instrumentation (WMI). Use this option if you have disabled Internet Control Message Protocol (ICMP) on all available network adapters. If Network Load Balancing Manager attempts to contact a host that is not available, you will experience a delay when using this option. |
| /hostlist | Loads the hosts specified in filename into Network Load Balancing Manager. |
| /autorefresh | Causes Network Load Balancing Manager to refresh its host and cluster information every seconds. If no interval is specified, the information is refreshed every 60 seconds. |
| /? | Displays help at the command prompt. |

additional references

- [Command-Line Syntax Key](#)

nslookup

7/11/2018 • 5 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Displays information that you can use to diagnose Domain Name System (DNS) infrastructure. Before using this tool, you should be familiar with how DNS works. The nslookup command-line tool is available only if you have installed the TCP/IP protocol.

Syntax

```
nslookup [<-SubCommand ...>] [{<computerToFind> | -<Server>}]
nslookup /exit
nslookup /finger [<UserName>] [{<[>] <FileName>|<[>>] <FileName>}]
nslookup /{help | ?}
nslookup /ls [<Option>] <DNSDomain> [{<[>] <FileName>|<[>>] <FileName>}]
nslookup /lserver <DNSDomain>
nslookup /root
nslookup /server <DNSDomain>
nslookup /set <Keyword>[=<Value>]
nslookup /set all
nslookup /set class=<Class>
nslookup /set [no]d2
nslookup /set [no]debug
nslookup /set [no]defname
nslookup /set domain=<DomainName>
nslookup /set [no]ignore
nslookup /set port=<Port>
nslookup /set querytype=<ResourceRecordtype>
nslookup /set [no]recurse
nslookup /set retry=<Number>
nslookup /set root=<RootServer>
nslookup /set [no]search
nslookup /set srchlist=<DomainName>[/...]
nslookup /set timeout=<Number>
nslookup /set type=<ResourceRecordtype>
nslookup /set [no]vc
nslookup /view <FileName>
```

Parameters

| PARAMETER | DESCRIPTION |
|---|--|
| nslookup exit Command | exits nslookup . |
| nslookup finger Command | Connects with the finger server on the current computer. |
| nslookup help | Displays a short summary of nslookup subcommands. |
| nslookup ls | lists information for a DNS domain. |
| nslookup lserver | changes the default server to the specified DNS domain. |

| PARAMETER | DESCRIPTION |
|--|---|
| nslookup root | changes the default server to the server for the root of the DNS domain name space. |
| nslookup server | changes the default server to the specified DNS domain. |
| nslookup set | changes configuration settings that affect how lookups function. |
| nslookup set all | prints the current values of the configuration settings. |
| nslookup set class | changes the query class. The class specifies the protocol group of the information. |
| nslookup set d2 | Turns exhaustive Debugging mode on or off. All fields of every packet are printed. |
| nslookup set debug | Turns Debugging mode on or off. |
| nslookup /set defname | appends the default DNS domain name to a single component lookup request. A single component is a component that contains no periods. |
| nslookup set domain | changes the default DNS domain name to the name specified. |
| nslookup /set ignore | Ignores packet truncation errors. |
| nslookup set port | changes the default TCP/UDP DNS name server port to the value specified. |
| nslookup set querytype | changes the resource record type for the query. |
| nslookup set recurse | Tells the DNS name server to query other servers if it does not have the information. |
| nslookup set retry | Sets the number of retries. |
| nslookup set root | changes the name of the root server used for queries. |
| nslookup set search | appends the DNS domain names in the DNS domain search list to the request until an answer is received. This applies when the set and the lookup request contain at least one period, but do not end with a trailing period. |
| nslookup set srclist | changes the default DNS domain name and search list. |
| nslookup set timeout | changes the initial number of seconds to wait for a reply to a request. |
| nslookup set type | changes the resource record type for the query. |
| nslookup set vc | Specifies to use or not use a virtual circuit when sending requests to the server. |

| PARAMETER | DESCRIPTION |
|-------------------------------|--|
| nslookup view | sorts and lists the output of the previous ls subcommand or commands. |

Remarks

- if *computerToFind* is an IP address and the query is for an A or PTR resource record type, the name of the computer is returned. If *computerToFind* is a name and does not have a trailing period, the default DNS domain name is appended to the name. This behavior depends on the state of the following **set** subcommands: **domain**, **srchlist**, **defname**, and **search**.
- if you type a hyphen (-) instead of *computerToFind*, the command prompt changes to **nslookup** interactive mode.
- The command-line length must be less than 256 characters.
- **nslookup** has two modes: interactive and noninteractive. if you need to look up only a single piece of data, use noninteractive mode. For the first parameter, type the name or IP address of the computer that you want to look up. For the second parameter, type the name or IP address of a DNS name server. If you omit the second argument, **nslookup** uses the default DNS name server. if you need to look up more than one piece of data, you can use interactive mode. type a hyphen (-) for the first parameter and the name or IP address of a DNS name server for the second parameter. Or, omit both parameters and **nslookup** uses the default DNS name server. Following are some tips about working in interactive mode:
 - To interrupt interactive commands at any time, press CTRL+B.
 - To exit, type **exit**.
 - To treat a built-in command as a computer name, precede it with the escape character (\).
 - An unrecognized command is interpreted as a computer name.
- if the lookup request fails, **nslookup** prints an error message. The following table lists possible error messages.

| Error message | Description |
|-------------------------|---|
| timed out | The server did not respond to a request after a certain amount of time and a certain number of retries. You can set the time-out period with the set timeout subcommand. You can set the number of retries with the set retry subcommand. |
| No response from server | No DNS name server is running on the server computer. |
| No records | The DNS name server does not have resource records of the current query type for the computer, although the computer name is valid. The query type is specified with the set querytype command. |
| Nonexistent domain | The computer or DNS domain name does not exist. |
| Connection refused | |

-or-

| | |
|------------------------|---|
| Network is unreachable | The connection to the DNS name server or finger server could not be made. This error commonly occurs with ls and finger requests. |
| Server failure | The DNS name server found an internal inconsistency in its database and could not return a valid answer. |
| Refused | The DNS name server refused to service the request. |
| format error | The DNS name server found that the request packet was not in the proper format. It may indicate an error in nslookup . |

- for more information about the **nslookup** command and DNS, see the following resources:
 - Lee, T., Davies, J. 2000. *Microsoft Windows 2000 TCP/IP Protocols and Services Technical Reference*. Redmond, Washington: Microsoft Press.
 - Albitz, P., Loukides, M. and C. Liu. 2001. *DNS and BIND, Fourth edition*. Sebastopol, California: O'Reilly and associates, Inc.
 - Larson, M. and C. Liu. 2001. *DNS on Windows 2000*. Sebastopol, California: O'Reilly and associates, Inc.
- #### Examples Each command-line option consists of a hyphen (-) followed immediately by the command name and, in some cases, an equal sign (=) and then a value. For example, to change the default query type to host (computer) information and the initial time-out to 10 seconds, type: **nslookup**

-querytype=hinfo -timeout=10 ## See Also [Command-Line Syntax Key](#)

nslookup exit Command

4/13/2018 • 2 minutes to read • [Edit Online](#)

Exits **nslookup**.

Syntax

```
Nslookup /exit
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|-------------|
| {help | ? |

Additional references

[Command-Line Syntax Key](#)

nslookup finger Command

4/13/2018 • 2 minutes to read • [Edit Online](#)

Connects with the finger server on the current computer.

Syntax

```
finger [<UserName>] [{> <FileName>|>> <FileName>}]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|---|
| <UserName> | Specifies the name of the user to look up. |
| <FileName> | Specifies a file name in which to save the output. You can use the greater than (>) and double greater than (>>) characters to redirect the output in the usual manner. |
| {help | ?} |

Additional references

[Command-Line Syntax Key](#)

nslookup help

4/13/2018 • 2 minutes to read • [Edit Online](#)

Displays a short summary of **nslookup** subcommands.

Syntax

```
{help | ?}
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|-------------|
| {help | ? |

Additional references

[Command-Line Syntax Key](#)

nslookup ls

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Lists information for a Domain Name System (DNS) domain.

Syntax

```
ls [<Option>] <DNSDomain> [{>] <FileName>|>>] <FileName>}]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|---|
| | <p>The following table lists valid options.</p> <ul style="list-style-type: none">- -t: lists all records of the specified type. For a description of , see setquerytype in additional references.- -a: lists aliases of computers in the DNS domain. This parameter is a synonym for -t CNAME- -d: lists all records for the DNS domain. This parameter is a synonym for -t ANY- -h: lists CPU and operating system information for the DNS domain. This parameter is a synonym for -t HINFO- -s: lists well-known services of computers in the DNS domain. This parameter is a synonym for -t WKS. |
| | Specifies the DNS domain for which you want information. |
| | Specifies a file name in which to save the output. You can use the greater than (>) and double greater than (>>) characters to redirect the output in the usual manner. |
| {help ?} | Displays a short summary of nslookup subcommands. |

Remarks

- The default output contains computer names and their IP addresses. When output is directed to a file, hash marks are printed for every 50 records received from the server ## additional references [Command-Line Syntax Key nslookup set querytype](#)

nslookup lserver

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

changes the default server to the specified Domain Name System (DNS) domain.

Syntax

```
lserver <DNSDomain>
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|--|
| | Specifies the new DNS domain for the default server. |
| {help ?} | Displays a short summary of nslookup subcommands. |

Remarks

- The **lserver** command uses the initial server to look up the information about the specified DNS domain. This is in contrast to the **server** command, which uses the current default server. ## additional references [Command-Line Syntax Key nslookup server](#)

nslookup root

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

changes the default server to the server for the root of the Domain Name System (DNS) domain name space.

Syntax

```
root
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|--|
| {help ?} | Displays a short summary of nslookup subcommands. |

Remarks

- Currently, the ns.nic.ddn.mil name server is used. This command is a synonym for lserver ns.nic.ddn.mil. You can change the name of the root server with the **set root** command. ## additional references [Command-Line Syntax Key nslookup set root](#)

nslookup server

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

changes the default server to the specified Domain Name System (DNS) domain.

Syntax

```
server <DNSDomain>
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|--|
| | Required. Specifies the new DNS domain for the default server. |
| {help ?} | Displays a short summary of nslookup subcommands. |

Remarks

- The **server** command uses the current default server to look up the information about the specified DNS domain. This is in contrast to the **lserver** command, which uses the initial server. ## additional references [Command-Line Syntax Key nslookup lserver](#)

nslookup set

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

changes configuration settings that affect how lookups function.

Syntax

```
set <KeyWord>[=<Value>]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|--|
| | Identifies subcommands that are derived from the set subcommand. For example, the subcommand set d2 has a keyword of [no]d2 . For the list of subcommands that are derived from the set subcommand, see additional references. |
| | Specifies the nslookup configuration setting value for each subcommand. |
| {help ?} | Displays a short summary of nslookup subcommands. |

Remarks

Use **set all** to see a listing of the current settings.

additional references

[Command-Line Syntax Key nslookup set all](#)

nslookup set all

4/13/2018 • 2 minutes to read • [Edit Online](#)

Prints the current values of the configuration settings.

Syntax

```
set all
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|-------------|
| {help | ? |

Remarks

- **Set all** also prints information about the default server and computer (that is, the host).

Additional references

[Command-Line Syntax Key](#)

nslookup set class

4/13/2018 • 2 minutes to read • [Edit Online](#)

Changes the query class. The class specifies the protocol group of the information.

Syntax

```
set class=<Class>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| <Class> | The default class is IN. The following lists the valid values for this command. <ul style="list-style-type: none">- IN: Specifies the Internet class.- CHAOS: Specifies the Chaos class.- HESIOD: Specifies the MIT Athena Hesiod class.- ANY: Specifies any of the previously listed wildcards. |
| {help | ? |

Additional references

[Command-Line Syntax Key](#)

nslookup set d2

4/13/2018 • 2 minutes to read • [Edit Online](#)

Turns exhaustive Debugging Mode on or off. All fields of every packet are printed.

Syntax

```
set [no]d2
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------|--|
| nod2 | Turns off exhaustive Debugging Mode. The default syntax is nod2 . |
| d2 | Turns on exhaustive Debugging Mode. |
| {help | ? |

Additional references

[Command-Line Syntax Key](#)

nslookup set debug

4/13/2018 • 2 minutes to read • [Edit Online](#)

Turns Debugging Mode on or off.

Syntax

```
set [no]debug
```

Parameters

| PARAMETER | DESCRIPTION |
|----------------|--|
| nodebug | Turns off Debugging Mode. The default syntax is nodebug . |
| debug | Turns on Debugging Mode. |
| **{help | ?}** |

Remarks

- With Debugging Mode turned on, more information is printed about the packet sent to the server and the resulting answer.

Additional references

[Command-Line Syntax Key](#)

nslookup set domain

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

changes the default Domain Name System (DNS) domain name to the name specified.

Syntax

```
set domain=<DomainName>
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|---|
| | Specifies a new name for the default DNS domain name. The default domain name is the host name. |
| {help ?} | Displays a short summary of nslookup subcommands. |

Remarks

- The default DNS domain name is appended to a lookup request depending on the state of the **defname** and **search** options. The DNS domain search list contains the parents of the default DNS domain if it has at least two components in its name. For example, if the default DNS domain is mfg.widgets.com, the search list is named both mfg.widgets.com and widgets.com. Use the **set srchlist** command to specify a different list and the **set all** command to display the list. ## additional references [Command-Line Syntax Key nslookup set srchlist](#)
[nslookup set all](#)

nslookup set port

4/13/2018 • 2 minutes to read • [Edit Online](#)

Changes the default TCP/UDP Domain Name System (DNS) name server port to the value specified.

Syntax

```
set port=<Port>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| <Port> | Specifies the new value for the default TCP/UDP DNS name server port. The default port is 53. |
| {help | ? |

Additional references

[Command-Line Syntax Key](#)

nslookup set querytype

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

changes the resource record type for the query.

Syntax

```
set querytype=<ResourceRecordtype>
```

Parameters

Specifies a DNS resource record type. The default resource record type is A. The following table lists the valid values for this command.

| VALUE | DESCRIPTION |
|-------|---|
| A | Specifies a computer's IP address |
| ANY | Specifies a computer's IP address. |
| CNAME | Specifies a canonical name for an alias. |
| GID | Specifies a group identifier of a group name. |
| HINFO | Specifies a computer's CPU and type of operating system. |
| MB | Specifies a mailbox domain name. |
| MG | Specifies a mail group member. |
| MINFO | Specifies mailbox or mail list information. |
| MR | Specifies the mail rename domain name. |
| MX | Specifies the mail exchanger. |
| NS | Specifies a DNS name server for the named zone. |
| PTR | Specifies a computer name if the query is an IP address; otherwise, specifies the pointer to other information. |
| SOA | Specifies the start-of-authority for a DNS zone. |
| TXT | Specifies the text information. |

| VALUE | DESCRIPTION |
|-------|---------------------------------|
| UID | Specifies the user identifier. |
| UINFO | Specifies the user information. |
| WKS | Describes a well-known service. |
| {help | ? |

Displays a short summary of **nslookup** subcommands

Remarks

- The **set type** command performs the same function as the **set querytype** command.
- for more information about resource record types, see Request for Comment (Rfc) 1035. ## additional references [Command-Line Syntax Key nslookup set type](#)

nslookup set recurse

4/13/2018 • 2 minutes to read • [Edit Online](#)

Tells the Domain Name System (DNS) name server to query other servers if it does not have the information.

Syntax

```
set [no]recurse
```

Parameters

| PARAMETER | DESCRIPTION |
|------------------|---|
| norecurse | Stops the Domain Name System (DNS) name server from querying other servers if it does not have the information. |
| recurse | Tells the Domain Name System (DNS) name server to query other servers if it does not have the information. The default syntax is recurse . |
| {help | ? |

Additional references

[Command-Line Syntax Key](#)

nslookup set retry

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Sets the number of retries.

Syntax

```
set retry=<Number>
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|--|
| | Specifies the new value for the number of retries. The default number of retries is 4. |
| {help ?} | Displays a short summary of nslookup subcommands. |

Remarks

- When a reply to a request is not received within a certain amount of time, the time-out period is doubled and the request is resent. The retry value controls how many times a request is resent before giving up. You can change the time-out period with the **set timeout** subcommand. ## additional references [Command-Line Syntax Key nslookup set timeout](#)

nslookup set root

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

changes the name of the root server used for queries.

Syntax

```
set root=<RootServer>
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|--|
| | Specifies the new name for the root server. The default value is ns.nic.ddn.mil. |
| {help ?} | Displays a short summary of nslookup subcommands. |

Remarks

- The **set root** subcommand affects the **root** subcommand. ## additional references [Command-Line Syntax Key nslookup root](#)

nslookup set search

4/13/2018 • 2 minutes to read • [Edit Online](#)

Appends the Domain Name System (DNS) domain names in the DNS domain search list to the request until an answer is received. This applies when the set and the lookup request contain at least one period, but do not end with a trailing period.

Syntax

```
set [no]search
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------|---|
| nosearch | Stops appending the Domain Name System (DNS) domain names in the DNS domain search list to the request. |
| search | Appends the Domain Name System (DNS) domain names in the DNS domain search list to the request until an answer is received. The default syntax is search . |
| {help | ?} |

Additional references

[Command-Line Syntax Key](#)

nslookup set srchlist

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

changes the default Domain Name System (DNS) domain name and search list.

Syntax

```
Set srchlist=<DomainName>[...]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|---|
| | Specifies new names for the default DNS domain and search list. The default domain name value is based on the host name. You can specify a maximum of six names separated by slashes (/). |
| {help ?} | Displays a short summary of nslookup subcommands. |

Remarks

- The **set srchlist** command overrides the default DNS domain name and search list of the **set domain** command. Use the **set all** command to display the list. ## Examples The following example sets the DNS domain to mfg.widgets.com and the search list to the three names:

```
set srchlist=mfg.widgets.com/mrp2.widgets.com/widgets.com ## additional references Command-Line Syntax
```

[Key nslookup set domain nslookup set all](#)

nslookup set timeout

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

changes the initial number of seconds to wait for a reply to a lookup request.

Syntax

```
set timeout=<Number>
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|--|
| | Specifies the number of seconds to wait for a reply. The default number of seconds to wait is 5. |
| {help ?} | Displays a short summary of nslookup subcommands. |

Remarks

- When a reply to a request is not received within the specified time period, the time-out is doubled and the request is sent again. You can use the **set retry** command to control the number of retries. ## Examples The following example sets the timeout for getting a response to 2 seconds: `set timeout=2` ## additional references [Command-Line Syntax Key nslookup set retry](#)

nslookup set type

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

changes the resource record type for the query.

Syntax

```
set type=<ResourceRecordtype>
```

Parameters

Specifies a DNS resource record type. The default resource record type is A. The following table lists the valid values for this command.

| VALUE | DESCRIPTION |
|-------|---|
| A | Specifies a computer's IP address |
| ANY | Specifies a computer's IP address. |
| CNAME | Specifies a canonical name for an alias. |
| GID | Specifies a group identifier of a group name. |
| HINFO | Specifies a computer's CPU and type of operating system. |
| MB | Specifies a mailbox domain name. |
| MG | Specifies a mail group member. |
| MINFO | Specifies mailbox or mail list information. |
| MR | Specifies the mail rename domain name. |
| MX | Specifies the mail exchanger. |
| NS | Specifies a DNS name server for the named zone. |
| PTR | Specifies a computer name if the query is an IP address; otherwise, specifies the pointer to other information. |
| SOA | Specifies the start-of-authority for a DNS zone. |
| TXT | Specifies the text information. |

| VALUE | DESCRIPTION |
|-------|---------------------------------|
| UID | Specifies the user identifier. |
| UINFO | Specifies the user information. |
| WKS | Describes a well-known service. |
| {help | ?} |

Displays a short summary of **nslookup** subcommands

Remarks

- The **set type** command performs the same function as the **set querytype** command.
- for more information about resource record types, see Request for Comment (Rfc) 1035. ## additional references [Command-Line Syntax Key nslookup set querytype](#)

nslookup set vc

4/13/2018 • 2 minutes to read • [Edit Online](#)

Specifies to use or not use a virtual circuit when sending requests to the server.

Syntax

```
set [no]vc
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------|--|
| novc | Specifies to never use a virtual circuit when sending requests to the server. The default is novc . |
| vc | Specifies to always use a virtual circuit when sending requests to the server. |
| {help | ? |

Additional references

[Command-Line Syntax Key](#)

nslookup view

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

sorts and lists the output of the previous **ls** subcommand or commands.

Syntax

```
view <FileName>
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|--|
| | Specifies the name of the file containing output from the previous ls subcommand or commands. |
| {help ?} | Displays a short summary of nslookup subcommands. |

additional references

[Command-Line Syntax Key](#)

[nslookup ls](#)

ntbackup

4/13/2018 • 2 minutes to read • [Edit Online](#)

The **ntbackup** command is not available in Windows Vista or Windows Server 2008. Instead, you should use the **wbadmin** command and subcommands to back up and restore your computer and files from a command prompt.

You cannot recover backups that you created with **ntbackup** by using **wbadmin**. However, a version of **ntbackup** is available as a download for Windows Server 2008 and Windows Vista users who want to recover backups that they created using **ntbackup**. This downloadable version of **ntbackup** enables you to perform recoveries only of legacy backups, and it cannot be used on computers running Windows Server 2008 or Windows Vista to create new backups. To download this version of **ntbackup**, see <https://go.microsoft.com/fwlink/?LinkId=82917>.

Additional references

[Command-Line Syntax Key](#)

[Wbadmin](#)

ntcmdprompt

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Runs the command interpreter **Cmd.exe**, rather than **Command.com**, after running a Terminate and Stay Resident (TSR) or after starting the command prompt from within an MS-DOS application.

Syntax

```
ntcmdprompt
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------------|
| /? | Displays help at the command prompt. |

Remarks

- When **Command.com** is running, some features of **Cmd.exe**, such as the **doskey** display of command history, are not available. If you would prefer to run the **Cmd.exe** command interpreter after you have started a Terminate and Stay Resident (TSR) or started the command prompt from within an application based on MS-DOS, you can use the **ntcmdprompt** command. However, keep in mind that the TSR may not be available for use when you are running **Cmd.exe**. You can include the **ntcmdprompt** command in your **Config.nt** file or the equivalent custom startup file in an application's program information file (Pif). ## Examples To include **ntcmdprompt** in your **Config.nt** file, or the configuration startup file specified in the Pif, type: **ntcmdprompt** ## additional references
- [Command-Line Syntax Key](#)

ntfrsutl

4/13/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Dumps the internal tables, thread, and memory information for the NT File Replication Service (NTFRS). It runs against local and remote servers. The recovery setting for NTFRS in Service Control Manager (SCM) can be critical to locating and keeping IMPORTANT log events on the computer. This tool provides a convenient method of reviewing those settings.

Syntax

```
ntfrsutl[idtable|configtable|inlog|outlog][<computer>]
ntfrsutl[memory|threads|stage][<computer>]
ntfrsutl ds[<computer>]
ntfrsutl [sets][<computer>]
ntfrsutl [version][<computer>]
ntfrsutl poll[/quickly[=<N>]][/slowly[=<N>]][/now][<computer>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------|---|
| idtable | ID table |
| configtable | FRS configuration table |
| inlog | Inbound log |
| outlog | Outbound log |
| | Specifies the computer. |
| memory | Memory usage |
| threads | |
| stage | |
| ds | lists the NTFRS service's view of the DS. |
| sets | Specifies the active replica sets |
| version | Specifies the API and NTFRS service versions. |

| PARAMETER | DESCRIPTION |
|-----------|--|
| poll | <p>Specifies the current polling intervals.</p> <p>Parameters:</p> <ul style="list-style-type: none"> • /quickly[=<i>N</i>] (Polls quickly) <ul style="list-style-type: none"> • quickly - Polls quickly until stable configuration is retrieved • quickly= - Polls quickly every default minutes. • quickly= - Polls quickly every <i>N</i> minutes • /slowly[=<i>N</i>] (Polls slowly) <ul style="list-style-type: none"> • slowly - Polls slowly until stable configuration is retrieved • slowly= - Polls slowly every default minutes • slowly= - Polls quickly every <i>N</i> minutes • /now (Polls now) |
| /? | Displays help at the command prompt. |

Examples

To determine the polling interval for file replication:

```
C:\Program Files\SupportTools>ntfrsutl poll wrkstn-1
```

To determine the current NTFRS application program interface (API) version:

```
C:\Program Files\SupportTools>ntfrsutl version
```

additional references

- [Command-Line Syntax Key](#)

openfiles

7/11/2018 • 4 minutes to read • [Edit Online](#)

Enables an administrator to query, display, or disconnect files and directories that have been opened on a system. Also enables or disables the system Maintain Objects List global flag.

This topic includes information about the following commands:

- [openfiles /disconnect](#)
- [openfiles /query](#)
- [openfiles /local](#)

openfiles /disconnect

Enables an administrator to disconnect files and folders that have been opened remotely through a shared folder.

Syntax

```
openfiles /disconnect [/s <System> [/u [<Domain>\]<UserName> [/p [<Password>]]]] {[ /id <OpenFileID> ] | [ /a <AccessedBy> ] | [ /o {read | write | read/write} ]} [/op <OpenFile>]
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------------------------|---|
| /s <System> | Specifies the remote system to connect to (by name or IP address). Do not use backslashes. If you do not use the /s option, the command is executed on the local computer by default. This parameter applies to all files and folders that are specified in the command. |
| /u [<Domain>] | Executes the command by using the permissions of the specified user account. If you do not use the /u option, system permissions are used by default. |
| /p [<Password>] | Specifies the password of the user account that is specified in the /u option. If you do not use the /p option, a password prompt appears when the command is executed. |
| /id <OpenFileID> | Disconnects open files by the specified file ID. The wildcard character (*) can be used with this parameter. Note: You can use the openfiles /query command to find the file ID. |
| /a <AccessedBy> | Disconnects all open files associated with the user name that is specified in the <i>AccessedBy</i> parameter. The wildcard character (*) can be used with this parameter. |
| /o {read write read/write} | Disconnects all open files with the specified open mode value. Valid values are Read, Write, or Read/Write. The wildcard character (*) can be used with this parameter. |

| PARAMETER | DESCRIPTION |
|----------------|--|
| /op <OpenFile> | Disconnects all open file connections that are created by a specific open file name. The wildcard character (*) can be used with this parameter. |
| /? | Displays help at the command prompt. |

Examples

To disconnect all open files with the file ID 26843578, type:

```
openfiles /disconnect /id 26843578
```

To disconnect all open files and directories accessed by the user "hiropln," type:

```
openfiles /disconnect /a hiropln
```

To disconnect all open files and directories with read/write mode, type:

```
openfiles /disconnect /o read/write
```

To disconnect the directory with the Open File name "C:\TestShare", regardless of who is accessing it, type:

```
openfiles /disconnect /a * /op "c:\testshare\"
```

To disconnect all open files on the remote computer "srvmain" that are being accessed by the user "hiropln," regardless of their ID, type:

```
openfiles /disconnect /s srvmain /u maindom\hiropln /id *
```

openfiles /query

Queries and displays all open files.

Syntax

```
openfiles /query [/s <System> [/u [<Domain>\]<UserName> [/p [<Password>]]]] [/fo {TABLE | LIST | CSV}] [/nh] [/v]
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------|---|
| /s <System> | Specifies the remote system to connect to (by name or IP address). Do not use backslashes. If you do not use the /s option, the command is executed on the local computer by default. This parameter applies to all files and folders that are specified in the command. |

| PARAMETER | DESCRIPTION |
|----------------------------|--|
| /u [<Domain>] | Executes the command by using the permissions of the specified user account. If you do not use the /u option, system permissions are used by default. |
| /p [<Password>] | Specifies the password of the user account that is specified in the /u option. If you do not use the /p option, a password prompt appears when the command is executed. |
| [/fo {TABLE LIST CSV}] | Displays the output in the specified format. Valid values for <i>Format</i> are: TABLE: Displays output in a table. LIST: Displays output in a list. CSV: Displays output in Comma Separated Values format. |
| /nh | Suppresses column header in the output. Valid only when the /fo parameter is set to TABLE or CSV . |
| /v | Specifies that detailed information be displayed in the output. |
| /? | Displays help at the command prompt. |

Examples

To query and display all open files, type:

```
openfiles /query
```

To query and display all open files in table format without headers, type:

```
openfiles /query /fo table /nh
```

To query and display all open files in list format with detailed information, type:

```
openfiles /query /fo list /v
```

To query and display all open files on the remote system "srvmain" by using the credentials for the user "hiropln" on the "maindom" domain, type:

```
openfiles /query /s srvmain /u maindom\hiropln /p p@ssw23
```

NOTE

In this example, the password is supplied on the command line. To prevent displaying the password, leave out the **/p** option. You will be prompted for the password, which will not be echoed to the screen.

openfiles /local

Enables or disables the system Maintain Objects List global flag. If used without parameters, **openfiles /local** displays the current status of the Maintain Objects List global flag.

Syntax

```
openfiles /local [on | off]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|--|
| [on off] | Enables or disables the system Maintain Objects List global flag, which tracks local file handles. |
| /? | Displays help at the command prompt. |

Remarks

- Enabling the Maintain Objects List global flag may slow down your system.
- Changes made by using the **on** or **off** option do not take effect until you restart the system.

Examples

To check the current status of the Maintain Objects List global flag, type:

```
openfiles /local
```

By default, the Maintain Objects List global flag is disabled, and the following output is displayed:

```
INFO: The system global flag 'maintain objects list' is currently disabled.
```

To enable the Maintain Objects List global flag, type:

```
openfiles /local on
```

The following message is displayed when the global flag is enabled:

```
SUCCESS: The system global flag 'maintain objects list' is enabled.  
This will take effect after the system is restarted.
```

To disable the Maintain Objects List global flag, type:

```
openfiles /local off
```

Additional references

[Command-Line Syntax Key](#)

pagefileconfig

11/6/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

pagefileconfig is deprecated, and is not guaranteed to be supported in future releases of Windows. This tool is included in Windows Server 2003. For more information see [pagefileconfig](#).

path

7/11/2018 • 2 minutes to read • [Edit Online](#)

Sets the command path in the PATH environment variable (the set of directories used to search for executable files). If used without parameters, **path** displays the current command path.

For examples of how to use this command, see [Examples](#).

Syntax

```
path [[<Drive>:]<Path>[;...][;%PATH%]]
path ;
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|--|
| [<Drive>:] | Specifies the drive and directory to set in the command path. |
| ; | Separates directories in the command path. If used without other parameters, ; clears the existing command paths from the PATH environment variable and directs Cmd.exe to search only in the current directory. |
| %PATH% | Appends the command path to the existing set of directories listed in the PATH environment variable. |
| /? | Displays help at the command prompt. |

Remarks

- When you include **%PATH%** in the syntax, Cmd.exe replaces it with the command path values found in the PATH environment variable, eliminating the need to manually enter these values at the command prompt.
- The current directory is always searched before the directories specified in the command path.
- You might have files in a directory that share the same file name but have different extensions. For example, you might have a file named Accnt.com that starts an accounting program and another file named Accnt.bat that connects your server to the accounting system network.

The Windows operating system searches for a file by using default file name extensions in the following order of precedence: .exe, .com, .bat, and .cmd. To run Accnt.bat when Accnt.com exists in the same directory, you must include the .bat extension at the command prompt.

- If two or more files in the command path have the same file name and extension, **path** first searches for the specified file name in the current directory. Then it searches the directories in the command path in the order that they are listed in the PATH environment variable.
- If you place the **path** command in your Autoexec.nt file, the Windows operating system automatically appends the specified MS-DOS subsystem search path every time you log on to your computer. Cmd.exe does not use the Autoexec.nt file. When started from a shortcut, Cmd.exe inherits the environment variables set in My Computer/Properties/Advanced/Environment.

Examples

To search the paths C:\User\Taxes, B:\User\Invest, and B:\Bin for external commands, type:

```
path c:\user\taxes;b:\user\invest;b:\bin
```

Additional references

[Command-Line Syntax Key](#)

pathping

7/11/2018 • 4 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Provides information about network latency and network loss at intermediate hops between a source and destination. **pathping** sends multiple echo Request messages to each router between a source and destination over a period of time and then computes results based on the packets returned from each router. Because **pathping** displays the degree of packet loss at any given router or link, you can determine which routers or subnets might be having network problems.

pathping performs the equivalent of the **tracert** command by identifying which routers are on the path. It then sends pings periodically to all of the routers over a specified time period and computes statistics based on the number returned from each. Used without parameters, **pathping** displays help.

Syntax

```
pathping [/n] [/h] [/g <Hostlist>] [/p <Period>] [/q <NumQueries>] [/w <timeout>] [/i <IPAddress>] [/4 <IPv4>] [/6 <IPv6>][<TargetName>]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------------|---|
| /n | Prevents pathping from attempting to resolve the IP addresses of intermediate routers to their names. This might expedite the display of pathping results. |
| /h <MaximumHops> | Specifies the maximum number of hops in the path to search for the target (destination). The default is 30 hops. |
| /g <Hostlist> | Specifies that the echo Request messages use the Loose Source Route option in the IP header with the set of intermediate destinations specified in <i>Hostlist</i> . With loose source routing, successive intermediate destinations can be separated by one or multiple routers. The maximum number of addresses or names in the host list is 9. The <i>Hostlist</i> is a series of IP addresses (in dotted decimal notation) separated by spaces. |
| /p <Period> | Specifies the number of milliseconds to wait between consecutive pings. The default is 250 milliseconds (1/4 second). |
| /q <NumQueries> | Specifies the number of echo Request messages sent to each router in the path. The default is 100 queries. |
| /w <timeout> | Specifies the number of milliseconds to wait for each reply. The default is 3000 milliseconds (3 seconds). |

| PARAMETER | DESCRIPTION |
|----------------|---|
| /i <IPAddress> | Specifies the source address. |
| /4 <IPv4> | Specifies that pathping uses IPv4 only. |
| /6 <IPv6> | Specifies that pathping uses IPv6 only. |
| <TargetName> | Specifies the destination, which is identified either by IP address or host name. |
| /? | Displays help at the command prompt. |

Remarks

- **pathping** parameters are case-sensitive.
- To avoid network congestion, pings should be sent at a sufficiently slow pace.
- To minimize the effects of burst losses, do not send pings too frequently.
- When using the **/p** parameter, pings are sent individually to each intermediate hop. Because of this, the interval between two pings sent to the same hop is *period* multiplied by the number of hops.
- When using the **/w** parameter, multiple pings can be sent in parallel. Because of this, the amount of time specified in the *timeout* parameter is not bounded by the amount of time specified in the *Period* parameter for waiting between pings.
- This command is available only if the Internet Protocol (TCP/IP) protocol is installed as a component in the properties of a network adapter in Network Connections.

Examples

The following example shows **pathping** command output:

```
D:\>pathping /n corp1
Tracing route to corp1 [10.54.1.196]
over a maximum of 30 hops:
 0  172.16.87.35
 1  172.16.87.218
 2  192.168.52.1
 3  192.168.80.1
 4  10.54.247.14
 5  10.54.1.196
computing statistics for 125 seconds...
      Source to Here   This Node/Link
Hop  RTT    Lost/Sent = Pct  Lost/Sent = Pct  address
 0                                0/ 100 = 0%    |
                                0/ 100 = 0%    172.16.87.218
 1   41ms    0/ 100 = 0%      13/ 100 = 13%    |
                                3/ 100 = 3%    192.168.52.1
 2   22ms   16/ 100 = 16%    0/ 100 = 0%     |
                                0/ 100 = 0%    192.168.80.1
 3   24ms   13/ 100 = 13%    0/ 100 = 0%     |
                                0/ 100 = 0%    10.54.247.14
 4   21ms   14/ 100 = 14%    1/ 100 = 1%     |
                                0/ 100 = 0%    10.54.1.196
 5   24ms   13/ 100 = 13%    0/ 100 = 0%
Trace complete.
```

When **pathping** is run, the first results list the path. This is the same path that is shown using the **tracert**

command. Next, a busy message is displayed for approximately 90 seconds (the time varies by hop count). During this time, information is gathered from all routers previously listed and from the links between them. at the end of this period, the test results are displayed.

In the sample report above, the **This Node/Link, Lost/Sent = Pct** and **address** columns show that the link between 172.16.87.218 and 192.168.52.1 is dropping 13 percent of the packets. The routers at hops 2 and 4 also are dropping packets addressed to them, but this loss does not affect their ability to forward traffic that is not addressed to them.

The loss rates displayed for the links, identified as a vertical bar (|) in the **address** column, indicate link congestion that is causing the loss of packets that are being forwarded on the path. The loss rates displayed for routers (identified by their IP addresses) indicate that these routers might be overloaded.

Additional references

- [Command-Line Syntax Key](#)

pause

4/13/2018 • 2 minutes to read • [Edit Online](#)

Suspends the processing of a batch program and displays the following prompt:

```
Press any key to continue . . .
```

For examples of how to use this command, see [Examples](#).

Syntax

```
pause
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------------|
| /? | Displays help at the command prompt. |

Remarks

- When you run the **pause** command, the following message appears:

```
Press any key to continue . . .
```

- If you press CTRL+C to stop a batch program, the following message appears:

```
Terminate batch job (Y/N)?
```

If you press Y (for yes) in response to this message, the batch program ends and control returns to the operating system.

- You can insert the **pause** command before a section of the batch file that you might not want to process. When **pause** suspends processing of the batch program, you can press CTRL+C and then press Y to stop the batch program.

Examples

To create a batch program that prompts the user to change disks in one of the drives, type:

```
@echo off
:Begin
copy a:*. *
echo Put a new disk into drive A
pause
goto begin
```

In this example, all the files on the disk in drive A are copied to the current directory. After the message prompts you to put a new disk in drive A, the **pause** command suspends processing so that you can change disks and then press any key to resume processing. This batch program runs in an endless loop—the **goto begin** command sends the command interpreter to the Begin label of the batch file. To stop this batch program, press CTRL+C and then press Y.

Additional references

[Command-Line Syntax Key](#)

pbadmin

4/13/2018 • 2 minutes to read • [Edit Online](#)

Pbadmin is deprecated, and is not guaranteed to be supported in future releases of Windows.

This tool is included in Windows Server 2003. For more information see [Pbadmin](#).

pentnt

4/13/2018 • 2 minutes to read • [Edit Online](#)

Pentnt is deprecated, and is not guaranteed to be supported in future releases of Windows.

This tool is included in Windows Server 2003. For more information see [Pentnt](#).

Start Windows Reliability and Performance Monitor in a specific standalone mode.

Syntax

```
perfmom </res|report|rel|sys>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| /res | Start Resource View. |
| /report | Start the System Diagnostics Data Collector Set and display a report of the results. |
| /rel | Start Reliability Monitor. |
| /sys | Start Performance Monitor. |

Additional references

[Windows Performance Monitor](#)

ping

7/11/2018 • 4 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

The **ping** command verifies IP-level connectivity to another TCP/IP computer by sending Internet Control Message Protocol (ICMP) echo Request messages. The receipt of corresponding echo Reply messages are displayed, along with round-trip times. ping is the primary TCP/IP command used to troubleshoot connectivity, reachability, and name resolution. Used without parameters, **ping** displays help.

Syntax

```
ping [/t] [/a] [/n <Count>] [/l <Size>] [/f] [/I <TTL>] [/v <TOS>] [/r <Count>] [/s <Count>] [{/j <Hostlist> | /k <Hostlist>}] [/w <timeout>] [/R] [/S <Srcaddr>] [/4] [/6] <TargetName>
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|---|
| /t | Specifies that ping continue sending echo Request messages to the destination until interrupted. To interrupt and display statistics, press CTRL+break. To interrupt and quit ping , press CTRL+C. |
| /a | Specifies that reverse name resolution is performed on the destination IP address. If this is successful, ping displays the corresponding host name. |
| /n <Count> | Specifies the number of echo Request messages sent. The default is 4. |
| /l <Size> | Specifies the length, in bytes, of the Data field in the echo Request messages sent. The default is 32. The maximum Size is 65,527. |
| /f | Specifies that echo Request messages are sent with the Do not Fragment flag in the IP header set to 1 (available on IPv4 only). The echo Request message cannot be fragmented by routers in the path to the destination. This parameter is useful for troubleshooting path Maximum Transmission Unit (PMTU) problems. |
| /I <TTL> | Specifies the value of the TTL field in the IP header for echo Request messages sent. The default is the default TTL value for the host. The maximum TTL is 255. |
| /v <TOS> | Specifies the value of the type of Service (TOS) field in the IP header for echo Request messages sent (available on IPv4 only). The default is 0. TOS is specified as a decimal value from 0 through 255. |

| PARAMETER | DESCRIPTION |
|---------------|---|
| /r <Count> | Specifies that the Record Route option in the IP header is used to record the path taken by the echo Request message and corresponding echo Reply message (available on IPv4 only). Each hop in the path uses an entry in the Record Route option. If possible, specify a <i>Count</i> that is equal to or greater than the number of hops between the source and destination. The <i>Count</i> must be a minimum of 1 and a maximum of 9. |
| /s <Count> | Specifies that the Internet timestamp option in the IP header is used to record the time of arrival for the echo Request message and corresponding echo Reply message for each hop. The <i>Count</i> must be a minimum of 1 and a maximum of 4. This is required for link-local destination addresses. |
| /j <Hostlist> | Specifies that the echo Request messages use the Loose Source Route option in the IP header with the set of intermediate destinations specified in <i>Hostlist</i> (available on IPv4 only). With loose source routing, successive intermediate destinations can be separated by one or multiple routers. The maximum number of addresses or names in the host list is 9. The host list is a series of IP addresses (in dotted decimal notation) separated by spaces. |
| /k <Hostlist> | Specifies that the echo Request messages use the Strict Source Route option in the IP header with the set of intermediate destinations specified in <i>Hostlist</i> (available on IPv4 only). With strict source routing, the next intermediate destination must be directly reachable (it must be a neighbor on an interface of the router). The maximum number of addresses or names in the host list is 9. The host list is a series of IP addresses (in dotted decimal notation) separated by spaces. |
| /w <timeout> | Specifies the amount of time, in milliseconds, to wait for the echo Reply message that corresponds to a given echo Request message to be received. If the echo Reply message is not received within the time-out, the "Request timed out" error message is displayed. The default time-out is 4000 (4 seconds). |
| /R | Specifies that the round-trip path is traced (available on IPv6 only). |
| /S <Srcaddr> | Specifies the source address to use (available on IPv6 only). |
| /4 | Specifies that IPv4 is used to ping. This parameter is not required to identify the target host with an IPv4 address. It is only required to identify the target host by name. |
| /6 | Specifies that IPv6 is used to ping. This parameter is not required to identify the target host with an IPv6 address. It is only required to identify the target host by name. |
| <TargetName> | Specifies the host name or IP address of the destination. |
| /? | Displays help at the command prompt. |

Remarks

- You can use **ping** to test both the computer name and the IP address of the computer. If pinging the IP address is successful, but pinging the computer name is not, you might have a name resolution problem. In this case, ensure that the computer name you are specifying can be resolved through the local Hosts file, by using Domain Name System (DNS) queries, or through NetBIOS name resolution techniques.
- This command is available only if the Internet Protocol (TCP/IP) protocol is installed as a component in the properties of a network adapter in Network Connections.

Examples

The following example shows **ping** command output:

```
C:\>ping example.microsoft.com
    pinging example.microsoft.com [192.168.239.132] with 32 bytes of data:
    Reply from 192.168.239.132: bytes=32 time=101ms TTL=124
    Reply from 192.168.239.132: bytes=32 time=100ms TTL=124
    Reply from 192.168.239.132: bytes=32 time=120ms TTL=124
    Reply from 192.168.239.132: bytes=32 time=120ms TTL=124
```

To ping the destination 10.0.99.221 and resolve 10.0.99.221 to its host name, type:

```
ping /a 10.0.99.221
```

To ping the destination 10.0.99.221 with 10 echo Request messages, each of which has a Data field of 1000 bytes, type:

```
ping /n 10 /l 1000 10.0.99.221
```

To ping the destination 10.0.99.221 and record the route for 4 hops, type:

```
ping /r 4 10.0.99.221
```

To ping the destination 10.0.99.221 and specify the loose source route of 10.12.0.1-10.29.3.1-10.1.44.1, type:

```
ping /j 10.12.0.1 10.29.3.1 10.1.44.1 10.0.99.221
```

additional references

- [Command-Line Syntax Key](#)

pnpunattend

7/11/2018 • 2 minutes to read • [Edit Online](#)

Audits a computer for device drivers, and perform unattended driver installations, or search for drivers without installing and, optionally, report the results to the command line. Use this command to specify the installation of specific drivers for specific hardware devices. See Remarks.

Syntax

```
PnPUnattend.exe auditSystem [/help] [/?] [/h] [/s] [/L]
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------|---|
| auditSystem | Specifies online driver install. Required, except when pnpunattend is run with either the /Help or /? parameters. |
| /s | Optional. Specifies to search for drivers without installing. |
| /L | Optional. Specifies to display the log information for this command in the command prompt. |
| /? | Optional. Displays help for this command at the command prompt. |

Remarks

Preliminary preparation is required. Prior to using this command, you must complete the following tasks:

1. Create a directory for the drivers you want to install. For example, create a folder at **C:\Drivers\Video** for video adapter drivers.
2. Download and extract the driver package for your device. Copy the contents of the subfolder that contains the INF file for your version of the operating system and any subfolders to the video folder that you created. For example, copy the video driver files to C:\Drivers\Video.
3. Add a system environment path variable to the folder you created in step 1. For example, **C:\Drivers\Video**.
4. Create the following registry key, and then for the **DriverPaths** key you create, set the **Value Data** to **1**.
5. For Windows® 7 navigate the Registry path: **HKEY_LOCAL_MACHINE\Software\Microsoft\Windows NT\CurrentVersion**, and then create the keys: **UnattendSettings\PnPUnattend\DriverPaths**
6. For Windows Vista, navigate to the Registry path: **HK_LM\Software\Microsoft\Windows NT\CurrentVersion**, and then create the keys = **\UnattendSettings\PnPUnattend\DriverPaths**.

Examples

The following example command shows how to use the **PNPUnattend.exe** to audit a computer for possible driver updates, and then report the findings to the command prompt.

```
pnpunattend auditsystem /s /l
```

Additional references

[Command-Line Syntax Key](#)

pnputil

7/11/2018 • 2 minutes to read • [Edit Online](#)

Pnputil.exe is a command line utility that you can use to manage the driver store. You can use Pnputil to add driver packages, remove driver packages, and list driver packages that are in the store.

Syntax

```
pnputil.exe [-f | -i] [ -? | -a | -d | -e ] <INF name>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| -a | Specifies to add the identified INF file. |
| -d | Specifies to delete the identified INF file. |
| -e | Specifies to enumerate all third-party INF files. |
| -f | Specifies to force the deletion of the identified INF file. Cannot be used in conjunction with the -i parameter. |
| -i | Specifies to install the identified INF file. Cannot be used in conjunction with the -f parameter. |
| /? | Displays help at the command prompt. |

Examples

- pnputil.exe -a a:\usbcam\USBCAM.INF Adds the INF file that is specified by USBCAM.INF
- pnputil.exe -a c:\drivers*.inf Adds all INF files in c:\drivers\
- pnputil.exe -i -a a:\usbcam\USBCAM.INF Adds and installs the specified driver.
- pnputil.exe -e Enumerates all third-party drivers.
- pnputil.exe -d oem0.inf Deletes the specified.
- pnputil.exe -f -d oem0.inf Forces the deletion of the specified INF file.

Additional references

[Command-Line Syntax Key](#)

[Popd](#)

popd

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Changes the current directory to the directory that was most recently stored by the **pushd** command. for examples of how to use this command, see [Examples](#).

Syntax

```
popd
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------------|
| /? | Displays help at the command prompt. |

Remarks

- Every time you use the **pushd** command, a single directory is stored for your use. However, you can store multiple directories by using the **pushd** command multiple times. The directories are stored sequentially in a virtual stack. If you use the **pushd** command once, the directory in which you use the command is placed at the bottom of the stack. If you use the command again, the second directory is placed on top of the first one. The process repeats every time you use the **pushd** command. You can use the **popd** command to change the current directory to the directory most recently stored by the **pushd** command. If you use the **popd** command, the directory on the top of the stack is removed from the stack and the current directory is changed to that directory. If you use the **popd** command again, the next directory on the stack is removed.
- When command extensions are enabled, the **popd** command removes any drive-letter assignments created by **pushd**.

Examples

The following example shows how you can use the **pushd** command and the **popd** command in a batch program to change the current directory from the one in which the batch program was run and then change it back:

```
@echo off
rem This batch file deletes all .txt files in a specified directory
pushd %1
del *.txt
popd
cls
echo All text files deleted in the %1 directory
```

additional references

- [pushd](#)

- [Command-Line Syntax Key](#)

PowerShell

7/11/2018 • 2 minutes to read • [Edit Online](#)

Windows PowerShell is a task-based command-line shell and scripting language designed especially for system administration. Built on the .NET Framework, Windows PowerShell helps IT professionals and power users control and automate the administration of the Windows operating system and applications that run on Windows.

The **PowerShell.exe** command-line tool starts a Windows PowerShell session in a Command Prompt window. When you use **PowerShell.exe**, you can use its optional parameters to customize the session. For example, you can start a session that uses a particular execution policy or one that excludes a Windows PowerShell profile. Otherwise, the session is the same as any session that is started in the Windows PowerShell console.

Using PowerShell.exe

You can use the **PowerShell.exe** command-line tool to start a Windows PowerShell session in a Command Prompt window.

- To start a Windows PowerShell session in a command prompt window, type `PowerShell`. A **PS** prefix is added to the command prompt to indicate that you are in a Windows PowerShell session.
- To start a session with a particular execution policy, use the **ExecutionPolicy** parameter.
`PowerShell.exe -ExecutionPolicy Restricted`
- To start a Windows PowerShell session without your Windows PowerShell profiles, use the **NoProfile** parameter.
`PowerShell.exe -NoProfile`
- To start a session, use the **ExecutionPolicy** parameter.
`PowerShell.exe -ExecutionPolicy Restricted`
- To see the PowerShell.exe help file, use the following command format.
`PowerShell.exe -help, -?, /?`
- To end a Windows PowerShell session in a Command Prompt window, type `exit`. The typical command prompt returns.

For a complete list of the **PowerShell.exe** command-line parameters, see [about_PowerShell.Exe](#).

Other Ways to Start Windows PowerShell

For information about other ways to start Windows PowerShell, see [Starting Windows PowerShell](#).

Remarks

Windows PowerShell runs on the Server Core installation option of Windows Server operating systems. However, features that require a graphic user interface, such as the [Windows PowerShell Integrated Scripting Environment \(ISE\)](#), and the [Out-GridView](#) and [Show-Command](#) cmdlets, do not run on Server Core installations.

Additional references

[about_PowerShell.Exe](#) [about_PowerShell_Ise.exe](#) [Windows PowerShell Scripting with Windows PowerShell](#) See Also

PowerShell_ise

7/11/2018 • 2 minutes to read • [Edit Online](#)

Windows PowerShell Integrated Scripting Environment (ISE) is a graphical host application that enables you to read, write, run, debug, and test scripts and modules in a graphic-assisted environment. Key features such as IntelliSense, Show-Command, snippets, tab completion, syntax-coloring, visual debugging, and context-sensitive Help provide a rich scripting experience.

The **PowerShell_ISE.exe** tool starts a Windows PowerShell ISE session. When you use **PowerShell_ISE.exe**, you can use its optional parameters to open files in Windows PowerShell ISE or to start a Windows PowerShell ISE session with no profile or with a multithreaded apartment.

PowerShell_ISE.exe was introduced in Windows PowerShell 2.0 and expanded significantly in Windows PowerShell 3.0.

Using PowerShell_ISE.exe

You can use **PowerShell_ISE.exe** to start and end a Windows PowerShell session as follows:

- To start a Windows PowerShell ISE session, in a Command Prompt window, in Windows PowerShell, or at the Start menu, type:

```
PowerShell_Ise
```

- To open a script (.ps1), script module (.psm1), module manifest (.psd1), XML file, or any other supported file in Windows PowerShell ISE, use the following command format:

```
PowerShell_Ise <FilePath>
```

In Windows PowerShell 3.0, you can use the optional **File** parameter as follows:

```
PowerShell_Ise -File <FilePath>
```

- To start a Windows PowerShell ISE session without your Windows PowerShell profiles, use the **NoProfile** parameter. (The **NoProfile** parameter is introduced in Windows PowerShell 3.0.)

```
PowerShell_Ise -NoProfile
```

- To see the **PowerShell_ISE.exe** Help file in a Command Prompt window, use the following command format:

```
PowerShell_Ise -help, -?, /?
```

For a complete list of the **PowerShell_ISE.exe** command-line parameters, see [about_PowerShell_Ise.exe](#).

Start Windows PowerShell ISE in other ways

For information about other ways to start Windows PowerShell ISE, see [Starting Windows PowerShell](#).

Remarks

Windows PowerShell runs on the Server Core installation option of Windows Server operating systems. However, because Windows PowerShell ISE requires a graphic user interface, it does not run on Server Core installations.

Additional references

[about_PowerShell_Ise.exe](#) [about_PowerShell.exe](#) [Windows PowerShell Scripting with Windows PowerShell](#) See Also

print

4/13/2018 • 2 minutes to read • [Edit Online](#)

Sends a text file to a printer.

For examples of how to use this command, see [Examples](#).

Syntax

```
Print [/d:<PrinterName>] [<Drive>:][<Path><FileName>[ ...]]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------------|---|
| /d:<PrinterName> | Specifies the printer that you want to print the job. To print to a locally connected printer, specify the port on your computer where the printer is connected. <ul style="list-style-type: none">- Valid values for parallel ports are LPT1, LPT2, and LPT3.- Valid values for serial ports are COM1, COM2, COM3, and COM4. You can also specify a network printer by using its queue name (\\ServerName*PrinterName*). If you do not specify a printer, the print job is sent to LPT1 by default. |
| <Drive>: | Specifies the logical or physical drive where the file you want to print is located. This parameter is not required if the file you want to print is located on the current drive. |
| <Path> | Specifies the location of the file you want to print. This parameter is not required if the file you want to print is located in the current directory. |
| <FileName>[...] | Required. Specifies the file you want to print. You can include multiple files in one command. |
| /? | Displays help at the command prompt. |

Remarks

- A file can print in the background if you send it to a printer connected to a serial or parallel port on the local computer.
- You can perform many configuration tasks from the command prompt by using the **Mode** command.

See [Mode](#) for more information about:

- Configuring a printer connected to a parallel port
- Configuring a printer connected to a serial port
- Displaying the status of a printer
- Preparing a printer for code page switching

Examples

To send the file Report.txt in the current directory to a printer connected to LPT2 on the local computer, type:

```
print /d:lpt2 report.txt
```

To send the file Report.txt in the c:\Accounting directory to the Printer1 print queue on the \\CopyRoom server, type:

```
print /d:\\copyroom\printer1 c:\accounting\report.txt
```

Additional references

[Command-Line Syntax Key](#)

[Print Command Reference](#)

[Mode](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Configures or displays configuration information about a printer.

Syntax

```
cscript Prncnfg {-g | -t | -x | -?} [-S <ServerName>] [-P <printerName>] [-z <NewprinterName>] [-u <UserName>] [-w <Password>] [-r <PortName>] [-l <Location>] [-h <Sharename>] [-m <Comment>] [-f <SeparatorFileName>] [-y <Datatype>] [-st <starttime>] [-ut <Untiltime>] [-i <DefaultPriority>] [-o <Priority>] [<+>|->shared] [<+>|->direct] [<+>|->hidden] [<+>|->published] [<+>|->rawonly] [<+>|->queued] [<+>|->enablebidi] [<+>|->keepprintedjobs] [<+>|->workoffline] [<+>|->enabledevq] [<+>|->docompletefirst]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------------------|---|
| -g | Displays configuration information about a printer. |
| -t | Configures a printer. |
| -x | renames a printer. |
| -S <ServerName> | Specifies the name of the remote computer that hosts the printer that you want to manage. If you do not specify a computer, the local computer is used. |
| -P <printerName> | Specifies the name of the printer that you want to manage. Required. |
| -z <NewprinterName> | Specifies the new printer name. Requires the -x and -P parameters. |
| -u <UserName> -w <Password> | Specifies an account with permissions to connect to the computer that hosts the printer that you want to manage. All members of the target computer's local Administrators group have these permissions, but the permissions can also be granted to other users. If you do not specify an account, you must be logged on under an account with these permissions for the command to work. |
| -r <PortName> | Specifies the port where the printer is connected. If this is a parallel or a serial port, then use the ID of the port (for example, LPT1 or COM1). If this is a TCP/IP port, use the port name that was specified when the port was added. |
| -l <Location> | Specifies the printer location, such as "copy Room." |

| PARAMETER | DESCRIPTION |
|------------------------|---|
| -h <Sharename> | Specifies the printer's share name. |
| -m <Comment> | Specifies the printer's comment string. |
| -f <SeparatorFileName> | Specifies a file that contains the text that appears on the separator page. |
| -y <Datatype> | Specifies the data types that the printer can accept. |
| -st <starttime> | Configures the printer for limited availability. Specifies the time of day the printer is available. If you send a document to a printer when it is unavailable, the document is held (spooled) until the printer becomes available. You must specify time as a 24-hour clock. For example, to specify 11:00 P.M., type 2300 . |
| -ut <Endtime> | Configures the printer for limited availability. Specifies the time of day the printer is no longer available. If you send a document to a printer when it is unavailable, the document is held (spooled) until the printer becomes available. You must specify time as a 24-hour clock. For example, to specify 11:00 P.M., type 2300 . |
| -o <Priority> | Specifies a priority that the spooler uses to route print jobs into the print queue. A print queue with a higher priority receives all its jobs before any queue with a lower priority. |
| -i <DefaultPriority> | Specifies the default priority assigned to each print job. |
| {+ -}shared | Specifies whether this printer is shared on the network. |
| {+ -}direct | Specifies whether the document should be sent directly to the printer without being spooled. |
| {+ -}published | Specifies whether this printer should be published in active directory. If you publish the printer, other users can search for it based on its location and capabilities (such as color printing and stapling). |
| {+ -}hidden | Reserved function. |
| {+ -}rawonly | Specifies whether only raw data print jobs can be spooled in this queue. |
| {+ -}queued | Specifies that the printer should not begin to print until after the last page of the document is spooled. The printing program is unavailable until the document has finished printing. However, using this parameter ensures that the whole document is available to the printer. |
| {+ -}keepprintedjobs | Specifies whether the spooler should retain documents after they are printed. Enabling this option allows a user to resubmit a document to the printer from the print queue instead of from the printing program. |

| PARAMETER | DESCRIPTION |
|------------------------|--|
| {+ -}workoffline | Specifies whether a user is able to send print jobs to the print queue if the computer is not connected to the network. |
| {+ -}enabledevq | Specifies whether print jobs that do not match the printer setup (for example, PostScript files spooled to non-PostScript printers) should be held in the queue rather than being printed. |
| {+ -}docompletefirst | Specifies whether the spooler should send print jobs with a lower priority that have completed spooling before sending print jobs with a higher priority that have not completed spooling. If this option is enabled and no documents have completed spooling, the spooler will send larger documents before smaller ones. You should enable this option if you want to maximize printer efficiency at the cost of job priority. If this option is disabled, the spooler always sends higher priority jobs to their respective queues first. |
| {+ -}enablebidi | Specifies whether the printer sends status information to the spooler. |
| /? | Displays help at the command prompt. |

Remarks

- The **prncnfg** command is a Visual Basic script located in the %WINDir%\System32\printing_Admin_Scripts\ directory. To use this command, at a command prompt, type **cscript** followed by the full path to the prncnfg file, or change directories to the appropriate folder. For example:

```
cscript %WINDir%\System32\printing_Admin_Scripts\en-US\prncnfg
```

- if the information that you supply contains spaces, use quotation marks around the text (for example, "computer Name").

Examples

To display configuration information for the printer named colorprinter_2 with a print queue hosted by the remote computer named HRServer, type:

```
cscript prncnfg -g -S HRServer -P colorprinter_2
```

To configure a printer named colorprinter_2 so that the spooler in the remote computer named HRServer keeps print jobs after they have been printed, type:

```
cscript prncnfg -t -S HRServer -P colorprinter_2 +keepprintedjobs
```

To change the name of a printer on the remote computer named HRServer from colorprinter_2 to colorprinter 3, type:

```
cscript prncnfg -x -S HRServer -P colorprinter_2 -z "colorprinter 3"
```

additional references

[Command-Line Syntax Key print Command Reference](#)

prndrvr

7/11/2018 • 3 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Use the **prndrvr** command to add, delete, and list printer drivers.

Syntax

```
cscript prndrvr {-a | -d | -l | -x | -?} [-m <modelName>] [-v {0|1|2|3}]  
[-e <environment>] [-s <ServerName>] [-u <UserName>] [-w <Password>]  
[-h <path>] [-i <inf file>]
```

Parameters

| PARAMETER | DESCRIPTION |
|----------------------|---|
| -a | Installs a driver. |
| -d | deletes a driver. |
| -l | lists all printer drivers installed on the server specified by the -s parameter. If you do not specify a server, Windows lists the printer drivers installed on the local computer. |
| -x | deletes all printer drivers and additional printer drivers not in use by a logical printer on the server specified by the -s parameter. If you do not specify a server to remove from the list, Windows deletes all unused printer drivers on the local computer. |
| -m <DrivermodelName> | Specifies (by name) the driver you want to install. Drivers are often named for the model of printer they support. See the printer documentation for more information. |
| -v {0 1 2 3} | <p>Specifies the version of the driver you want to install. See the description of the -e parameter for information on which versions are available for which environment. If you do not specify a version, the version of the driver appropriate for the version of Windows running on the computer where you are installing the driver is installed.</p> <ul style="list-style-type: none">- version 0 supports Windows 95, Windows 98, and Windows Millennium edition.- version 1 supports Windows NT 3.51.- version 2 supports Windows NT 4.0.- version 3 supports Windows Vista, Windows XP, Windows 2000, and the Windows Server 2003 operating systems. Note that this is the only printer driver version that Windows Vista supports. |

| PARAMETER | DESCRIPTION |
|-----------------------------|---|
| -e <Environment> | <p>Specifies the environment for the driver you want to install. If you do not specify an environment, the environment of the computer where you are installing the driver is used. The supported environment parameters are:</p> <ul style="list-style-type: none"> - "Windows NT x86" - "Windows x64" - "Windows IA64" |
| -s <ServerName> | Specifies the name of the remote computer that hosts the printer that you want to manage. If you do not specify a computer, the local computer is used. |
| -u <UserName> -w <Password> | Specifies an account with permissions to connect to the computer that hosts the printer that you want to manage. All members of the target computer's local Administrators group have these permissions, but the permissions can also be granted to other users. If you do not specify an account, you must be logged on under an account with these permissions for the command to work. |
| -h <path> | Specifies the path to the driver file. If you do not specify a path, the path to the location where Windows was installed is used. |
| -i <Filename.inf> | <p>Specifies the complete path and file name for the driver you want to install. If you do not specify a file name, the script uses one of the inbox printer .inf files in the inf subdirectory of the Windows directory.</p> <p>if the driver path is not specified, the script searches for driver files in the driver.cab file.</p> |
| /? | Displays help at the command prompt. |

Remarks

- The **pnldr** command is a Visual Basic script located in the %WINDir%\System32\printing_Admin_Scripts\ directory. To use this command, at a command prompt, type **cscript** followed by the full path to the pnldr file, or change directories to the appropriate folder.

For example:

```
cscript %WINDir%\System32\printing_Admin_Scripts\en-US\pnldr
```

- If the information that you supply contains spaces, use quotation marks around the text (for example, "computer Name").
- The -x option deletes all additional printer drivers (drivers installed for use on clients running alternate versions of Windows), even if the primary driver is in use. If the fax component is installed, this option also deletes fax drivers. The primary fax driver is deleted if it is not in use (that is, if there is no queue using it). If the primary fax driver is deleted, the only way to re-enable fax is to reinstall the fax component.
- Used without parameters, **pnldr** displays command-line help for the **pnldr** command.

Examples

To list all drivers on the \\printServer1 server, type:

```
cscript Prndrvr -l -s
```

To add a version 3 Windows x64 printer driver for the "Laser printer model 1" model of printer using the C:\temp\Laserprinter1.inf driver information file for a driver stored in the C:\temp folder type:

```
cscript Prndrvr -a -m "Laser printer model 1" -v 3 -e "Windows x64" -i c:\temp\Laserprinter1.inf -h c:\temp
```

To delete a version 3 Windows NT x86 printer driver for "Laser printer model 1", type:

```
cscript Prndrvr -a -m "Laser printer model 1" -v 3 -e "Windows NT x86"
```

additional references

[Command-Line Syntax Key print Command Reference](#)

prnjobs

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Pauses, resumes, cancels, and lists print jobs.

Syntax

```
cscript Prnjobs {-z | -m | -x | -l | -?} [-s <ServerName>]
[-p <printerName>] [-j <JobID>] [-u <UserName>] [-w <Password>]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------------|---|
| -z | pauses the print job specified with the -j parameter. |
| -m | Resumes the print job specified with the -j parameter. |
| -x | Cancels the print job specified with the -j parameter. |
| -l | lists all the print jobs in a print queue. |
| -s <ServerName> | Specifies the name of the remote computer that hosts the printer that you want to manage. If you do not specify a computer, the local computer is used. |
| -p <printerName> | Specifies the name of the printer that you want to manage. Required. |
| -j <JobID> | Specifies (by ID number) the print job you want to cancel. |
| -u <UserName> -w | Specifies an account with permissions to connect to the computer that hosts the printer that you want to manage. All members of the target computer's local Administrators group have these permissions, but the permissions can also be granted to other users. If you do not specify an account, you must be logged on under an account with these permissions for the command to work. |
| /? | Displays help at the command prompt. |

Remarks

- The **prnjobs** command is a Visual Basic script located in the %WINDIR%\System32\printing_Admin_Scripts\ directory. To use this command, at a command prompt, type **cscript** followed by the full path to the prnjobs file, or change directories to the appropriate folder. For example:

```
cscript %WINDir%\System32\printing_Admin_Scripts\en-US\prnjobs
```

- If the information that you supply contains spaces, use quotation marks around the text (for example, "computer Name").

Examples

To pause a print job with a job ID of 27 sent to the remote computer named HRServer for printing on the printer named colorprinter, type:

```
cscript prnjobs -z -s HRServer -p colorprinter -j 27
```

To list all current print jobs in the queue for the local printer named colorprinter_2, type:

```
cscript prnjobs -l -p colorprinter_2
```

Additional references

[Command-Line Syntax Key print Command Reference](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Adds, deletes, and lists printers or printer connections, in addition to setting and displaying the default printer.

Syntax

```
cscript Prnmngr {-a | -d | -x | -g | -t | -l | -?}[c] [-s <ServerName>]  
[-p <printerName>] [-m <printermodel>] [-r <PortName>] [-u <UserName>]  
[-w <Password>]
```

Parameters

| PARAMETER | DESCRIPTION |
|----------------------|--|
| -a | adds a local printer connection. |
| -d | deletes a printer connection. |
| -x | deletes all printers from the server specified with the -s parameter. If you do not specify a server, Windows deletes all printers on the local computer. |
| -g | Displays the default printer. |
| -t | Sets the default printer to the printer specified by the -p parameter. |
| -l | lists all printers installed on the server specified by the -s parameter. If you do not specify a server, Windows lists the printers installed on the local computer. |
| c | Specifies that the parameter applies to printer connections. Can be used with the -a and -x parameters. |
| -s | Specifies the name of the remote computer that hosts the printer that you want to manage. If you do not specify a computer, the local computer is used. |
| -p <printerName> | Specifies the name of the printer that you want to manage. |
| -m <DrivermodelName> | Specifies (by name) the driver you want to install. Drivers are often named for the model of printer they support. See the printer documentation for more information. |

| PARAMETER | DESCRIPTION |
|-----------------------------|---|
| -r <PortName> | Specifies the port where the printer is connected. If this is a parallel or a serial port, use the ID of the port (for example, LPT1: or COM1:). If this is a TCP/IP port, use the port name that was specified when the port was added. |
| -u <UserName> -w <Password> | Specifies an account with permissions to connect to the computer that hosts the printer that you want to manage. All members of the target computer's local Administrators group have these permissions, but the permissions can also be granted to other users. If you do not specify an account, you must be logged on under an account with these permissions for the command to work. |
| /? | Displays help at the command prompt. |

Remarks

- The **pnmndrvr** command is a Visual Basic script located in the %WINDir%\System32\printing_Admin_Scripts\ directory. To use this command, at a command prompt, type **cscript** followed by the full path to the **prnmngr** file, or change directories to the appropriate folder. For example:

```
cscript %WINDir%\System32\printing_Admin_Scripts\en-US\prnmngr
```

- if the information that you supply contains spaces, use quotation marks around the text (for example, "computer Name").

Examples

To add a printer named colorprinter_2 that is connected to LPT1 on the local computer and requires a printer driver called color printer Driver1, type:

```
cscript prnmngr -a -p colorprinter_2 -m "color printer Driver1" -r lpt1:
```

To delete the printer named colorprinter_2 from the remote computer named HRServer, type:

```
cscript prnmngr -d -s HRServer -p colorprinter_2
```

additional references

[Command-Line Syntax Key print Command Reference](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Creates, deletes, and lists standard TCP/IP printer ports, in addition to displaying and changing port configuration.

Syntax

```
cscript prnport {-a | -d | -l | -g | -t | -?} [-r <PortName>]
[-s <ServerName>] [-u <UserName>] [-w <Password>] [-o {raw | lpr}]
[-h <Hostaddress>] [-q <QueueName>] [-n <PortNumber>] -m{e | d}
[-i <SNMPIndex>] [-y <CommunityName>] -2{e | -d}
```

Parameters

| PARAMETER | DESCRIPTION |
|------------------|---|
| -a | creates a standard TCP/IP printer port. |
| -d | deletes a standard TCP/IP printer port. |
| -l | lists all standard TCP/IP printer ports on the computer specified with the -s parameter. |
| -g | Displays the configuration of a standard TCP/IP printer port. |
| -t | Configures the port settings for a standard TCP/IP printer port. |
| -r <PortName> | Specifies the port to which the printer is connected. |
| -s <ServerName> | Specifies the name of the remote computer that hosts the printer that you want to manage. If you do not specify a computer, the local computer is used. |
| -u <UserName> -w | Specifies an account with permissions to connect to the computer that hosts the printer that you want to manage. All members of the target computer's local Administrators group have these permissions, but the permissions can also be granted to other users. If you do not specify an account, you must be logged on under an account with these permissions for the command to work. |
| -o {raw lpr} | Specifies which protocol the port uses: TCP raw or TCP lpr. If you use TCP raw, you can optionally specify the port number by using the -n parameter. The default port number is 9100. |

| PARAMETER | DESCRIPTION |
|--------------------|--|
| -h <Hostaddress> | Specifies (by IP address) the printer for which you want to configure the port. |
| -q <QueueName> | Specifies the queue name for a TCP raw port. |
| -n <PortNumber> | Specifies the port number for a TCP raw port. The default port number is 9100. |
| -m{e d} | Specifies whether SNMP is enabled. The parameter e enables SNMP. The parameter d disables SNMP. |
| -i <SNMPIndex> | Specifies the SNMP index, if SNMP is enabled. For more information, see Rfc 1759 at the Rfc editor Web site . |
| -y <CommunityName> | Specifies the SNMP community name, if SNMP is enabled. |
| -2{e -d} | Specifies whether double spools (also known as respooling) are enabled for TCP lpr ports. Double spools are necessary because TCP lpr must include an accurate byte count in the control file that is sent to the printer, but the protocol cannot get the count from the local print provider. Therefore, when a file is spooled to a TCP lpr print queue, it is also spooled as a temporary file in the system32 directory. TCP lpr determines the size of the temporary file and sends the size to the server running LPD. The parameter e enables double spools. The parameter d disables double spools. |
| /? | Displays help at the command prompt. |

Remarks

- The **prnport** command is a Visual Basic script located in the %WINDir%\System32\printing_Admin_Scripts\ directory. To use this command, at a command prompt, type **cscript** followed by the full path to the prnport file, or change directories to the appropriate folder. For example:

```
cscript %WINDir%\System32\printing_Admin_Scripts\en-US\prnport
```

- if the information that you supply contains spaces, use quotation marks around the text (for example, "computer Name").
- The TCP raw protocol is a higher performance protocol on Windows than the lpr protocol.

Examples

To display all standard TCP/IP printing ports on the server \\Server1, type:

```
cscript prnport -l -s Server1
```

To delete the standard TCP/IP printing port on the server \\Server1 that connects to a network printer at 10.2.3.4, type:

```
cscript prnport -d -s Server1 -r IP_10.2.3.4
```

To add a standard TCP/IP printing port on the server \\Server1 that connects to a network printer at 10.2.3.4 and

uses the TCP raw protocol on port 9100, type:

```
cscript prnport -a -s Server1 -r IP_10.2.3.4 -h 10.2.3.4 -o raw -n 9100
```

To enable SNMP, specify the "public" community name and set the SNMP index to 1 on a network printer at 10.2.3.4 shared by the server \\Server1, type:

```
cscript prnport -t -s Server1 -r IP_10.2.3.4 -me -y public -i 1 -n 9100
```

To add a standard TCP/IP printing port on the local computer that connects to a network printer at 10.2.3.4 and automatically get the device settings from the printer, type:

```
cscript prnport -a -r IP_10.2.3.4 -h 10.2.3.4
```

additional references

[Command-Line Syntax Key print Command Reference](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Prints a test page, pauses or resumes a printer, and clears a printer queue.

Syntax

```
cscript Prnqctl {-z | -m | -e | -x | -?} [-s <ServerName>]  
[-p <printerName>] [-u <UserName>] [-w <Password>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------------------|---|
| -z | pauses printing on the printer specified with the -p parameter. |
| -m | Resumes printing on the printer specified with the -p parameter. |
| -e | prints a test page on the printer specified with the -p parameter. |
| -x | Cancels all print jobs on the printer specified with the -p parameter. |
| -s <ServerName> | Specifies the name of the remote computer that hosts the printer that you want to manage. If you do not specify a computer, the local computer is used. |
| -p <printerName> | Specifies the name of the printer that you want to manage. Required. |
| -u <UserName> -w <Password> | Specifies an account with permissions to connect to the computer that hosts the printer that you want to manage. All members of the target computer's local Administrators group have these permissions, but the permissions can also be granted to other users. If you do not specify an account, you must be logged on under an account with these permissions for the command to work. |
| /? | Displays help at the command prompt. |

Remarks

- The **prnqctl** command is a Visual Basic script located in the %WINDir%\System32\printing_Admin_Scripts\ directory. To use this command, at a command prompt, type **cscript** followed by the full path to the prnqctl file, or change directories to the appropriate folder. For example:

```
cscript %WINDir%\System32\printing_Admin_Scripts\en-US\prnqctl
```

- if the information that you supply contains spaces, use quotation marks around the text (for example, "computer Name").

Examples

To print a test page on the Laserprinter1 printer shared by the \\Server1 computer, type:

```
cscript Prnqctl -e -s Server1 -p Laserprinter1
```

To pause printing on the Laserprinter1 printer on the local computer, type:

```
cscript Prnqctl -z -p Laserprinter1
```

To cancel all print jobs on the Laserprinter1 printer on the local computer, type:

```
cscript Prnqctl -x -p Laserprinter1
```

additional references

[Command-Line Syntax Key](#)
[print Command Reference](#)

prompt

7/11/2018 • 2 minutes to read • [Edit Online](#)

Changes the Cmd.exe command prompt. If used without parameters, **prompt** resets the command prompt to the default setting, which is the current drive letter and directory followed by the greater than symbol (>).

For examples of how to use this command, see [Examples](#).

Syntax

```
prompt [<Text>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| <Text> | Specifies the text and information that you want to include in the command prompt. |
| /? | Displays help at the command prompt. |

Remarks

You can customize the command prompt to display any text you want, including such information as the name of the current directory, the time and date, and the Microsoft Windows version number.

The following table lists the character combinations that you can include instead of, or in addition to, one or more character strings in the *Text* parameter. The list includes a brief description of the text or information that each character combination adds to your command prompt.

| CHARACTER | DESCRIPTION |
|-----------|------------------------|
| \$q | = (equal sign) |
| \$\$ | \$ (dollar sign) |
| \$t | Current time |
| \$d | Current date |
| \$p | Current drive and path |
| \$v | Windows version number |
| \$n | Current drive |
| \$g | > (greater than sign) |

| CHARACTER | DESCRIPTION |
|-----------|---|
| \$l | < (less than sign) |
| \$b | |
| \$_ | ENTER-LINEFEED |
| \$e | ANSI escape code (code 27) |
| \$h | Backspace (to delete a character that has been written to the command line) |
| \$a | & (ampersand) |
| \$c | ((left parenthesis) |
| \$f |) (right parenthesis) |
| \$s | space |

When command extensions are enabled (that is, the default) the **prompt** command supports the following formatting characters:

| CHARACTER | DESCRIPTION |
|-----------|--|
| \$+ | Zero or more plus sign (+) characters, depending on the depth of the pushd directory stack (one character for each level pushed). |
| \$m | The remote name associated with the current drive letter or the empty string if current drive is not a network drive. |

If you include the **\$p** character in the text parameter, your disk is read after you enter each command (to determine the current drive and path). This can take extra time, especially for floppy disk drives.

Examples

To set a two-line command prompt with the current time and date on the first line and the greater than sign on the next line, type:

```
prompt $d$s$s$t$_$g
```

The prompt is changed as follows, where the date and time are current:

```
Fri 06/01/2007 13:53:28.91
>
```

To set the command prompt to display as an arrow (`-->`), type:

```
prompt --$g
```

To manually change the command prompt to the default setting (the current drive and path followed by the greater than sign), type:

```
prompt $p$g
```

Additional references

[Command-Line Syntax Key](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Publishes a printer to the active directory Domain Services.

Syntax

```
cscript pubprn {<ServerName> | <UNCprinterpath>}  
"LDAP://CN=<Container>,DC=<Container>"
```

Parameters

| PARAMETER | DESCRIPTION |
|------------------|---|
| <ServerName> | Specifies the name of the Windows server that hosts the printer that you want to publish. If you do not specify a computer, the local computer is used. |
| <UNCprinterpath> | The Universal Naming Convention (UNC) path to the shared printer that you want to publish. |
| "LDAP://CN=,DC=" | Specifies the path to the container in active directory Domain Services where you want to publish the printer. |
| /? | Displays help at the command prompt. |

Remarks

- The **pubprn** command is a Visual Basic script located in the %WINDir%\System32\printing_Admin_Scripts\ directory. To use this command, at a command prompt, type **cscript** followed by the full path to the pubprn file, or change directories to the appropriate folder. For example:

```
cscript %WINDir%\System32\printing_Admin_Scripts\en-US\pubprn
```

- if the information that you supply contains spaces, use quotation marks around the text (for example, "computer Name").

Examples

To publish all printers on the \\Server1 computer to the MyContainer container in the MyDomain.company.Com domain, type:

```
cscript Ppubprn Server1 "LDAP://CN=MyContainer,DC=MyDomain,DC=company,DC=Com"
```

To publish the Laserprinter1 printer on the \\Server1 server to the MyContainer container in the MyDomain.company.Com domain, type:

```
cscript Ppubprn \\Server1\Laserprinter1 "LDAP://CN=MyContainer,DC=MyDomain,DC=company,DC=Com"
```

additional references

[Command-Line Syntax Key print Command Reference](#)

pushd

4/13/2018 • 2 minutes to read • [Edit Online](#)

Stores the current directory for use by the **popd** command, and then changes to the specified directory.

For examples of how to use this command, see [Examples](#).

Syntax

```
pushd [<Path>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| <Path> | Specifies the directory to make the current directory. This command supports relative paths. |
| /? | Displays help at the command prompt. |

Remarks

- Every time you use the **pushd** command, a single directory is stored for your use. However, you can store multiple directories by using the **pushd** command multiple times.

The directories are stored sequentially in a virtual stack. If you use the **pushd** command once, the directory in which you use the command is placed at the bottom of the stack. If you use the command again, the second directory is placed on top of the first one. The process repeats every time you use the **pushd** command.

You can use the **popd** command to change the current directory to the directory most recently stored by the **pushd** command. If you use the **popd** command, the directory on the top of the stack is removed from the stack and the current directory is changed to that directory. If you use the **popd** command again, the next directory on the stack is removed.

- If command extensions are enabled, the **pushd** command accepts either a network path or a local drive letter and path.
- If you specify a network path, the **pushd** command temporarily assigns the highest unused drive letter (starting with Z:) to the specified network resource. The command then changes the current drive and directory to the specified directory on the newly assigned drive. If you use the **popd** command with command extensions enabled, the **popd** command removes the drive-letter assignment created by **pushd**.

Examples

The following example shows how you can use the **pushd** command and the **popd** command in a batch program to change the current directory from the one in which the batch program was run and then change it back:

```
@echo off
rem This batch file deletes all .txt files in a specified directory
pushd %1
del *.txt
popd
cls
echo All text files deleted in the %1 directory
```

Additional references

[Command-Line Syntax Key](#)

[Popd](#)

pushprinterconnections

7/11/2018 • 2 minutes to read • [Edit Online](#)

Reads Deployed Printer Connection settings from Group Policy and deploys/removes printer connections as needed.

Syntax

```
pushprinterconnections <-log> <-?>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| <-log> | Writes a per user debug log file to %temp, or writes a per machine debug log to %windir%\temp. |
| <-?> | Displays Help at the command prompt. |

Remarks

This utility is for use in machine startup or user logon scripts, and should not be run from the command line.

Additional references

- [Command-Line Syntax Key](#)
- [Deploy Printers by Using Group Policy](#)

qappsrv

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Displays a list of all Remote Desktop Session Host (RD Session Host) servers on the network.

NOTE

In Windows Server 2008 R2, Terminal Services was renamed Remote Desktop Services. To find out what's new in the latest version, see [What's New in Remote Desktop Services in Windows Server 2012](#) in the Windows Server TechNet Library.

Remarks

This command is the same as the **query termserver** command.

additional references

[query termserver Command-Line Syntax Key Remote Desktop Services \(Terminal Services\) Command Reference](#)

qprocess

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Displays information about processes that are running on a Remote Desktop Session Host (RD Session Host) server.

NOTE

In Windows Server 2008 R2, Terminal Services was renamed Remote Desktop Services. To find out what's new in the latest version, see [What's New in Remote Desktop Services in Windows Server 2012](#) in the Windows Server TechNet Library.

Remarks

This command is the same as the **query process** command.

additional references

[query process Command-Line Syntax Key Remote Desktop Services \(Terminal Services\) Command Reference](#)

query

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Displays information about processes, sessions, and Remote Desktop Session Host (RD Session Host) servers.

NOTE

In Windows Server 2008 R2, Terminal Services was renamed Remote Desktop Services. To find out what's new in the latest version, see [What's New in Remote Desktop Services in Windows Server 2012](#) in the Windows Server TechNet Library.

Syntax

```
query process
query session
query termserver
query user
```

Parameters

| PARAMETER | DESCRIPTION |
|----------------------------------|---|
| query process | Displays information about processes that are running on an rd Session Host server. |
| query session | Displays information about sessions on an rd Session Host server. |
| query termserver | Displays a list of all rd Session Host servers on the network. |
| query user | Displays information about user sessions on an rd Session Host server. |

additional references

[Command-Line Syntax Key Remote Desktop Services \(Terminal Services\) Command Reference](#)

quser

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Displays information about user sessions on a Remote Desktop Session Host (rd Session Host) server.

NOTE

In Windows Server 2008 R2, Terminal Services was renamed Remote Desktop Services. To find out what's new in the latest version, see [What's New in Remote Desktop Services in Windows Server 2012](#) in the Windows Server TechNet Library.

Remarks

This command is the same as the **query user** command.

additional references

[query user](#)

[Command-Line Syntax Key](#)

[Remote Desktop Services \(Terminal Services\) Command Reference](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Displays information about sessions on a Remote Desktop Session Host (RD Session Host) server.

NOTE

In Windows Server 2008 R2, Terminal Services was renamed Remote Desktop Services. To find out what's new in the latest version, see [What's New in Remote Desktop Services in Windows Server 2012](#) in the Windows Server TechNet Library.

Remarks

This command is the same as the **query session** command.

additional references

[query session Command-Line Syntax Key Remote Desktop Services \(Terminal Services\) Command Reference](#)

rcp

4/13/2018 • 2 minutes to read • [Edit Online](#)

Copies files between computers. This command has been deprecated. You can install the Subsystem for UNIX-based Applications using the Add Features Wizard. For more information, see [Windows Server 2008 UNIX Interoperability Components](#) at the Microsoft Web site. After installation, you can then open a C Shell or Korn Shell command window and run **rcp**. For more information, type **man rcp** at the C Shell or Korn Shell prompt.

rd

4/13/2018 • 2 minutes to read • [Edit Online](#)

Deletes a directory. This command is the same as the **rmdir** command.

For examples of how to use this command, see [Examples](#).

Syntax

```
rd [<Drive>:]<Path> [/s [/q]]
rmdir [<Drive>:]<Path> [/s [/q]]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|--|
| [<Drive>:] | Specifies the location and the name of the directory that you want to delete. <i>Path</i> is required. |
| /s | Deletes a directory tree (the specified directory and all its subdirectories, including all files). |
| /q | Specifies quiet mode. Does not prompt for confirmation when deleting a directory tree. (Note that /q works only if /s is specified.) |
| /? | Displays help at the command prompt. |

Remarks

- You cannot delete a directory that contains files, including hidden or system files. If you attempt to do so, the following message appears:

```
The directory is not empty
```

Use the **dir /a** command to list all files (including hidden and system files). Then use the **attrib** command with **-h** to remove hidden file attributes, **-s** to remove system file attributes, or **-h -s** to remove both hidden and system file attributes. After the hidden and file attributes have been removed, you can delete the files.

- If you insert a backslash () at the beginning of *Path*, *Path* will start at the root directory (regardless of the current directory).
- You cannot use **rd** to delete the current directory. If you attempt to delete the current directory, the following error message appears:

```
The process cannot access the file because it is being used by another process.
```

If you receive this error message, you must change to a different directory (not a subdirectory of the current directory), and then use **rd** (specify *Path* if necessary).

- The **rd** command, with different parameters, is available from the Recovery Console.

Examples

You cannot delete the directory that you are currently working in. You must change to a directory that is not within the current directory. For example, to change to the parent directory, type:

```
cd ..
```

You can now safely remove the desired directory.

Use the **/s** option to remove a directory tree. For example, to remove a directory named Test (and all its subdirectories and files) from the current directory, type:

```
rd /s test
```

To run the previous example in quiet mode, type:

```
rd /s /q test
```

Caution

When you run **rd /s** in quiet mode, the entire directory tree is deleted without confirmation. Ensure that important files are moved or backed up before using the **/q** command-line option.

Additional references

[Command-Line Syntax Key](#)

rdpsign

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Enables you to digitally sign a Remote Desktop Protocol (.rdp) file. for examples of how to use this command, see [Examples](#).

NOTE

In Windows Server 2008 R2, Terminal Services was renamed Remote Desktop Services. To find out what's new in the latest version, see [What's New in Remote Desktop Services in Windows Server 2012](#) in the Windows Server TechNet Library.

Syntax

```
rdpsign /sha1 <hash> [/q | /v |] [/l] <file_name.rdp>
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------|--|
| /sha1 <hash> | Specifies the thumbprint, which is the Secure Hash Algorithm 1 (SHA1) hash of the signing certificate that is included in the certificate store. |
| /q | Quiet mode. No output when the command succeeds and minimal output if the command fails. |
| /v | verbose mode. Displays all warnings, messages, and status. |
| /l | Tests the signing and output results without actually replacing any of the input files. |
| /? | Displays help at the command prompt. |

Remarks

- The SHA1 certificate thumbprint should represent a trusted .rdp file publisher. To obtain the certificate thumbprint, open the Certificates snap-in, double-click the certificate that you want to use (either in the local computer's certificates store or in your personal certificates store), click the **details** tab, and then in the **Field** list, click **Thumbprint**.

NOTE

When you copy the thumbprint for use with the rdpsign.exe tool, you must remove any spaces.

- You must specify the .rdp file (or files) to sign by using the full file name. Wildcard characters are not accepted.
- The signed output files will overwrite the input files.
- If any of the .rdp files cannot be read or written to, the tool will continue to the next file if multiple files are specified.

Examples

- To sign an .rdp file that is named File1.rdp, navigate to the folder where you saved the .rdp file, and then type the following: `rdpsign /sha1 hash file1.rdp` > [!NOTE] > The *hash* value represents the SHA1 certificate thumbprint, without any spaces.
- To test whether digital signing will succeed for an .rdp file without actually signing the file, type the following:
`rdpsign /sha1 hash /l file1.rdp`
- To sign multiple .rdp files, separate the file names by using spaces. For example, to sign multiple .rdp files that are named File1.rdp, File2.rdp, and File3.rdp, type the following:
`rdpsign /sha1 hash file1.rdp file2.rdp file3.rdp` ## See Also [Command-Line Syntax Key Remote Desktop Services \(Terminal Services\) Command Reference](#)

recover

4/13/2018 • 2 minutes to read • [Edit Online](#)

Recovers readable information from a bad or defective disk.

For examples of how to use this command, see [Examples](#).

Syntax

```
recover [<Drive>:][<Path>]<FileName>
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------------|--|
| [<Drive>:][<Path>] | Specifies the location and name of the file that you want to recover. <i>FileName</i> is required. |
| /? | Displays help at the command prompt. |

Remarks

- The **recover** command reads a file, sector-by-sector, and recovers data from the good sectors. Data in bad sectors is lost.
- Bad sectors reported by **chkdsk** were marked as "bad" when your disk was prepared for operation. They pose no danger, and **recover** does not affect them.
- Because all data in bad sectors is lost when you recover a file, you should recover only one file at a time.
- You cannot use wildcard characters (* and ?) with the **recover** command. You must specify a file (and the location of the file if it is not in the current directory).

Examples

To recover the file Story.txt in the \Fiction directory on drive D, type:

```
recover d:\fiction\story.txt
```

Additional references

[Command-Line Syntax Key](#)

reg

4/13/2018 • 2 minutes to read • [Edit Online](#)

Performs operations on registry subkey information and values in registry entries. The **reg** commands include:

[Reg add](#)

[Reg compare](#)

[Reg copy](#)

[Reg delete](#)

[Reg export](#)

[Reg import](#)

[Reg load](#)

[Reg query](#)

[Reg restore](#)

[Reg save](#)

[Reg unload](#)

Some operations enable you to view or configure registry entries on local or remote computers, while others allow you to configure only local computers. Using **reg** to configure the registry of remote computers limits the parameters that you can use in some operations. Check the syntax and parameters for each operation to verify that they can be used on remote computers

reg add

4/13/2018 • 2 minutes to read • [Edit Online](#)

Adds a new subkey or entry to the registry.

Syntax

```
reg add <KeyName> [{/v ValueName | /ve}] [/t DataType] [/s Separator] [/d Data] [/f]
```

For examples of how to use this command, see [Examples](#).

Parameters

| PARAMETER | DESCRIPTION |
|----------------|---|
| <KeyName> | Specifies the full path of the subkey or entry to be added. To specify a remote computer, include the computer name (in the format \\<ComputerName>) as part of the <i>KeyName</i> . Omitting \\ComputerName\ causes the operation to default to the local computer. The <i>KeyName</i> must include a valid root key. Valid root keys for the local computer are: HKLM, HKCU, HKCR, HKU, and HKCC. If a remote computer is specified, valid root keys are: HKLM and HKU. |
| /v <ValueName> | Specifies the name of the registry entry to be added under the specified subkey. |
| /ve | Specifies that the registry entry that is added to the registry has a null value. |
| /t <Type> | Specifies the type for the registry entry. <i>Type</i> must be one of the following: REG_SZ REG_MULTI_SZ REG_DWORD_BIG_ENDIAN REG_DWORD REG_BINARY REG_DWORD_LITTLE_ENDIAN REG_LINK REG_FULL_RESOURCE_DESCRIPTOR REG_EXPAND_SZ |
| /s <Separator> | Specifies the character to be used to separate multiple instances of data when the REG_MULTI_SZ data type is specified and more than one entry needs to be listed. If not specified, the default separator is \0. |
| /d <Data> | Specifies the data for the new registry entry. |
| /f | Adds the registry entry without prompting for confirmation. |
| /? | Displays help for reg add at the command prompt. |

Remarks

- Subtrees cannot be added with this operation. This version of **reg** does not ask for confirmation when adding a subkey.
- The following table lists the return values for the **reg add** operation.

| VALUE | DESCRIPTION |
|-------|-------------|
| 0 | Success |
| 1 | Failure |

- For the REG_EXPAND_SZ key type, use the caret symbol (^) with %" inside the /d parameter

Examples

To add the key HKLM\Software\MyCo on remote computer ABC, type:

```
REG ADD \\ABC\HKLM\Software\MyCo
```

To add a registry entry to HKLM\Software\MyCo with a value named **Data** of type REG_BINARY and data of **fe340ead**, type:

```
REG ADD HKLM\Software\MyCo /v Data /t REG_BINARY /d fe340ead
```

To add a multivalued registry entry to HKLM\Software\MyCo with a value name of **MRU** of type REG_MULTI_SZ and data of **fax\0mail\0\0**, type:

```
REG ADD HKLM\Software\MyCo /v MRU /t REG_MULTI_SZ /d fax\0mail\0\0
```

To add an expanded registry entry to HKLM\Software\MyCo with a value name of **Path** of type REG_EXPAND_SZ and data of **%systemroot%**, type:

```
REG ADD HKLM\Software\MyCo /v Path /t REG_EXPAND_SZ /d ^%systemroot^%
```

Additional references

[Command-Line Syntax Key](#)

reg compare

4/13/2018 • 2 minutes to read • [Edit Online](#)

Compares specified registry subkeys or entries.

For examples of how to use this command, see [Examples](#).

Syntax

```
reg compare <KeyName1> <KeyName2> [{/v ValueName | /ve}] [{/oa | /od | /os | on}] [/s]
```

Parameters

| PARAMETER | DESCRIPTION |
|----------------|---|
| <KeyName1> | Specifies the full path of the first subkey to be compared. To specify a remote computer, include the computer name (in the format \\ComputerName) as part of the <i>KeyName</i> . Omitting \\ComputerName\ causes the operation to default to the local computer. The <i>KeyName</i> must include a valid root key. Valid root keys for the local computer are: HKLM, HKCU, HKCR, HKU, and HKCC. If a remote computer is specified, valid root keys are: HKLM and HKU. |
| <KeyName2> | Specifies the full path of the second subkey to be compared. To specify a remote computer, include the computer name (in the format \\ComputerName) as part of the <i>KeyName</i> . Omitting \\ComputerName\ causes the operation to default to the local computer. Specifying only the computer name in <i>KeyName2</i> causes the operation to use the path to the subkey specified in <i>KeyName1</i> . The <i>KeyName</i> must include a valid root key. Valid root keys for the local computer are: HKLM, HKCU, HKCR, HKU, and HKCC. If a remote computer is specified, valid root keys are: HKLM and HKU. |
| /v <ValueName> | Specifies the value name to compare under the subkey. |
| /ve | Specifies that only entries that have a value name of null should be compared. |
| {/oa | /od |
| /oa | Specifies that all differences and matches are displayed. By default, only the differences are listed. |
| /od | Specifies that only differences are displayed. This is the default behavior. |
| /os | Specifies that only matches are displayed. By default, only the differences are listed. |

| PARAMETER | DESCRIPTION |
|-----------|---|
| /on | Specifies that nothing is displayed. By default, only the differences are listed. |
| /s | Compares all subkeys and entries recursively. |
| /? | Displays help for reg compare at the command prompt. |

Remarks

The following table lists the return values for **reg compare**.

| VALUE | DESCRIPTION |
|-------|---|
| 0 | The comparison is successful and the result is identical. |
| 1 | The comparison failed. |
| 2 | The comparison was successful and differences were found. |

The following table lists the symbols displayed in the results.

| SYMBOL | DESCRIPTION |
|--------|--|
| = | <i>KeyName1</i> data is equal to <i>KeyName2</i> data. |
| < | <i>KeyName1</i> data is less than <i>KeyName2</i> data. |
| > | <i>KeyName1</i> data is greater than <i>KeyName2</i> data. |

Examples

To compare all values under the key **MyApp** with all values under the key **SaveMyApp**, type:

```
REG COMPARE HKLM\Software\MyCo\MyApp HKLM\Software\MyCo\SaveMyApp
```

To compare the value for the Version under the key **MyCo** and the value for the Version under the key **MyCo1**, type:

```
REG COMPARE HKLM\Software\MyCo HKLM\Software\MyCo1 /v Version
```

To compare all subkeys and values under HKLM\Software\MyCo on the computer named ZODIAC with all subkeys and values under HKLM\Software\MyCo on the local computer, type:

```
REG COMPARE \\ZODIAC\HKLM\Software\MyCo \\. /s
```

Additional references

[Command-Line Syntax Key](#)

reg copy

4/13/2018 • 2 minutes to read • [Edit Online](#)

Copies a registry entry to a specified location on the local or remote computer.

For examples of how to use this command, see [Examples](#).

Syntax

```
reg copy <KeyName1> <KeyName2> [/s] [/f]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|--|
| <KeyName1> | Specifies the full path of the subkey to copy. To specify a remote computer, include the computer name (in the format \\ComputerName) as part of the <i>KeyName</i> . Omitting \\ComputerName\ causes the operation to default to the local computer. The <i>KeyName</i> must include a valid root key. Valid root keys for the local computer are: HKLM, HKCU, HKCR, HKU, and HKCC. If a remote computer is specified, valid root keys are: HKLM and HKU. |
| <KeyName2> | Specifies the full path of the subkey destination. To specify a remote computer, include the computer name (in the format \\ComputerName) as part of the <i>KeyName</i> . Omitting \\ComputerName\ causes the operation to default to the local computer. The <i>KeyName</i> must include a valid root key. Valid root keys for the local computer are: HKLM, HKCU, HKCR, HKU, and HKCC. If a remote computer is specified, valid root keys are: HKLM and HKU. |
| /s | Copies all subkeys and entries under the specified subkey. |
| /f | Copies the subkey without prompting for confirmation. |
| /? | Displays help for reg copy at the command prompt. |

Remarks

- Reg does not ask for confirmation when copying a subkey.
- The following table lists the return values for the **reg copy** operation.

| VALUE | DESCRIPTION |
|-------|-------------|
| 0 | Success |
| 1 | Failure |

Examples

To copy all subkeys and values under the key MyApp to the key SaveMyApp, type:

```
REG COPY HKLM\Software\MyCo\MyApp HKLM\Software\MyCo\SaveMyApp /s
```

To copy all values under the key MyCo on the computer named ZODIAC to the key MyCo1 on the current computer, type:

```
REG COPY \\ZODIAC\HKLM\Software\MyCo HKLM\Software\MyCo1
```

Additional references

[Command-Line Syntax Key](#)

reg delete

4/13/2018 • 2 minutes to read • [Edit Online](#)

Deletes a subkey or entries from the registry.

For examples of how to use this command, see [Examples](#).

Syntax

```
Reg delete <KeyName> [{/v ValueName | /ve | /va}] [/f]
```

Parameters

| PARAMETER | DESCRIPTION |
|----------------|---|
| <KeyName> | Specifies the full path of the subkey or entry to be deleted. To specify a remote computer, include the computer name (in the format \\ComputerName) as part of the <i>KeyName</i> . Omitting \\ComputerName\ causes the operation to default to the local computer. The <i>KeyName</i> must include a valid root key. Valid root keys for the local computer are: HKLM, HKCU, HKCR, HKU, and HKCC. If a remote computer is specified, valid root keys are: HKLM and HKU. |
| /v <ValueName> | Deletes a specific entry under the subkey. If no entry is specified, then all entries and subkeys under the subkey will be deleted. |
| /ve | Specifies that only entries that have no value will be deleted. |
| /va | Deletes all entries under the specified subkey. Subkeys under the specified subkey are not deleted. |
| /f | Deletes the existing registry subkey or entry without asking for confirmation. |
| /? | Displays help for reg delete at the command prompt. |

Remarks

The following table lists the return values for the **reg delete** operation.

| VALUE | DESCRIPTION |
|-------|-------------|
| 0 | Success |
| 1 | Failure |

Examples

To delete the registry key Timeout and its all subkeys and values, type:

```
REG DELETE HKLM\Software\MyCo\MyApp\Timeout
```

To delete the registry value MTU under HKLM\Software\MyCo on the computer named ZODIAC, type:

```
REG DELETE \\ZODIAC\HKLM\Software\MyCo /v MTU
```

Additional references

[Command-Line Syntax Key](#)

reg export

4/13/2018 • 2 minutes to read • [Edit Online](#)

Copies the specified subkeys, entries, and values of the local computer into a file for transfer to other servers.

For examples of how to use this command, see [Examples](#).

Syntax

```
Reg export KeyName FileName [/y]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|--|
| <KeyName> | Specifies the full path of the subkey. The export operation works only with the local computer. The KeyName must include a valid root key. Valid root keys are: HKLM, HKCU, HKCR, HKU, and HKCC. |
| <FileName> | Specifies the name and path of the file to be created during the operation. The file must have a .reg extension. |
| /y | Overwrites any existing file with the name <i>FileName</i> without prompting for confirmation. |
| /? | Displays help for reg export at the command prompt. |

Remarks

The following table lists the return values for the **reg export** operation.

| VALUE | DESCRIPTION |
|-------|-------------|
| 0 | Success |
| 1 | Failure |

Examples

To export the contents of all subkeys and values of the key MyApp to the file AppBkUp.reg, type:

```
reg export HKLM\Software\MyCo\MyApp AppBkUp.reg
```

Additional references

[Command-Line Syntax Key](#)

reg import

4/13/2018 • 2 minutes to read • [Edit Online](#)

Copies the contents of a file that contains exported registry subkeys, entries, and values into the registry of the local computer.

For examples of how to use this command, see [Examples](#).

Syntax

```
Reg import FileName
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|--|
| <FileName> | Specifies the name and path of the file that has content to be copied into the registry of the local computer. This file must be created in advance by using reg export . |
| /? | Displays help for reg import at the command prompt. |

Remarks

The following table lists the return values for the **reg import** operation.

| VALUE | DESCRIPTION |
|-------|-------------|
| 0 | Success |
| 1 | Failure |

Examples

To import registry entries from the file named AppBkUp.reg, type:

```
reg import AppBkUp.reg
```

Additional references

[Command-Line Syntax Key](#)

reg load

4/13/2018 • 2 minutes to read • [Edit Online](#)

Writes saved subkeys and entries into a different subkey in the registry. Intended for use with temporary files that are used for troubleshooting or editing registry entries.

For examples of how to use this command, see [Examples](#).

Syntax

```
reg load KeyName FileName
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|--|
| <KeyName> | Specifies the full path of the subkey to be loaded. For specifying remote computers, include the computer name (in the format \\ComputerName) as part of the <i>KeyName</i> . Omitting \\ComputerName\ causes the operation to default to the local computer. The <i>KeyName</i> must include a valid root key. Valid root keys for the local computer are: HKLM, HKCU, HKCR, HKU, and HKCC. If a remote computer is specified, valid root keys are: HKLM and HKU. |
| <FileName> | Specifies the name and path of the file to be loaded. This file must be created in advance by using the reg save operation and a .hiv extension. |
| /? | Displays help for reg load at the command prompt. |

Remarks

The following table lists the return values for the **reg load** operation.

| VALUE | DESCRIPTION |
|-------|-------------|
| 0 | Success |
| 1 | Failure |

Examples

To load the file named TempHive.hiv to the key HKLM\TempHive, type:

```
REG LOAD HKLM\TempHive TempHive.hiv
```

Additional references

[Command-Line Syntax Key](#)

reg query

4/13/2018 • 2 minutes to read • [Edit Online](#)

Returns a list of the next tier of subkeys and entries that are located under a specified subkey in the registry.

For examples of how to use this command, see [Examples](#).

Syntax

```
reg query <KeyName> [{/v <ValueName> | /ve}] [/s] [/se <Separator>] [/f <Data>] [{/k | /d}] [/c] [/e] [/t <Type>] [/z]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------|---|
| <KeyName> | Specifies the full path of the subkey. For specifying remote computers, include the computer name (in the format \\ComputerName) as part of the <i>KeyName</i> . Omitting \\ComputerName\ causes the operation to default to the local computer. The <i>KeyName</i> must include a valid root key. Valid root keys for the local computer are: HKLM, HKCU, HKCR, HKU, and HKCC. If a remote computer is specified, valid root keys are: HKLM and HKU. |
| /v <ValueName> | Specifies the registry value name that is to be queried. If omitted, all value names for <i>KeyName</i> are returned. <i>ValueName</i> for this parameter is optional if the /f option is also used. |
| /ve | Runs a query for value names that are empty. |
| /s | Specifies to query all subkeys and value names recursively. |
| /se <Separator> | Specifies the single value separator to search for in the value name type REG_MULTI_SZ. If <i>Separator</i> is not specified, \0 is used. |
| /f <Data> | Specifies the data or pattern to search for. Use double quotes if a string contains spaces. If not specified, a wildcard (*) is used as the search pattern. |
| /k | Specifies to search in key names only. |
| /d | Specifies to search in data only. |
| /c | Specifies that the query is case sensitive. By default, queries are not case sensitive. |
| /e | Specifies to return only exact matches. By default, all the matches are returned. |

| PARAMETER | DESCRIPTION |
|-----------|--|
| /t <Type> | Specifies registry types to search. Valid types are: REG_SZ, REG_MULTI_SZ, REG_EXPAND_SZ, REG_DWORD, REG_BINARY, REG_NONE. If not specified, all types are searched. |
| /z | Specifies to include the numeric equivalent for the registry type in search results. |
| /? | Displays help for reg query at the command prompt. |

Remarks <optional section>

The following table lists the return values for the **reg query** operation.

| VALUE | DESCRIPTION |
|-------|-------------|
| 0 | Success |
| 1 | Failure |

Examples

To display the value of the name value Version in the HKLM\Software\Microsoft\ResKit key, type:

```
REG QUERY HKLM\Software\Microsoft\ResKit /v Version
```

To display all subkeys and values under the key HKLM\Software\Microsoft\ResKit\Nt\Setup on a remote computer named ABC, type:

```
REG QUERY \\ABC\HKLM\Software\Microsoft\ResKit\Nt\Setup /s
```

To display all the subkeys and values of the type REG_MULTI_SZ using # as the separator, type:

```
REG QUERY HKLM\Software\Microsoft\ResKit\Nt\Setup /se #
```

To display the key, value, and data for exact and case sensitive matches of SYSTEM under the HKLM root of data type REG_SZ, type:

```
REG QUERY HKLM /f SYSTEM /t REG_SZ /c /e
```

To display the key, value, and data that match **0F** in the data under the HKCU root key of data type REG_BINARY.

```
REG QUERY HKCU /f 0F /d /t REG_BINARY
```

To display the value and data for value names of null (default) under HKLM\SOFTWARE, type:

```
REG QUERY HKLM\SOFTWARE /ve
```

Additional references

reg restore

4/13/2018 • 2 minutes to read • [Edit Online](#)

Writes saved subkeys and entries back to the registry.

For examples of how to use this command, see [Examples](#).

Syntax

```
Reg restore <KeyName> <FileName>
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|--|
| <KeyName> | Specifies the full path of the subkey to be restored. The restore operation works only with the local computer. The KeyName must include a valid root key. Valid root keys are: HKLM, HKCU, HKCR, HKU, and HKCC. |
| <FileName> | Specifies the name and path of the file with content to be written into the registry. This file must be created in advance with the reg save operation using a .hiv extension. |
| /? | Displays help for reg restore at the command prompt. |

Remarks

- Before editing any registry entries, save the parent subkey with the **reg save** operation. If the edit fails, restore the original subkey with the **reg restore** operation.
- The following table lists the return values for the **reg restore** operation.

| VALUE | DESCRIPTION |
|-------|-------------|
| 0 | Success |
| 1 | Failure |

Examples

To restore the file named NTRKBkUp.hiv into the key HKLM\Software\Microsoft\ResKit, and overwrite the existing contents of the key, type:

```
REG RESTORE HKLM\Software\Microsoft\ResKit NTRKBkUp.hiv
```

Additional references

[Command-Line Syntax Key](#)

reg save

4/13/2018 • 2 minutes to read • [Edit Online](#)

Saves a copy of specified subkeys, entries, and values of the registry in a specified file.

For examples of how to use this command, see [Examples](#).

Syntax

```
reg save <KeyName> <FileName> [/y]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|---|
| <KeyName> | Specifies the full path of the subkey. For specifying remote computers, include the computer name (in the format \\ComputerName) as part of the <i>KeyName</i> . Omitting \\ComputerName\ causes the operation to default to the local computer. The <i>KeyName</i> must include a valid root key. Valid root keys for the local computer are: HKLM, HKCU, HKCR, HKU, and HKCC. If a remote computer is specified, valid root keys are: HKLM and HKU. |
| <FileName> | Specifies the name and path of the file that is created. If no path is specified, the current path is used. |
| /y | Overwrites an existing file with the name <i>FileName</i> without prompting for confirmation. |
| /? | Displays help for reg save at the command prompt. |

Remarks <optional section>

- The following table lists the return values for the **reg save** operation.

| VALUE | DESCRIPTION |
|-------|-------------|
| 0 | Success |
| 1 | Failure |

- Before editing any registry entries, save the parent subkey with the **reg save** operation. If the edit fails, restore the original subkey with the **reg restore** operation.

Examples

To save the hive MyApp into the current folder as a file named AppBkUp.hiv, type:

```
REG SAVE HKLM\Software\MyCo\MyApp AppBkUp.hiv
```

Additional references

[Command-Line Syntax Key](#)

reg unload

4/13/2018 • 2 minutes to read • [Edit Online](#)

Removes a section of the registry that was loaded using the **reg load** operation.

For examples of how to use this command, see [Examples](#).

Syntax

```
reg unload <KeyName>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| <KeyName> | Specifies the full path of the subkey to be unloaded. For specifying remote computers, include the computer name (in the format \\ComputerName) as part of the <i>KeyName</i> . Omitting \\ComputerName\ causes the operation to default to the local computer. The <i>KeyName</i> must include a valid root key. Valid root keys for the local computer are HKLM, HKCU, HKCR, HKU, and HKCC. If a remote computer is specified, valid root keys are HKLM and HKU. |
| /? | Displays help for reg unload at the command prompt. |

Remarks

The following table lists the return values for the **reg unload** option.

| VALUE | DESCRIPTION |
|-------|-------------|
| 0 | Success |
| 1 | Failure |

Examples

To unload the hive TempHive in the file HKLM, type:

```
REG UNLOAD HKLM\TempHive
```

Caution

Do not edit the registry directly unless you have no alternative. The registry editor bypasses standard safeguards, allowing settings that can degrade performance, damage your system, or even require you to reinstall Windows. You can safely alter most registry settings by using the programs in Control Panel or Microsoft Management Console (MMC). If you must edit the registry directly, back it up first.

Additional references

regini

8/21/2018 • 2 minutes to read • [Edit Online](#)

Modifies the registry from the command line or a script, and applies changes that were preset in one or more text files. You can create, modify, or delete registry keys, in addition to modifying the permissions on the registry keys.

For details on the format and content of the text script file that Regini.exe uses to make changes to the registry, see [How to change registry values or permissions from a command line or a script](#).

Syntax

```
regini [-m \\machinename | -h hivefile hiveroot][-i n] [-o outputWidth][-b] textFiles...
```

Parameters

|Parameter|Description|

| | |
|------------------------|--|
| -M <\\COMPUTERNAME> | SPECIFIES THE REMOTE COMPUTER NAME WITH A REGISTRY THAT IS TO BE MODIFIED. USE THE FORMAT \\COMPUTERNAME. |
| -h <hivefile hiveroot> | Specifies the local registry hive to modify. You must specify the name of the hive file and the root of the hive in the format hivefile hiveroot . |
| -i <n> | Specifies the level of indentation to use to indicate the tree structure of registry keys in the command output. The Regdmp.exe tool (which gets a registry key's current permissions in binary format) uses indentation in multiples of four, so the default value is 4 . |
| -o <outputwidth> | Specifies the width of the command output, in characters. If the output will appear in the command window, the default value is the width of the window. If the output is directed to a file, the default value is 240 characters. |
| -b | Specifies that Regini.exe output is backward compatible with previous versions of Regini.exe . See the Remarks section for details. |
| textfiles | Specifies the name of one or more text files that contain registry data. Any number of ANSI or Unicode text files can be listed. |

Remarks

The following guidelines apply primarily to the content of the text files that contain registry data that you apply by using **Regini.exe**.

- Use the semicolon as an end-of-line comment character. It must be the first non-blank character in a line.
- Use the backslash to indicate continuation of a line. The command will ignore all characters from the backslash up to (but not including) the first non-blank character of the next line. If you include more than one space before the backslash, it is replaced by a single space.

- Use hard-tab characters to control indentation. This indentation indicates the tree structure of the registry keys; however, these characters are converted to a single space regardless of their position.

Additional references

- [Command-Line Syntax Key](#)

regsvr32

4/13/2018 • 2 minutes to read • [Edit Online](#)

Registers .dll files as command components in the registry.

For examples of how to use this command, see [Examples](#).

Syntax

```
regsvr32 [/u] [/s] [/n] [/i[:cmdline]] <DllName>
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------|--|
| /u | Unregisters server. |
| /s | Runs Regsvr32 without displaying messages. |
| /n | Runs Regsvr32 without calling DllRegisterServer . (Requires the /i parameter.) |
| /i:<cmdline> | Passes an optional command-line string (<i>cmdline</i>) to DllInstall . If you use this parameter in conjunction with the /u parameter, it calls DllUninstall . |
| <DllName> | The name of the .dll file that will be registered. |
| /? | Displays help at the command prompt. |

Examples

To register the .dll for the Active Directory Schema, type:

```
regsvr32 schmmgmt.dll
```

Additional references

[Command-Line Syntax Key](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Extracts performance counters from performance counter logs into other formats, such as text-TSV (for tab-delimited text), text-CSV (for comma-delimited text), binary-BIN, or SQL.

Syntax

```
relog [<FileName> [<FileName> ...]] [/a] [/c <path> [<path> ...]] [/cf <FileName>] [/f {bin|csv|tsv|SQL}] [/t <Value>] [/o {OutputFile|DSN!CounterLog}] [/b <M/D/YYYY> [[<HH>:] <MM>:] <SS>] [/e <M/D/YYYY> [[<HH>:] <MM>:] <SS>] [/config {<FileName>|i}] [/q]
```

Parameters

| PARAMETER | DESCRIPTION |
|--|---|
| <i>FileName</i> [<i>FileName</i> ...] | Specifies the pathname of an existing performance counter log. You can specify multiple input files. |
| -a | Appends output file instead of overwriting. This option does not apply to SQL format where the default is always to append. |
| -c <i>path</i> [<i>path</i> ...] | Specifies the performance counter path to log. To specify multiple counter paths, separate them with a space and enclose the counter paths in quotation marks (for example, " <i>Counterpath1 Counterpath2</i> ") |
| -cf <i>FileName</i> | Specifies the pathname of the text file that lists the performance counters to be included in a relog file. Use this option to list counter paths in an input file, one per line. Default setting is all counters in the original log file are relogged. |
| -f {bin csv tsv SQL} | Specifies the pathname of the output file format. The default format is bin . For a SQL database, the output file specifies the <i>DSN!CounterLog</i> . You can specify the database location by using the ODBC manager to configure the DSN (Database System Name). |
| -t <i>Value</i> | Specifies sample intervals in "N" records. Includes every nth data point in the relog file. Default is every data point. |
| -o { <i>OutputFile</i> "SQL:DSN!Counter_Log"} where DSN is a ODMC DSN defined on the system. | Specifies the pathname of the output file or SQL database where the counters will be written. Note: For the 64-bit and 32-bit versions of Relog.exe, you need to define a DSN in the ODBC Data Source (64-bit and 32-bit respectively) |

| PARAMETER | DESCRIPTION |
|----------------------------|---|
| -b <M/D/YYYY> [[HH:]MM:]SS | Specifies begin time for copying first record from the input file. date and time must be in this exact format <i>M/D/YYYYHH:MM:SS</i> . |
| -e <M/D/YYYY> [[HH:]MM:]SS | Specifies end time for copying last record from the input file. date and time must be in this exact format <i>M/D/YYYYHH:MM:SS</i> . |
| -config {FileName i} | Specifies the pathname of the settings file that contains command-line parameters. Use -i in the configuration file as a placeholder for a list of input files that can be placed on the command line. On the command line, however, you do not need to use i. You can also use wildcards such as *.blg to specify many input file names. |
| -q | Displays the performance counters and time ranges of log files specified in the input file. |
| -y | Bypasses prompting by answering "yes" to all questions. |
| /? | Displays help at the command prompt. |

Remarks

Counter path format:

- The general format for counter paths is as follows: [`<computer>`] \<Object>[<Parent>\<Instance#Index>] \<Counter>] where the parent, instance, index, and counter components of the format may contain either a valid name or a wildcard character. The computer, parent, instance, and index components are not necessary for all counters.
- You determine the counter paths to use based on the counter itself. For example, the LogicalDisk object has an instance , so you must provide the <#index> or a wildcard. Therefore, you could use the following format:
\LogicalDisk(*/*#*)*
- In comparison, the Process object does not require an instance <Index>. Therefore, you could use the following format: **\Process(*)\ID Process**
- If a wildcard character is specified in the parent name, all instances of the specified object that match the specified instance and counter fields will be returned.
- If a wildcard character is specified in the instance name, all instances of the specified object and parent object will be returned if all instance names corresponding to the specified index match the wildcard character.
- If a wildcard character is specified in the counter name, all counters of the specified object are returned.
- Partial counter path string matches (for example, pro*) are not supported.

Counter files:

- Counter files are text files that list one or more of the performance counters in the existing log. Copy the full counter name from the log or the **/q** output in <computer>\<Object>\<Instance>\<Counter> format. list one counter path on each line.

Copying counters:

- When executed, **relog** copies specified counters from every record in the input file, converting the format if necessary. Wildcard paths are allowed in the counter file.

Saving input file subsets:

- Use the **/t** parameter to specify that input files are inserted into output files at intervals of every *th* record. By default, data is relogged from every record.
Using **/b** and **/e** parameters with log files
- You can specify that your output logs include records from before begin-time (that is, **/b**) to provide data for counters that require computation values of the formatted value. The output file will have the last records from input files with timestamps less than the **/e** (that is, end time) parameter.
Using the **/config** option:
- The contents of the setting file used with the **/config** option should have the following format:
 - <CommandOption>\<Value>, where <CommandOption> is a command line option and <Value> specifies its value.

For more information about incorporating **relog** into your Windows Management Instrumentation (WMI) scripts, see "Scripting WMI" at the [Microsoft Windows Resource Kits Web site](#).

Examples

To resample existing trace logs at fixed intervals of 30, list counter paths, output files and formats:

```
relog c:\perflogs\daily_trace_log.blg /cf counter_file.txt /o c:\perflogs\reduced_log.csv /t 30 /f csv
```

To resample existing trace logs at fixed intervals of 30, list counter paths and output file:

```
relog c:\perflogs\daily_trace_log.blg /cf counter_file.txt /o c:\perflogs\reduced_log.blg /t 30
```

To resample existing trace logs into a database use:

```
relog "c:\perflogs\daily_trace_log.blg" -f sql -o "SQL:sql2016x64odbc!counter_log"
```

Additional references

- [Command-Line Syntax Key](#)

rem

4/13/2018 • 2 minutes to read • [Edit Online](#)

Records comments (remarks) in a batch file or CONFIG.SYS. If no comment is specified, **rem** adds vertical spacing.

For examples of how to use this command, see [Examples](#).

Syntax

```
rem [<Comment>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| <Comment> | Specifies a string of characters to include as a comment. |
| /? | Displays help at the command prompt. |

Remarks

- The **rem** command does not display comments on the screen. You must use the **echo on** command in your batch or CONFIG.SYS file to display comments on the screen.
- You cannot use a redirection character (< or >) or pipe (|) in a batch file comment.
- Although you can use **rem** without a comment to add vertical spacing to a batch file, you can also use blank lines. Blank lines are ignored when a batch program is processed.

Examples

The following example shows a batch file that uses remarks for comments and for vertical spacing:

```
@echo off
rem This batch program formats and checks new disks.
rem It is named Checknew.bat.
rem
rem echo Insert new disk in Drive B.
pause
format b: /v chkdisk b:
```

To include an explanatory comment before the **prompt** command in your CONFIG.SYS file, add the following lines to CONFIG.SYS:

```
rem Set prompt to indicate current directory
prompt $p$g
```

Additional references

[Command-Line Syntax Key](#)

ren

7/11/2018 • 2 minutes to read • [Edit Online](#)

Renames files or directories. This command is the same as the **rename** command.

For examples of how to use this command, see [Examples](#).

Syntax

```
ren [<Drive>:][<Path>]<FileName1> <FileName2>
rename [<Drive>:][<Path>]<FileName1> <FileName2>
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------------------------|---|
| [<Drive>:][<Path>]<FileName1> | Specifies the location and name of the file or set of files you want to rename. <i>FileName1</i> can include wildcard characters (* and ?). |
| <FileName2> | Specifies the new name for the file. You can use wildcard characters to specify new names for multiple files. |
| /? | Displays help at the command prompt. |

Remarks

- You cannot specify a new drive or path when renaming files.
- You cannot use the **ren** command to rename files across drives or to move files to a different directory.
- You can use wildcard characters (* and ?) in either *FileName* parameter. Characters that are represented by wildcard characters in *FileName2* will be identical to the corresponding characters in *FileName1*.
- FileName2* must be a unique file name. If *FileName2* matches an existing file name, **ren** displays the following message:

```
Duplicate file name or file not found
```

Examples

To change all the .txt file name extensions in the current directory to .doc extensions, type:

```
ren *.txt *.doc
```

To change the name of a directory from Chap10 to Part10, type:

```
ren chap10 part10
```

Additional references

[Command-Line Syntax Key](#)

rename

4/13/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

This is the same as the **ren** command. See [ren](#) for syntax and parameters.

additional references

- [Command-Line Syntax Key](#)

repair-bde

4/13/2018 • 3 minutes to read • [Edit Online](#)

Accesses encrypted data on a severely damaged hard disk if the drive was encrypted by using BitLocker. Repair-bde can reconstruct critical parts of the drive and salvage recoverable data as long as a valid recovery password or recovery key is used to decrypt the data. If the BitLocker metadata data on the drive has become corrupt, you must be able to supply a backup key package in addition to the recovery password or recovery key. This key package is backed up in Active Directory Domain Services (AD DS) if you used the default setting for AD DS backup. With this key package and either the recovery password or recovery key, you can decrypt portions of a BitLocker-protected drive if the disk is corrupted. Each key package will work only for a drive that has the corresponding drive identifier. You can use the [BitLocker Recovery Password Viewer for Active Directory](#) to obtain this key package from AD DS.

NOTE

The BitLocker Recovery Password Viewer is included as one of the optional management features installable using Server Manage on Windows Server 2012.

The following limitations exist for the Repair-bde command-line tool:

- Repair-bde cannot repair a drive that failed during the encryption or decryption process.
- Repair-bde assumes that if the drive has any encryption, then the drive has been fully encrypted.

For examples of how this command can be used, see [Examples](#).

Syntax

```
repair-bde <InputVolume> <OutputVolumeorImage> [-rk] [-rp] [-pw] [-kp] [-lf] [-f] [{-?|/?}]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------------|---|
| <InputVolume> | Identifies the drive letter of the BitLocker-encrypted drive that you want to repair. The drive letter must include a colon; for example: C: . |
| <OutputVolumeorImage> | Identifies the drive on which to store the content of the repaired drive. All information on the output drive will be overwritten. |
| -rk | Identifies the location of the recovery key that should be used to unlock the volume. This command may also be specified as -recoverykey . |
| -rp | Identifies the numerical recovery password that should be used to unlock the volume. This command may also be specified as -recoverypassword . |

| PARAMETER | DESCRIPTION |
|-----------|---|
| -pw | Identifies the password that should be used to unlock the volume. This command may also be specified as -password |
| -kp | Identifies the recovery key package that can be used to unlock the volume. This command may also be specified as -keypackage . |
| -lf | Specifies the path to the file that will store Repair-bde error, warning, and information messages. This command may also be specified as -logfile . |
| -f | Forces a volume to be dismounted even if it cannot be locked. This command may also be specified as -force . |
| -? or /? | Displays Help at the command prompt. |

Remarks

If the path to a key package is not specified, **repair-bde** will search the drive for a key package. However, if the hard drive has been damaged, **repair-bde** may not be able to find the package and will prompt you to provide the path.

Examples

The following example attempts to repair drive C and write the content from drive C to drive D by using the recovery key file (RecoveryKey.bek) stored on drive F and writes the results of this attempt to the log file (log.txt) on drive Z.

```
repair-bde C: D: -rk F:\RecoveryKey.bek -lf Z:\log.txt
```

The following example attempts to repair drive C and write the content on drive C to drive D by using the 48-digit recovery password specified. The recovery password should be typed in eight blocks of six digits with a hyphen separating each block.

```
repair-bde C: D: -rp 111111-222222-333333-444444-555555-666666-777777-888888
```

The following example forces drive C to be dismounted and then attempts to repair drive C and write the content on drive C to drive D by using the recovery key package and recovery key file (RecoveryKey.bek) stored on drive F.

```
repair-bde C: D: -kp F:\RecoveryKeyPackage -rk F:\RecoveryKey.bek -f
```

The following example attempts to repair drive C and write the content from drive C to drive D and you must type a password to unlock drive C: when prompted:

```
repair-bde C: D: -pw
```

Additional references

- [Command-Line Syntax Key](#)

replace

7/11/2018 • 2 minutes to read • [Edit Online](#)

Replaces files. If used with the **/a** option, **replace** adds new files to a directory instead of replacing existing files.

For examples of how to use this command, see [Examples](#).

Syntax

```
replace [<Drive1>:][<Path1>]<FileName> [<Drive2>:][<Path2>] [/a] [/p] [/r] [/w]
replace [<Drive1>:][<Path1>]<FileName> [<Drive2>:][<Path2>] [/p] [/r] [/s] [/w] [/u]
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------------------------|---|
| [<Drive1>:][<Path1>]<FileName> | Specifies the location and name of the source file or set of files. <i>FileName</i> is required, and can include wildcard characters (* and ?). |
| [<Drive2>:][<Path2>] | Specifies the location of the destination file. You cannot specify a file name for files you replace. If you do not specify a drive or path, replace uses the current drive and directory as the destination. |
| /a | Adds new files to the destination directory instead of replacing existing files. You cannot use this command-line option with the /s or /u command-line option. |
| /p | Prompts you for confirmation before replacing a destination file or adding a source file. |
| /r | Replaces Read-only and unprotected files. If you attempt to replace a Read-only file, but you do not specify /r , an error results and stops the replacement operation. |
| /w | Waits for you to insert a disk before the search for source files begins. If you do not specify /w , replace begins replacing or adding files immediately after you press ENTER. |
| /s | Searches all subdirectories in the destination directory and replaces matching files. You cannot use /s with the /a command-line option. The replace command does not search subdirectories that are specified in <i>Path1</i> . |
| /u | Replaces only those files on the destination directory that are older than those in the source directory. You cannot use /u with the /a command-line option. |
| /? | Displays help at the command prompt. |

Remarks

- As **replace** adds or replaces files, the file names are displayed on the screen. After **replace** is finished, a summary line is displayed in one of the following formats:

```
nnn files added nnn files replaced no file added no file replaced
```

- If you are using floppy disks and you need to switch disks during the **replace** operation, you can specify the **/w** command-line option so that **replace** will wait for you to switch the disks.
- You cannot use **replace** to update hidden files or system files.
- The following table shows each exit code and a brief description of its meaning:

| Exit code | Description |
|-----------|---|
| 0 | The replace command successfully replaced or added the files. |
| 1 | The replace command encountered an incorrect version of MS-DOS. |
| 2 | The replace command could not find the source files. |
| 3 | The replace command could not find the source or destination path. |
| 5 | The user does not have access to the files that you want to replace. |
| 8 | There is insufficient system memory to carry out the command. |
| 11 | The user used the wrong syntax on the command line. |

NOTE

You can use the ERRORLEVEL parameter on the **if** command line in a batch program to process exit codes that are returned by **replace**.

Examples

To update all the versions of a file named Phones.cli (which appear in multiple directories on drive C), with the latest version of the Phones.cli file from a floppy disk in drive A, type:

```
replace a:\phones.cli c:\ /s
```

Additional references

[Command-Line Syntax Key](#)

reset session

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Enables you to reset (delete) a session on a Remote Desktop Session Host (rd Session Host) server. for examples of how to use this command, see [Examples](#).

NOTE

In Windows Server 2008 R2, Terminal Services was renamed Remote Desktop Services. To find out what's new in the latest version, see [What's New in Remote Desktop Services in Windows Server 2012](#) in the Windows Server TechNet Library.

Syntax

```
reset session {<SessionName> | <SessionID>} [/server:<ServerName>] [/v]
```

Parameters

| PARAMETER | DESCRIPTION |
|----------------------|---|
| <SessionName> | Specifies the name of the session that you want to reset. To determine the name of the session, use the query session command. |
| <SessionID> | Specifies the ID of the session to reset. |
| /server:<ServerName> | Specifies the terminal server containing the session that you want to reset. Otherwise, the current rd Session Host server is used. |
| /v | Displays information about the actions being performed. |
| /? | Displays help at the command prompt. |

Remarks

- You can always reset your own sessions, but you must have Full Control access permission to reset another user's session.
- Be aware that resetting a user's session without warning the user can result in the loss of data at the session.
- You should reset a session only when it malfunctions or appears to have stopped responding.
- The **/server** parameter is required only if you use **reset session** from a remote server.

Examples

- To reset the session designated rdp-tcp#6, type:

```
reset session rdp-tcp#6
```

- To reset the session that uses session ID 3, type:

```
reset session 3
```

Additional references

[Command-Line Syntax Key](#)

[Remote Desktop Services \(Terminal Services\) Command Reference](#)

rexec

4/13/2018 • 2 minutes to read • [Edit Online](#)

Rexec is deprecated, and is not guaranteed to be supported in future releases of Windows.

This tool is included in Windows Server 2003. For more information see [Rexec](#).

risetup

4/13/2018 • 2 minutes to read • [Edit Online](#)

The **risetup** command is deprecated in Windows Server® 2008 and Windows Server 2008 R2.

rmdir

4/13/2018 • 2 minutes to read • [Edit Online](#)

This command is the same as the **rd** command. See [Rd](#) for syntax and parameters.

robocopy

7/26/2018 • 6 minutes to read • [Edit Online](#)

Copies file data.

Syntax

```
robocopy <Source> <Destination> [<File>[ ...]] [<Options>]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------|---|
| <Source> | Specifies the path to the source directory. |
| <Destination> | Specifies the path to the destination directory. |
| <File> | Specifies the file or files to be copied. You can use wildcard characters (* or ?), if you want. If the File parameter is not specified, *.* is used as the default value. |
| <Options> | Specifies options to be used with the robocopy command. |

Copy options

| OPTION | DESCRIPTION |
|----------|--|
| /s | Copies subdirectories. Note that this option excludes empty directories. |
| /e | Copies subdirectories. Note that this option includes empty directories. For additional information, see Remarks . |
| /lev:<N> | Copies only the top <i>N</i> levels of the source directory tree. |
| /z | Copies files in restartable mode. |
| /b | Copies files in Backup mode. |
| /zb | Uses restartable mode. If access is denied, this option uses Backup mode. |
| /efsraw | Copies all encrypted files in EFS RAW mode. |

| OPTION | DESCRIPTION |
|--------------------|--|
| /copy:<CopyFlags> | Specifies the file properties to be copied. The following are the valid values for this option: D Data A Attributes T Time stamps S NTFS access control list (ACL) O Owner information U Auditing information The default value for CopyFlags is DAT (data, attributes, and time stamps). |
| /dcopy:<copyflags> | Defines what to copy for directories. Default is DA. Options are D = data, A = attributes, and T = timestamps. |
| /sec | Copies files with security (equivalent to /copy:DATS). |
| /copyall | Copies all file information (equivalent to /copy:DATSOU). |
| /nocopy | Copies no file information (useful with /purge). |
| /secfix | Fixes file security on all files, even skipped ones. |
| /timfix | Fixes file times on all files, even skipped ones. |
| /purge | Deletes destination files and directories that no longer exist in the source. For additional information, see Remarks . |
| /mir | Mirrors a directory tree (equivalent to /e plus /purge). For additional information, see Remarks . |
| /mov | Moves files, and deletes them from the source after they are copied. |
| /move | Moves files and directories, and deletes them from the source after they are copied. |
| /a+: [RASHCNET] | Adds the specified attributes to copied files. |
| /a-: [RASHCNET] | Removes the specified attributes from copied files. |
| /create | Creates a directory tree and zero-length files only. |
| /fat | Creates destination files by using 8.3 character-length FAT file names only. |
| /256 | Turns off support for very long paths (longer than 256 characters). |
| /mon:<N> | Monitors the source, and runs again when more than <i>N</i> changes are detected. |
| /mot:<M> | Monitors source, and runs again in <i>M</i> minutes if changes are detected. |

| OPTION | DESCRIPTION |
|---------------|--|
| /MT[:N] | Creates multi-threaded copies with <i>N</i> threads. <i>N</i> must be an integer between 1 and 128. The default value for <i>N</i> is 8. The /MT parameter cannot be used with the /IPG and /EFSRAW parameters. Redirect output using /LOG option for better performance. Note: The /MT parameter applies to Windows Server 2008 R2 and Windows 7. |
| /rh:hhmm-hhmm | Specifies run times when new copies may be started. |
| /pf | Checks run times on a per-file (not per-pass) basis. |
| /ipg:n | Specifies the inter-packet gap to free bandwidth on slow lines. |
| /sl | Follows the symbolic link and copies the target. |

IMPORTANT

When using the **/SECFIX** copy option, specify the type of security information you want to copy by also using one of these additional copy options:

- > - **/COPYALL**
- > - **/COPY:O**
- > - **/COPY:S**
- > - **/COPY:U**
- > - **/SEC**

File selection options

| OPTION | DESCRIPTION |
|-----------------------|--|
| /a | Copies only files for which the Archive attribute is set. |
| /m | Copies only files for which the Archive attribute is set, and resets the Archive attribute. |
| /ia:[RASHCNETO] | Includes only files for which any of the specified attributes are set. |
| /xa:[RASHCNETO] | Excludes files for which any of the specified attributes are set. |
| /xf <FileName>[...] | Excludes files that match the specified names or paths. Note that <i>FileName</i> can include wildcard characters (* and ?). |
| /xd <Directory>[...] | Excludes directories that match the specified names and paths. |
| /xc | Excludes changed files. |
| /xn | Excludes newer files. |
| /xo | Excludes older files. |
| /xx | Excludes extra files and directories. |

| OPTION | DESCRIPTION |
|-------------|---|
| /xl | Excludes "lonely" files and directories. |
| /is | Includes the same files. |
| /it | Includes "tweaked" files. |
| /max:<N> | Specifies the maximum file size (to exclude files bigger than <i>N</i> bytes). |
| /min:<N> | Specifies the minimum file size (to exclude files smaller than <i>N</i> bytes). |
| /maxage:<N> | Specifies the maximum file age (to exclude files older than <i>N</i> days or date). |
| /minage:<N> | Specifies the minimum file age (exclude files newer than <i>N</i> days or date). |
| /maxlad:<N> | Specifies the maximum last access date (excludes files unused since <i>N</i>). |
| /minlad:<N> | Specifies the minimum last access date (excludes files used since <i>N</i>) If <i>N</i> is less than 1900, <i>N</i> specifies the number of days. Otherwise, <i>N</i> specifies a date in the format YYYYMMDD. |
| /xj | Excludes junction points, which are normally included by default. |
| /fft | Assumes FAT file times (two-second precision). |
| /dst | Compensates for one-hour DST time differences. |
| /xjd | Excludes junction points for directories. |
| /xjf | Excludes junction points for files. |

Retry options

| OPTION | DESCRIPTION |
|--------|---|
| /r:<N> | Specifies the number of retries on failed copies. The default value of <i>N</i> is 1,000,000 (one million retries). |
| /w:<N> | Specifies the wait time between retries, in seconds. The default value of <i>N</i> is 30 (wait time 30 seconds). |
| /reg | Saves the values specified in the /r and /w options as default settings in the registry. |
| /tbd | Specifies that the system will wait for share names to be defined (retry error 67). |

Logging options

| OPTION | DESCRIPTION |
|--------------------|--|
| /l | Specifies that files are to be listed only (and not copied, deleted, or time stamped). |
| /x | Reports all extra files, not just those that are selected. |
| /v | Produces verbose output, and shows all skipped files. |
| /ts | Includes source file time stamps in the output. |
| /fp | Includes the full path names of the files in the output. |
| /bytes | Prints sizes, as bytes. |
| /ns | Specifies that file sizes are not to be logged. |
| /nc | Specifies that file classes are not to be logged. |
| /nfl | Specifies that file names are not to be logged. |
| /ndl | Specifies that directory names are not to be logged. |
| /np | Specifies that the progress of the copying operation (the number of files or directories copied so far) will not be displayed. |
| /eta | Shows the estimated time of arrival (ETA) of the copied files. |
| /log:<LogFile> | Writes the status output to the log file (overwrites the existing log file). |
| /log+:<LogFile> | Writes the status output to the log file (appends the output to the existing log file). |
| /unicode | Displays the status output as Unicode text. |
| /unilog:<LogFile> | Writes the status output to the log file as Unicode text (overwrites the existing log file). |
| /unilog+:<LogFile> | Writes the status output to the log file as Unicode text (appends the output to the existing log file). |
| /tee | Writes the status output to the console window, as well as to the log file. |
| /njh | Specifies that there is no job header. |
| /njs | Specifies that there is no job summary. |

Job options

| OPTION | DESCRIPTION |
|-----------------|--|
| /job:<JobName> | Specifies that parameters are to be derived from the named job file. |
| /save:<JobName> | Specifies that parameters are to be saved to the named job file. |
| /quit | Quits after processing command line (to view parameters). |
| /nosd | Indicates that no source directory is specified. |
| /nodd | Indicates that no destination directory is specified. |
| /if | Includes the specified files. |

Exit (return) codes

| VALUE | DESCRIPTION |
|-------|--|
| 0 | No files were copied. No failure was encountered. No files were mismatched. The files already exist in the destination directory; therefore, the copy operation was skipped. |
| 1 | All files were copied successfully. |
| 2 | There are some additional files in the destination directory that are not present in the source directory. No files were copied. |
| 3 | Some files were copied. Additional files were present. No failure was encountered. |
| 5 | Some files were copied. Some files were mismatched. No failure was encountered. |
| 6 | Additional files and mismatched files exist. No files were copied and no failures were encountered. This means that the files already exist in the destination directory. |
| 7 | Files were copied, a file mismatch was present, and additional files were present. |
| 8 | Several files did not copy. |

NOTE

Any value greater than 8 indicates that there was at least one failure during the copy operation.

Remarks

- The **/mir** option is equivalent to the **/e** plus **/purge** options with one small difference in behavior:
 - With the **/e** plus **/purge** options, if the destination directory exists, the destination directory security settings are not overwritten.
 - With the **/mir** option, if the destination directory exists, the destination directory security settings are

overwritten.

Additional references

[Command-Line Syntax Key](#)

route_ws2008

7/11/2018 • 6 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Displays and modifies the entries in the local IP routing table. Used without parameters, **route** displays help.

Syntax

```
route [/f] [/p] [<Command> [<Destination>] [mask <Netmask>] [<Gateway>] [metric <Metric>]] [if <Interface>]]
```

Parameters

| PARAMETER | DESCRIPTION |
|----------------|--|
| /f | Clears the routing table of all entries that are not host routes (routes with a netmask of 255.255.255.255), the loopback network route (routes with a destination of 127.0.0.0 and a netmask of 255.0.0.0), or a multicast route (routes with a destination of 224.0.0.0 and a netmask of 240.0.0.0). If this is used in conjunction with one of the commands (such as add, change, or delete), the table is cleared prior to running the command. |
| /p | When used with the add command, the specified route is added to the registry and is used to initialize the IP routing table whenever the TCP/IP protocol is started. By default, added routes are not preserved when the TCP/IP protocol is started. When used with the print command, the list of persistent routes is displayed. This parameter is ignored for all other commands. Persistent routes are stored in the registry location HKEY_LOCAL_MACHINE\SYSTEM\CurrentControlSet\Services\Tcpip\Parameters\PersistentRoutes. |
| <Command> | Specifies the command you want to run. The following table lists valid commands: - add: adds a route. - change: modifies an existing route. - delete: deletes a route or routes. - print: prints a route or routes. |
| <Destination> | Specifies the network destination of the route. The destination can be an IP network address (where the host bits of the network address are set to 0), an IP address for a host route, or 0.0.0.0 for the default route. |
| mask <Netmask> | Specifies the network destination of the route. The destination can be an IP network address (where the host bits of the network address are set to 0), an IP address for a host route, or 0.0.0.0 for the default route. |

| PARAMETER | DESCRIPTION |
|-----------------|---|
| <Gateway> | Specifies the forwarding or next hop IP address over which the set of addresses defined by the network destination and subnet mask are reachable. For locally attached subnet routes, the gateway address is the IP address assigned to the interface that is attached to the subnet. For remote routes, available across one or more routers, the gateway address is a directly reachable IP address that is assigned to a neighboring router. |
| metric <Metric> | Specifies an integer cost metric (ranging from 1 to 9999) for the route, which is used when choosing among multiple routes in the routing table that most closely match the destination address of a packet being forwarded. The route with the lowest metric is chosen. The metric can reflect the number of hops, the speed of the path, path reliability, path throughput, or administrative properties. |
| if <Interface> | Specifies the interface index for the interface over which the destination is reachable. For a list of interfaces and their corresponding interface indexes, use the display of the route print command. You can use either decimal or hexadecimal values for the interface index. For hexadecimal values, precede the hexadecimal number with 0x. When the if parameter is omitted, the interface is determined from the gateway address. |
| /? | Displays help at the command prompt. |

Remarks

- Large values in the **metric** column of the routing table are the result of allowing TCP/IP to automatically determine the metric for routes in the routing table based on the configuration of IP address, subnet mask, and default gateway for each LAN interface. Automatic determination of the interface metric, enabled by default, determines the speed of each interface and adjusts the metrics of routes for each interface so that the fastest interface creates the routes with the lowest metric. To remove the large metrics, disable the automatic determination of the interface metric from the advanced properties of the TCP/IP protocol for each LAN connection.
- Names can be used for *Destination* if an appropriate entry exists in the local Networks file stored in the **systemroot\System32\Drivers\Etc** folder. Names can be used for the *gateway* as long as they can be resolved to an IP address through standard host name resolution techniques such as Domain Name System (DNS) queries, use of the local Hosts file stored in the **systemroot\system32\drivers\etc** folder, and NetBIOS name resolution.
- if the command is **print** or **delete**, the *Gateway* parameter can be omitted and wildcards can be used for the destination and gateway. The *Destination* value can be a wildcard value specified by an asterisk (*). If the destination specified contains an asterisk (*) or a question mark (?), it is treated as a wildcard and only matching destination routes are printed or deleted. The asterisk matches any string, and the question mark matches any single character. For example, 10.*.1, 192.168.*, 127.*, and *224* are all valid uses of the asterisk wildcard.
- Using an invalid combination of a destination and subnet mask (netmask) value displays a "Route: bad gateway address netmask" error message. This error message appears when the destination contains one or more bits set to 1 in bit locations where the corresponding subnet mask bit is set to 0. To test this condition, express the destination and subnet mask using binary notation. The subnet mask in binary notation consists of a series of 1 bits, representing the network address portion of the destination, and a series of 0 bits, representing the host address portion of the destination. Check to determine whether there are bits in the destination that are set to 1

for the portion of the destination that is the host address (as defined by the subnet mask).

- The **/p** parameter is only supported on the route command for Windows NT 4.0, Windows 2000, Windows Millennium edition, Windows XP, and Windows Server 2003. This parameter is not supported by the **route** command for Windows 95 or Windows 98.
- This command is available only if the Internet Protocol (TCP/IP) protocol is installed as a component in the properties of a network adapter in Network Connections.

Examples

To display the entire contents of the IP routing table, type:

```
route print
```

To display the routes in the IP routing table that begin with 10, type:

```
route print 10.*
```

To add a default route with the default gateway address of 192.168.12.1, type:

```
route add 0.0.0.0 mask 0.0.0.0 192.168.12.1
```

To add a route to the destination 10.41.0.0 with the subnet mask of 255.255.0.0 and the next hop address of 10.27.0.1, type:

```
route add 10.41.0.0 mask 255.255.0.0 10.27.0.1
```

To add a persistent route to the destination 10.41.0.0 with the subnet mask of 255.255.0.0 and the next hop address of 10.27.0.1, type:

```
route /p add 10.41.0.0 mask 255.255.0.0 10.27.0.1
```

To add a route to the destination 10.41.0.0 with the subnet mask of 255.255.0.0, the next hop address of 10.27.0.1, and the cost metric of 7, type:

```
route add 10.41.0.0 mask 255.255.0.0 10.27.0.1 metric 7
```

To add a route to the destination 10.41.0.0 with the subnet mask of 255.255.0.0, the next hop address of 10.27.0.1, and using the interface index 0x3, type:

```
route add 10.41.0.0 mask 255.255.0.0 10.27.0.1 if 0x3
```

To delete the route to the destination 10.41.0.0 with the subnet mask of 255.255.0.0, type:

```
route delete 10.41.0.0 mask 255.255.0.0
```

To delete all routes in the IP routing table that begin with 10, type:

```
route delete 10.*
```

To change the next hop address of the route with the destination of 10.41.0.0 and the subnet mask of 255.255.0.0 from 10.27.0.1 to 10.27.0.25, type:

```
route change 10.41.0.0 mask 255.255.0.0 10.27.0.25
```

additional references

- [Command-Line Syntax Key](#)

rpcinfo

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Lists programs on remote computers. The **rpcinfo** command-line utility makes a remote procedure call (RPC) to an RPC server and reports what it finds.

Syntax

```
rpcinfo [/p [<Node>]] [/b <Program version>] [/t <Node Program> [<version>]] [/u <Node Program> [<version>]]
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------------------------|--|
| /p [<Node>] | lists all programs registered with the port mapper on the specified host. If you do not specify a node (computer) name, the program queries the port mapper on the local host. |
| /b <Program version> | Requests a response from all network nodes that have the specified program and version registered with the port mapper. You must specify both a program name or number and a version number. |
| /t <Node Program> [<version>] | Uses the TCP transport protocol to call the specified program. You must specify both a node (computer) name and a program name. If you do not specify a version, the program calls all versions. |
| /u <Node Program> [<version>] | Uses the UDP transport protocol to call the specified program. You must specify both a node (computer) name and a program name. If you do not specify a version, the program calls all versions. |
| /? | Displays help at the command prompt. |

Examples

To list all programs registered with the port mapper, type:

```
rpcinfo /p [<Node>]
```

To request a response from network nodes that have a specified program, type:

```
rpcinfo /b <Program version>
```

To use Transmission Control Protocol (TCP) to call a program, type:

```
rpcinfo /t <Node Program> [<version>]
```

Use User Datagram Protocol (UDP) to call a program:

```
rpcinfo /u <Node Program> [<version>]
```

Additional references

- [Command-Line Syntax Key](#)

rpcping

7/11/2018 • 6 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Confirms the RPC connectivity between the computer running Microsoft Exchange Server and any of the supported Microsoft Exchange Client workstations on the network. This utility can be used to check if the Microsoft Exchange Server services are responding to RPC requests from the client workstations via the network.

Syntax

```
rpcping [/t <protseq>] [/s <server_addr>] [/e <endpoint>]
        [/f <interface UUID>[,Majorver]] [/O <Interface Object UUID>]
        [/i <#_iterations>] [/u <security_package_id>] [/a <authn_level>]
        [/N <server_princ_name>] [/I <auth_identity>] [/C <capabilities>]
        [/T <identity_tracking>] [/M <impersonation_type>]
        [/S <server_sid>] [/P <proxy_auth_identity>] [/F <RPCHTTP_flags>]
        [/H <RPC/HTTP_authn_schemes>] [/o <binding_options>]
        [/B <server_certificate_subject>] [/b] [/E] [/q] [/c]
        [/A <http_proxy_auth_identity>] [/U <HTTP_proxy_authn_schemes>]
        [/r <report_results_interval>] [/v <verbose_level>] [/d]
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------------------------|---|
| /t <protseq> | <p>Specifies the protocol sequence to use. Can be one of the standard RPC protocol sequences, for example: ncacn_ip_tcp, ncacn_np, or ncacn_http.</p> <p>If not specified, default is ncacn_ip_tcp.</p> |
| /s <server_addr> | <p>Specifies the server address. If not specified, the local machine will be pinged.</p> |
| /e <endpoint> | <p>Specifies the endpoint to ping. If none is specified, the endpoint mapper on the target machine will be pinged.</p> <p>This option is mutually exclusive with the interface (/f) option.</p> |
| /o <binding_options> | <p>Specifies the binding options for the RPC ping.</p> |
| /f <interface UUID>[,Majorver] | <p>Specifies the interface to ping. This option is mutually exclusive with the endpoint option. The interface is specified as a UUID.</p> <p>if the <i>Majorver</i> is not specified, version 1 of the interface will be sought.</p> <p>When interface is specified, rpcping will query the endpoint mapper on the target machine to retrieve the endpoint for the specified interface. The endpoint mapper will be queried using the options specified in the command line.</p> |

| PARAMETER | DESCRIPTION |
|--------------------------|---|
| /O <Object UUID> | Specifies the object UUID if the interface registered one. |
| /i <#_iterations> | Specifies the number of calls to make. The default is 1. This option is useful for measuring connection latency if multiple iterations are specified. |
| /u <security_package_id> | <p>Specifies the security package (security provider) RPC will use to make the call. The security package is identified as a number or a name. If a number is used it is the same number as in the RpcBindingSetAuthInfoEx API. The list below shows the names and numbers. Names are not case sensitive:</p> <ul style="list-style-type: none"> - Negotiate / 9 or one of nego, snego or negotiate - NTLM / 10 or NTLM - SChannel / 14 or SChannel - Kerberos / 16 or Kerberos - Kernel / 20 or Kernel <p>if you specify this option, you must specify authentication level other than none. There is no default for this option. If it is not specified, RPC will not use security for the ping.</p> |
| /a <authn_level> | <p>Specifies the authentication level to use. Possible values are:</p> <ul style="list-style-type: none"> - connect - call - pkt - integrity - privacy <p>if this option is specified, the security package ID (/u) must also be specified. There is no default for this option.</p> <p>if this option is not specified, RPC will not use security for the ping.</p> |
| /N <server_princ_name> | <p>Specifies a server principal name.</p> <p>This field can be used only when authentication level and security package are selected.</p> |
| /I <auth_identity> | <p>Allows you to specify alternative identity to connect to the server. The identity is in the form user,domain,password. If the user name, domain, or password have special characters that can be interpreted by the shell, enclose the identity in double quotes. You can specify * instead of the password and RPC will prompt you to enter the password without echoing it on the screen. If this field is not specified, the identity of the logged on user will be used.</p> <p>This field can be used only when authentication level and security package are selected.</p> |
| /C <capabilities> | <p>Specifies a hexadecimal bitmask of flags. This field can be used only when authentication level and security package are selected.</p> |

| PARAMETER | DESCRIPTION |
|---------------------------------|--|
| /T <identity_tracking> | <p>Specifies static or dynamic. If not specified, dynamic is the default.</p> <p>This field can be used only when authentication level and security package are selected.</p> |
| /M <impersonation_type> | <p>Specifies anonymous, identify, impersonate or delegate. Default is impersonate.</p> <p>This field can be used only when authentication level and security package are selected.</p> |
| /S <server_sid> | <p>Specifies the expected SID of the server.</p> <p>This field can be used only when authentication level and security package are selected.</p> |
| /P <proxy_auth_identity> | <p>Specifies the identity to authenticate with to the RPC/HTTP proxy. Has the same format as for the /I option. You must specify security package (/u), authentication level (/a), and authentication schemes (/H) in order to use this option.</p> |
| /F <RPCHTTP_flags> | <p>Specifies the flags to pass for RPC/HTTP front end authentication. The flags may be specified as numbers or names The currently recognized flags are:</p> <ul style="list-style-type: none"> - Use SSL / 1 or ssl or use_ssl - Use first auth scheme / 2 or first or use_first <p>You must specify security package (/u) and authentication level (/a) in order to use this option.</p> |
| /H <RPC/HTTP_authn_schemes> | <p>Specifies the authentication schemes to use for RPC/HTTP front end authentication. This option is a list of numerical values or names separated by comma. Example: Basic,NTLM. Recognized values are (names are not case sensitive):</p> <ul style="list-style-type: none"> - Basic / 1 or Basic - NTLM / 2 or NTLM - Certificate / 65536 or Cert <p>You must specify security package (/u) and authentication level (/a) in order to use this option.</p> |
| /B <server_certificate_subject> | <p>Specifies the server certificate subject. You must use SSL for this option to work.</p> <p>You must specify security package (/u) and authentication level (/a) in order to use this option.</p> |
| /b | <p>Retrieves the server certificate subject from the certificate sent by the server and prints it to a screen or a log file. Valid only when the Proxy echo only option (/E) and the use SSL options are specified.</p> <p>You must specify security package (/u) and authentication level (/a) in order to use this option.</p> |

| PARAMETER | DESCRIPTION |
|-----------|---|
| /R | Specifies the HTTP proxy. If <i>none</i> , the RPC proxy is used. The value <i>default</i> means to use the IE settings in your client machine. Any other value will be treated as the explicit HTTP proxy. If you do not specify this flag, the default value is assumed, that is, the IE settings are checked. This flag is valid only when the /E (echo Only) flag is enabled. |
| /E | <p>Restricts the ping to the RPC/HTTP proxy only. The ping does not reach the server. Useful when trying to establish whether the RPC/HTTP proxy is reachable. To specify an HTTP proxy, use the /R flag. If an HTTP proxy is specified in the /o flag, this option will be ignored.</p> <p>You must specify security package (/u) and authentication level (/a) in order to use this option.</p> |
| /q | Specifies quiet mode. Does not issue any prompts except for passwords. Assumes Y response to all queries. Use this option with care. |
| /c | Use smart card certificate. rpcping will prompt user to choose smart card. |
| /A | <p>Specifies the identity with which to authenticate to the HTTP proxy. Has the same format as for the /l option.</p> <p>You must specify authentication schemes (/U), security package (/u) and authentication level (/a) in order to use this option.</p> |
| /U | <p>Specifies the authentication schemes to use for HTTP proxy authentication. This option is a list of numerical values or names separated by comma. Example: Basic,NTLM. Recognized values are (names are not case sensitive):</p> <ul style="list-style-type: none"> - Basic / 1 or Basic - NTLM / 2 or NTLM <p>You must specify security package (/u) and authentication level (/a) in order to use this option.</p> |
| /r | if multiple iterations are specified, this option will make rpcping display current execution statistics periodically instead after the last call. The report interval is given in seconds. Default is 15. |
| /v | Tells rpcping how verbose to make the output. Default value is 1. 2 and 3 provide more output from rpcping . |
| /d | Launches RPC network diagnostic UI. |
| /p | Specifies to prompt for credentials if authentication fails. |
| /? | Displays help at the command prompt. |

Examples

To find out if your Exchange server that you connect through RPC/HTTP is accessible, type:

```
rpcping /t ncacn_http /s exchange_server /o RpcProxy=front_end_proxy /P "username,domain,*" /H Basic /u NTLM /a  
connect /F 3
```

Additional references

- [Command-Line Syntax Key](#)

rsh

4/13/2018 • 2 minutes to read • [Edit Online](#)

Runs commands on remote computers running the RSH service or daemon. This command has been deprecated. You can install the Subsystem for UNIX-based Applications using the Add Features Wizard. For more information, see [Windows Server 2008 UNIX Interoperability Components](#) at the Microsoft Web site. After installation, you can then open a C Shell or Korn Shell command window and run **rsh**. For more information, type **man rsh** at the C Shell or Korn Shell prompt.

rundll32

4/13/2018 • 2 minutes to read • [Edit Online](#)

Loads and runs 32-bit dynamic-link libraries (DLLs). There are no configurable settings for Rundll32. Help information is provided for a specific DLL you run with the **rundll32** command.

You must run the **rundll32** command from an elevated command prompt. To open an elevated command prompt, click **Start**, right-click **Command Prompt**, and then click **Run as administrator**.

Syntax

```
Rundll32 <DLLname>
```

Commands

| PARAMETER | DESCRIPTION |
|---|-------------------------------------|
| Rundll32 printui.dll,PrintUIEntry | Displays the printer user interface |

Remarks

Rundll32 can only call functions from a DLL that are explicitly written to be called by Rundll32. For more information about Rundll32 requirements see [article 164787](#) in the Microsoft Knowledge Base (<https://go.microsoft.com/fwlink/?LinkID=165773>).

Additional references

[Command-Line Syntax Key](#)

rwinsta

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Enables you to reset (delete) a session on a Remote Desktop Session Host (rd Session Host) server.

NOTE

In Windows Server 2008 R2, Terminal Services was renamed Remote Desktop Services. To find out what's new in the latest version, see [What's New in Remote Desktop Services in Windows Server 2012](#) in the Windows Server TechNet Library.

Remarks

This command is the same as the **reset session** command.

additional references

[reset session Command-Line Syntax Key Remote Desktop Services \(Terminal Services\) Command Reference](#)

schtasks

6/11/2018 • 62 minutes to read • [Edit Online](#)

Schedules commands and programs to run periodically or at a specific time. Adds and removes tasks from the schedule, starts and stops tasks on demand, and displays and changes scheduled tasks.

To view the command syntax, click one of the following commands:

- [schtasks create](#)
- [schtasks change](#)
- [schtasks run](#)
- [schtasks end](#)
- [schtasks delete](#)
- [schtasks query](#)

Remarks

- **SchTasks.exe** performs the same operations as **Scheduled Tasks** in **Control Panel**. You can use these tools together and interchangeably.
- **Schtasks** replaces **At.exe**, a tool included in previous versions of Windows. Although **At.exe** is still included in the Windows Server 2003 family, **schtasks** is the recommended command-line task scheduling tool.
- The parameters in a **schtasks** command can appear in any order. Typing **schtasks** without any parameters performs a query.
- Permissions for **schtasks**
 - You must have permission to run the command. Any user can schedule a task on the local computer, and they can view and change the tasks that they scheduled. Members of the Administrators group can schedule, view, and change all tasks on the local computer.
 - To schedule, view, or change a task on a remote computer, you must be member of the Administrators group on the remote computer, or you must use the **/u** parameter to provide the credentials of an Administrator of the remote computer.
 - You can use the **/u** parameter in a **/create** or **/change** operation only when the local and remote computers are in the same domain or the local computer is in a domain that the remote computer domain trusts. Otherwise, the remote computer cannot authenticate the user account specified and it cannot verify that the account is a member of the Administrators group.
 - The task must have permission to run. The permissions required vary with the task. By default, tasks run with the permissions of the current user of the local computer, or with the permissions of the user specified by the **/u** parameter, if one is included. To run a task with permissions of a different user account or with system permissions, use the **/ru** parameter.
- To verify that a scheduled task ran or to find out why a scheduled task did not run, see the Task Scheduler service transaction log, *SystemRoot\SchedLgU.txt*. This log records attempted runs initiated by all tools that use the service, including **Scheduled Tasks** and **SchTasks.exe**.
- On rare occasions, task files become corrupted. Corrupted tasks do not run. When you try to perform an operation on corrupted tasks, **SchTasks.exe** displays the following error message:

```
ERROR: The data is invalid.
```

You cannot recover corrupted tasks. To restore the task scheduling features of the system, use **SchTasks.exe** or **Scheduled Tasks** to delete the tasks from the system and reschedule them.

schtasks create

Schedules a task.

Schtasks uses different parameter combinations for each schedule type. To see the combined syntax for creating tasks or to see the syntax for creating a task with a particular schedule type, click one of the following options.

- [Combined syntax and parameter descriptions](#)
- [To schedule a task that runs every N minutes](#)
- [To schedule a task that runs every N hours](#)
- [To schedule a task that runs every N days](#)
- [To schedule a task that runs every N weeks](#)
- [To schedule a task that runs every N months](#)
- [To schedule a task that runs on a specific day of the week](#)
- [To schedule a task that runs on a specific week of the month](#)
- [To schedule a task that runs on a specific date each month](#)
- [To schedule a task that runs on the last day of a month](#)
- [To schedule a task that runs once](#)
- [To schedule a task that runs every time the system starts](#)
- [To schedule a task that runs when a user logs on](#)
- [To schedule a task that runs when the system is idle](#)
- [To schedule a task that runs now](#)
- [To schedule a task that runs with different permissions](#)
- [To schedule a task that runs with system permissions](#)
- [To schedule a task that runs more than one program](#)
- [To schedule a task that runs on a remote computer](#)

Combined syntax and parameter descriptions

Syntax

```
schtasks /create /sc <ScheduleType> /tn <TaskName> /tr <TaskRun> [/s <Computer> [/u [<Domain>\  
<User> [/p <Password>]]] [/ru {[<Domain>\  
<User> | System]} [/rp <Password>] [/mo <Modifier>] [/d <Day>[,<Day>...] | *]  
[/m <Month>[,<Month>...]] [/i <IdleTime>] [/st <StartTime>] [/ri <Interval>] [{/et <EndTime> | /du <Duration>}  
[/k]] [/sd <StartDate>] [/ed <EndDate>] [/it] [/z] [/f]
```

Parameters

/sc <ScheduleType>

Specifies the schedule type. Valid values are MINUTE, HOURLY, DAILY, WEEKLY, MONTHLY, ONCE, ONSTART, ONLOGON, ONIDLE.

| SCHEDULE TYPE | DESCRIPTION |
|--|--|
| MINUTE, HOURLY, DAILY, WEEKLY, MONTHLY | Specifies the time unit for the schedule. |
| ONCE | The task runs once at a specified date and time. |
| ONSTART | The task runs every time the system starts. You can specify a start date, or run the task the next time the system starts. |
| ONLOGON | The task runs whenever a user (any user) logs on. You can specify a date, or run the task the next time the user logs on. |

| SCHEDULE TYPE | DESCRIPTION |
|---------------|---|
| ONIDLE | The task runs whenever the system is idle for a specified period of time. You can specify a date, or run the task the next time the system is idle. |

/tn <TaskName>

Specifies a name for the task. Each task on the system must have a unique name. The name must conform to the rules for file names and must not exceed 238 characters. Use quotation marks to enclose names that include spaces.

/tr <TaskRun>

Specifies the program or command that the task runs. Type the fully qualified path and file name of an executable file, script file, or batch file. The path name must not exceed 262 characters. If you omit the path, **schtasks** assumes that the file is in the *SystemRoot\System32* directory.

/s <Computer>

Schedules a task on the specified remote computer. Type the name or IP address of a remote computer (with or without backslashes). The default is the local computer. The **/u** and **/p** parameters are valid only when you use **/s**.

/u [<Domain>]

Runs this command with the permissions of the specified user account. The default is the permissions of the current user of the local computer. The **/u** and **/p** parameters are valid only for scheduling a task on a remote computer (**/s**).

The permissions of the specified account are used to schedule the task and to run the task. To run the task with the permissions of a different user, use the **/ru** parameter.

The user account must be a member of the Administrators group on the remote computer. Also, the local computer must be in the same domain as the remote computer, or must be in a domain that is trusted by the remote computer domain.

/p <Password>

Provides the password for the user account specified in the **/u** parameter. If you use the **/u** parameter, but omit the **/p** parameter or the password argument, **schtasks** prompts you for a password and obscures the text you type.

The **/u** and **/p** parameters are valid only for scheduling a task on a remote computer (**/s**).

/ru [{<Domain>} | System]

Runs the task with permissions of the specified user account. By default, the task runs with the permissions of the current user of the local computer, or with the permission of the user specified by the **/u** parameter, if one is included. The **/ru** parameter is valid when scheduling tasks on local or remote computers.

| VALUE | DESCRIPTION |
|--------------|---|
| [<Domain>] | Specifies an alternate user account. |
| System or "" | Specifies the local System account, a highly privileged account used by the operating system and system services. |

/rp <Password>

Provides the password for the user account that is specified in the **/ru** parameter. If you omit this parameter when specifying a user account, **SchTasks.exe** prompts you for the password and obscures the text you type.

Do not use the **/rp** parameter for tasks run with System account credentials (**/ru System**). The System account does not have a password and **SchTasks.exe** does not prompt for one.

/mo <Modifier>

Specifies how often the task runs within its schedule type. This parameter is valid, but optional, for a MINUTE, HOURLY, DAILY, WEEKLY, and MONTHLY schedule. The default value is 1.

| SCHEDULE TYPE | MODIFIER VALUES | DESCRIPTION |
|---------------|------------------------------------|---|
| MINUTE | 1 - 1439 | The task runs every <N> minutes. |
| HOURLY | 1 - 23 | The task runs every <N> hours. |
| DAILY | 1 - 365 | The task runs every <N> days. |
| WEEKLY | 1 - 52 | The task runs every <N> weeks. |
| ONCE | No modifiers. | The task runs once. |
| ONSTART | No modifiers. | The task runs at startup. |
| ONLOGON | No modifiers. | The task runs when the user specified by the /u parameter logs on. |
| ONIDLE | No modifiers. | The task runs after the system is idle for the number of minutes specified by the /i parameter, which is required for use with ONIDLE. |
| MONTHLY | 1 - 12 | The task runs every <N> months. |
| MONTHLY | LASTDAY | The task runs on the last day of the month. |
| MONTHLY | FIRST, SECOND, THIRD, FOURTH, LAST | Use with the /d <Day> parameter to run a task on a particular week and day. For example, on the third Wednesday of the month. |

/d Day[,Day...] | *

Specifies a day (or days) of the week or a day (or days) of a month. Valid only with a WEEKLY or MONTHLY schedule.

| SCHEDULE TYPE | MODIFIER | DAY VALUES (/D) | DESCRIPTION |
|---------------|------------------------------------|--------------------------|--|
| WEEKLY | 1 - 52 | MON - SUN[,MON - SUN...] | * |
| MONTHLY | FIRST, SECOND, THIRD, FOURTH, LAST | MON - SUN | Required for a specific week schedule. |
| MONTHLY | None or {1 - 12} | 1 - 31 | Optional and valid only with no modifier (/mo) parameter (a specific date schedule) or when the /mo is 1 - 12 (an "every <N> months" schedule). The default is day 1 (the first day of the month). |

/m Month[,Month...]

Specifies a month or months of the year during which the scheduled task should run. Valid values are JAN - DEC and * (every month). The **/m** parameter is valid only with a MONTHLY schedule. It is required when the LASTDAY modifier is used. Otherwise, it is optional and the default value is * (every month).

`/i <IdleTime>`

Specifies how many minutes the computer is idle before the task starts. A valid value is a whole number from 1 to 999. This parameter is valid only with an ONIDLE schedule, and then it is required.

`/st <StartTime>`

Specifies the time of day that the task starts (each time it starts) in `<HH:MM>` 24-hour format. The default value is the current time on the local computer. The `/st` parameter is valid with MINUTE, HOURLY, DAILY, WEEKLY, MONTHLY, and ONCE schedules. It is required for a ONCE schedule.

`/ri <Interval>`

Specifies the repetition interval in minutes. This is not applicable for schedule types: MINUTE, HOURLY, ONSTART, ONLOGON, and ONIDLE. Valid range is 1 to 599940 minutes (599940 minutes = 9999 hours). If either `/ET` or `/DU` is specified, then the repetition interval defaults to 10 minutes.

`/et <EndTime>`

Specifies the time of day that a minute or hourly task schedule ends in `<HH:MM>` 24-hour format. After the specified end time, **schtasks** does not start the task again until the start time recurs. By default, task schedules have no end time. This parameter is optional and valid only with a MINUTE or HOURLY schedule.

For an example, see:

- "To schedule a task that runs every 100 minutes during non-business hours" in the **To schedule a task that runs every <N> minutes** section.

`/du <Duration>`

Specifies a maximum length of time for a minute or hourly schedule in `<HHHH:MM>` 24-hour format. After the specified time elapses, **schtasks** does not start the task again until the start time recurs. By default, task schedules have no maximum duration. This parameter is optional and valid only with a MINUTE or HOURLY schedule.

For an example, see:

- "To schedule a task that runs every 3 hours for 10 hours" in the **To schedule a task that runs every <N> hours** section.

`/k`

Stops the program that the task runs at the time specified by `/et` or `/du`. Without `/k`, **schtasks** does not start the program again after it reaches the time specified by `/et` or `/du`, but it does not stop the program if it is still running. This parameter is optional and valid only with a MINUTE or HOURLY schedule.

For an example, see:

- "To schedule a task that runs every 100 minutes during non-business hours" in the **To schedule a task that runs every <N> minutes** section.

`/sd <StartDate>`

Specifies the date on which the task schedule starts. The default value is the current date on the local computer. The `/sd` parameter is valid and optional for all schedule types.

The format for *StartDate* varies with the locale selected for the local computer in **Regional and Language Options** in **Control Panel**. Only one format is valid for each locale.

The valid date formats are listed in the following table. Use the format most similar to the format selected for **Short date** in **Regional and Language Options** in **Control Panel** on the local computer.

| VALUE | DESCRIPTION |
|--------------------------|---|
| <code><MM>/</code> | Use for month-first formats, such as English (United States) and Spanish (Panama) . |

| VALUE | DESCRIPTION |
|-----------|--|
| <DD> // | Use for day-first formats, such as Bulgarian and Dutch (Netherlands) . |
| <YYYY> // | Use for year-first formats, such as Swedish and French (Canada) . |

/ed <EndDate>

Specifies the date on which the schedule ends. This parameter is optional. It is not valid in a ONCE, ONSTART, ONLOGON, or ONIDLE schedule. By default, schedules have no ending date.

The format for *EndDate* varies with the locale selected for the local computer in **Regional and Language Options** in **Control Panel**. Only one format is valid for each locale.

The valid date formats are listed in the following table. Use the format most similar to the format selected for **Short date** in **Regional and Language Options** in **Control Panel** on the local computer.

| VALUE | DESCRIPTION |
|-----------|---|
| <MM> / | Use for month-first formats, such as English (United States) and Spanish (Panama) . |
| <DD> // | Use for day-first formats, such as Bulgarian and Dutch (Netherlands) . |
| <YYYY> // | Use for year-first formats, such as Swedish and French (Canada) . |

/it

Specifies to run the task only when the "run as" user (the user account under which the task runs) is logged on to the computer. This parameter has no effect on tasks that run with system permissions.

By default, the "run as" user is the current user of the local computer when the task is scheduled or the account specified by the **/u** parameter, if one is used. However, if the command includes the **/ru** parameter, then the "run as" user is the account specified by the **/ru** parameter.

For examples, see:

- "To schedule a task that runs every 70 days if I am logged on" in the **To schedule a task that runs every N days** section.
- "To run a task only when a particular user is logged on" in the **To schedule a task that runs with different permissions** section.

/z

Specifies to delete the task upon completion of its schedule.

/f

Specifies to create the task and suppress warnings if the specified task already exists.

/?

Displays help at the command prompt.

To schedule a task that runs every N minutes

Minute Schedule Syntax

```
schtasks /create /tn <TaskName> /tr <TaskRun> /sc minute [/mo {1 - 1439}] [/st <HH:MM>] [/sd <StartDate>] [/ed <EndDate>] [{/et <HH:MM> | /du <HHHH:MM>} [/k]] [/it] [/ru {[<Domain>\]<User> [/rp <Password>] | System}] [/s <Computer> [/u [<Domain>\]<User> [/p <Password>]]]
```

Remarks

In a minute schedule, the **/sc minute** parameter is required. The **/mo** (modifier) parameter is optional and specifies the number of minutes between each run of the task. The default value for **/mo** is 1 (every minute). The **/et** (end time) and **/du** (duration) parameters are optional and can be used with or without the **/k** (end task) parameter.

Examples

To schedule a task that runs every 20 minutes

The following command schedules a security script, Sec.vbs, to run every 20 minutes. The command uses the **/sc** parameter to specify a minute schedule and the **/mo** parameter to specify an interval of 20 minutes.

Because the command does not include a starting date or time, the task starts 20 minutes after the command completes, and runs every 20 minutes thereafter whenever the system is running. Notice that the security script source file is located on a remote computer, but that the task is scheduled and executes on the local computer.

```
schtasks /create /sc minute /mo 20 /tn "Security Script" /tr \\central\data\scripts\sec.vbs
```

To schedule a task that runs every 100 minutes during non-business hours

The following command schedules a security script, Sec.vbs, to run on the local computer every 100 minutes between 5:00 P.M. and 7:59 A.M. each day. The command uses the **/sc** parameter to specify a minute schedule and the **/mo** parameter to specify an interval of 100 minutes. It uses the **/st** and **/et** parameters to specify the start time and end time of each day's schedule. It also uses the **/k** parameter to stop the script if it is still running at 7:59 A.M. Without **/k**, **schtasks** would not start the script after 7:59 A.M., but if the instance started at 6:20 A.M. was still running, it would not stop it.

```
schtasks /create /tn "Security Script" /tr sec.vbs /sc minute /mo 100 /st 17:00 /et 08:00 /k
```

To schedule a task that runs every N hours

Hourly Schedule Syntax

```
schtasks /create /tn <TaskName> /tr <TaskRun> /sc hourly [/mo {1 - 23}] [/st <HH:MM>] [/sd <StartDate>] [/ed <EndDate>] [{/et <HH:MM> | /du <HHHH:MM>} [/k]] [/it] [/ru {[<Domain>\]<User> [/rp <Password>] | System}] [/s <Computer> [/u [<Domain>\]<User> [/p <Password>]]]
```

Remarks

In an hourly schedule, the **/sc hourly** parameter is required. The **/mo** (modifier) parameter is optional and specifies the number of hours between each run of the task. The default value for **/mo** is 1 (every hour). The **/k** (end task) parameter is optional and can be used with either **/et** (end at the specified time) or **/du** (end after the specified interval).

Examples

To schedule a task that runs every five hours

The following command schedules the MyApp program to run every five hours beginning on the first day of March 2002. It uses the **/mo** parameter to specify the interval and the **/sd** parameter to specify the start date. Because the command does not specify a start time, the current time is used as the start time.

Because the local computer is set to use the **English (Zimbabwe)** option in **Regional and Language Options** in **Control Panel**, the format for the start date is MM/DD/YYYY (03/01/2002).

```
schtasks /create /sc hourly /mo 5 /sd 03/01/2002 /tn "My App" /tr c:\apps\myapp.exe
```

To schedule a task that runs every hour at five minutes past the hour

The following command schedules the MyApp program to run hourly beginning at five minutes past midnight. Because the **/mo** parameter is omitted, the command uses the default value for the hourly schedule, which is every (1) hour. If this command runs after 12:05 A.M., the program does not run until the next day.

```
schtasks /create /sc hourly /st 00:05 /tn "My App" /tr c:\apps\myapp.exe
```

To schedule a task that runs every 3 hours for 10 hours

The following command schedules the MyApp program to run every 3 hours for 10 hours.

The command uses the **/sc** parameter to specify an hourly schedule and the **/mo** parameter to specify the interval of 3 hours. It uses the **/st** parameter to start the schedule at midnight and the **/du** parameter to end the recurrences after 10 hours. Because the program runs for just a few minutes, the **/k** parameter, which stops the program if it is still running when the duration expires, is not necessary.

```
schtasks /create /tn "My App" /tr myapp.exe /sc hourly /mo 3 /st 00:00 /du 0010:00
```

In this example, the task runs at 12:00 A.M., 3:00 A.M., 6:00 A.M., and 9:00 A.M. Because the duration is 10 hours, the task is not run again at 12:00 P.M. Instead, it starts again at 12:00 A.M. the next day.

To schedule a task that runs every N days

Daily Schedule Syntax

```
schtasks /create /tn <TaskName> /tr <TaskRun> /sc daily [/mo {1 - 365}] [/st <HH:MM>] [/sd <StartDate>] [/ed <EndDate>] [/it] [/ru {[<Domain>\]<User> [/rp <Password>] | System}] [/s <Computer> [/u [<Domain>\]<User> [/p <Password>]]]
```

Remarks

In a daily schedule, the **/sc daily** parameter is required. The **/mo** (modifier) parameter is optional and specifies the number of days between each run of the task. The default value for **/mo** is 1 (every day).

Examples

To schedule a task that runs every day

The following example schedules the MyApp program to run once a day, every day, at 8:00 A.M. until December 31, 2002. Because it omits the **/mo** parameter, the default interval of 1 is used to run the command every day.

In this example, because the local computer system is set to the **English (United Kingdom)** option in **Regional and Language Options** in **Control Panel**, the format for the end date is DD/MM/YYYY (31/12/2002)

```
schtasks /create /tn "My App" /tr c:\apps\myapp.exe /sc daily /st 08:00 /ed 31/12/2002
```

To schedule a task that runs every 12 days

The following example schedules the MyApp program to run every twelve days at 1:00 P.M. (13:00) beginning on December 31, 2002. The command uses the **/mo** parameter to specify an interval of two (2) days and the **/sd** and **/st** parameters to specify the date and time.

In this example, because the system is set to the **English (Zimbabwe)** option in **Regional and Language Options** in **Control Panel**, the format for the end date is MM/DD/YYYY (12/31/2002)

```
schtasks /create /tn "My App" /tr c:\apps\myapp.exe /sc daily /mo 12 /sd 12/31/2002 /st 13:00
```

To schedule a task that runs every 70 days if I am logged on

The following command schedules a security script, Sec.vbs, to run every 70 days. The command uses the **/mo** parameter to specify an interval of 70 days. It also uses the **/it** parameter to specify that the task runs only when the user under whose account the task runs is logged onto the computer. Because the task will run with the permissions of my user account, then the task will run only when I am logged on.

```
schtasks /create /tn "Security Script" /tr sec.vbs /sc daily /mo 70 /it
```

NOTE

To identify tasks with the interactive-only (**/it**) property, use a verbose query (**/query /v**). In a verbose query display of a task with **/it**, the **Logon Mode** field has a value of **Interactive only**.

To schedule a task that runs every N weeks

Weekly Schedule Syntax

```
schtasks /create /tn <TaskName> /tr <TaskRun> /sc weekly [/mo {1 - 52}] [/d {<MON - SUN>[,MON - SUN...] | *}] [/st <HH:MM>] [/sd <StartDate>] [/ed <EndDate>] [/it] [/ru {[<Domain>\]<User> [/rp <Password>] | System}] [/s <Computer>] [/u [<Domain>\]<User> [/p <Password>]]
```

Remarks

In a weekly schedule, the **/sc weekly** parameter is required. The **/mo** (modifier) parameter is optional and specifies the number of weeks between each run of the task. The default value for **/mo** is 1 (every week).

Weekly schedules also have an optional **/d** parameter to schedule the task to run on specified days of the week, or on all days (). *The default is MON (Monday). The every day () option is equivalent to scheduling a daily task.*

Examples

To schedule a task that runs every six weeks

The following command schedules the MyApp program to run on a remote computer every six weeks. The command uses the **/mo** parameter to specify the interval. Because the command omits the **/d** parameter, the task runs on Mondays.

This command also uses the **/s** parameter to specify the remote computer and the **/u** parameter to run the command with the permissions of the user's Administrator account. Because the **/p** parameter is omitted, **SchTasks.exe** prompts the user for the Administrator account password.

Also, because the command is run remotely, all paths in the command, including the path to MyApp.exe, refer to paths on the remote computer.

```
schtasks /create /tn "My App" /tr c:\apps\myapp.exe /sc weekly /mo 6 /s Server16 /u Admin01
```

To schedule a task that runs every other week on Friday

The following command schedules a task to run every other Friday. It uses the **/mo** parameter to specify the two-week interval and the **/d** parameter to specify the day of the week. To schedule a task that runs every Friday, omit the **/mo** parameter or set it to 1.

```
schtasks /create /tn "My App" /tr c:\apps\myapp.exe /sc weekly /mo 2 /d FRI
```

To schedule a task that runs every N months

Syntax

```
schtasks /create /tn <TaskName> /tr <TaskRun> /sc monthly [/mo {1 - 12}] [/d {1 - 31}] [/st <HH:MM>] [/sd <StartDate>] [/ed <EndDate>] [/it] [/ru {[<Domain>\]<User> [/rp <Password>] | System}] [/s <Computer> [/u [<Domain>\]<User> [/p <Password>]]]
```

Remarks

In this schedule type, the **/sc monthly** parameter is required. The **/mo** (modifier) parameter, which specifies the number of months between each run of the task, is optional and the default is 1 (every month). This schedule type also has an optional **/d** parameter to schedule the task to run on a specified date of the month. The default is 1 (the first day of the month).

Examples

To schedule a task that runs on the first day of every month

The following command schedules the MyApp program to run on the first day of every month. Because a value of 1 is the default for both the **/mo** (modifier) parameter and the **/d** (day) parameter, these parameters are omitted from the command.

```
schtasks /create /tn "My App" /tr myapp.exe /sc monthly
```

To schedule a task that runs every three months

The following command schedules the MyApp program to run every three months. It uses the **/mo** parameter to specify the interval.

```
schtasks /create /tn "My App" /tr c:\apps\myapp.exe /sc monthly /mo 3
```

To schedule a task that runs at midnight on the 21st day of every other month

The following command schedules the MyApp program to run every other month on the 21st day of the month at midnight. The command specifies that this task should run for one year, from July 2, 2002 to June 30, 2003.

The command uses the **/mo** parameter to specify the monthly interval (every two months), the **/d** parameter to specify the date, and the **/st** to specify the time. It also uses the **/sd** and **/ed** parameters to specify the start date and end date, respectively. Because the local computer is set to the **English (South Africa)** option in **Regional and Language Options** in **Control Panel**, the dates are specified in the local format, YYYY/MM/DD.

```
schtasks /create /tn "My App" /tr c:\apps\myapp.exe /sc monthly /mo 2 /d 21 /st 00:00 /sd 2002/07/01 /ed 2003/06/30
```

To schedule a task that runs on a specific day of the week

Weekly Schedule Syntax

```
schtasks /create /tn <TaskName> /tr <TaskRun> /sc weekly [/d {<MON - SUN>[,MON - SUN...] | *}] [/mo {1 - 52}] [/st <HH:MM>] [/sd <StartDate>] [/ed <EndDate>] [/it] [/ru {[<Domain>\]<User> [/rp <Password>] | System}] [/s <Computer> [/u [<Domain>\]<User> [/p <Password>]]]
```

Remarks

The "day of the week" schedule is a variation of the weekly schedule. In a weekly schedule, the **/sc weekly** parameter is required. The **/mo** (modifier) parameter is optional and specifies the number of weeks between each run of the task. The default value for **/mo** is 1 (every week). The **/d** parameter, which is optional, schedules the task to run on specified days of the week, or on all days (.). *The default is MON (Monday). The every day option (* /d ***) is equivalent to scheduling a daily task.*

Examples

To schedule a task that runs every Wednesday

The following command schedules the MyApp program to run every week on Wednesday. The command uses the

/d parameter to specify the day of the week. Because the command omits the **/mo** parameter, the task runs every week.

```
schtasks /create /tn "My App" /tr c:\apps\myapp.exe /sc weekly /d WED
```

To schedule a task that runs every eight weeks on Monday and Friday

The following command schedules a task to run on Monday and Friday of every eighth week. It uses the **/d** parameter to specify the days and the **/mo** parameter to specify the eight-week interval.

```
schtasks /create /tn "My App" /tr c:\apps\myapp.exe /sc weekly /mo 8 /d MON,FRI
```

To schedule a task that runs on a specific week of the month

Specific Week Syntax

```
schtasks /create /tn <TaskName> /tr <TaskRun> /sc monthly /mo {FIRST | SECOND | THIRD | FOURTH | LAST} /d MON - SUN [/m {JAN - DEC[,JAN - DEC...] | *}] [/st <HH:MM>] [/sd <StartDate>] [/ed <EndDate>] [/it] [/ru {[<Domain>\] <User> [/rp <Password>] | System}] [/s <Computer> [/u [<Domain>\] <User> [/p <Password>]]]
```

Remarks

In this schedule type, the **/sc monthly** parameter, the **/mo** (modifier) parameter, and the **/d** (day) parameter are required. The **/mo** (modifier) parameter specifies the week on which the task runs. The **/d** parameter specifies the day of the week. (You can specify only one day of the week for this schedule type.) This schedule also has an optional **/m** (month) parameter that lets you schedule the task for particular months or every month (.). *The default for the ****/m*** parameter is every month (*).*

Examples

To schedule a task for the second Sunday of every month

The following command schedules the MyApp program to run on the second Sunday of every month. It uses the **/mo** parameter to specify the second week of the month and the **/d** parameter to specify the day.

```
schtasks /create /tn "My App" /tr c:\apps\myapp.exe /sc monthly /mo SECOND /d SUN
```

To schedule a task for the first Monday in March and September

The following command schedules the MyApp program to run on the first Monday in March and September. It uses the **/mo** parameter to specify the first week of the month and the **/d** parameter to specify the day. It uses **/m** parameter to specify the month, separating the month arguments with a comma.

```
schtasks /create /tn "My App" /tr c:\apps\myapp.exe /sc monthly /mo FIRST /d MON /m MAR,SEP
```

To schedule a task that runs on a specific date each month

Specific date syntax

```
schtasks /create /tn <TaskName> /tr <TaskRun> /sc monthly /d {1 - 31} [/m {JAN - DEC[,JAN - DEC...] | *}] [/st <HH:MM>] [/sd <StartDate>] [/ed <EndDate>] [/it] [/ru {[<Domain>\] <User> [/rp <Password>] | System}] [/s <Computer> [/u [<Domain>\] <User> [/p <Password>]]]
```

Remarks

In the specific date schedule type, the **/sc monthly** parameter and the **/d** (day) parameter are required. The **/d** parameter specifies a date of the month (1 - 31), not a day of the week. You can specify only one day in the schedule. The **/mo** (modifier) parameter is not valid with this schedule type.

The **/m** (month) parameter is optional for this schedule type and the default is every month (.). ***Schtasks** does not

let you schedule a task for a date that does not occur in a month specified by the **/m** parameter. However, if omit the **/m** parameter, and schedule a task for a date that does not appear in every month, such as the 31st day, then the task does not run in the shorter months. To schedule a task for the last day of the month, use the last day schedule type.

Examples

To schedule a task for the first day of every month

The following command schedules the MyApp program to run on the first day of every month. Because the default modifier is none (no modifier), the default day is day 1, and the default month is every month, the command does not need any additional parameters.

```
schtasks /create /tn "My App" /tr c:\apps\myapp.exe /sc monthly
```

To schedule a task for the 15th days of May and June

The following command schedules the MyApp program to run on May 15 and June 15 at 3:00 P.M. (15:00). It uses the **/m** parameter to specify the date and the **/m** parameter to specify the months. It also uses the **/st** parameter to specify the start time.

```
schtasks /create /tn "My App" /tr c:\apps\myapp.exe /sc monthly /d 15 /m MAY,JUN /st 15:00
```

To schedule a task that runs on the last day of a month

Last day syntax

```
schtasks /create /tn <TaskName> /tr <TaskRun> /sc monthly /mo LASTDAY /m {JAN - DEC[,JAN - DEC...] | *} [/st <HH:MM>] [/sd <StartDate>] [/ed <EndDate>] [/it] [/ru {[<Domain>\]<User> [/rp <Password>] | System}] [/s <Computer> [/u [<Domain>\]<User> [/p <Password>]]]
```

Remarks

In the last day schedule type, the **/sc monthly** parameter, the **/mo LASTDAY** (modifier) parameter, and the **/m** (month) parameter are required. The **/d** (day) parameter is not valid.

Examples

To schedule a task for the last day of every month

The following command schedules the MyApp program to run on the last day of every month. It uses the **/mo** parameter to specify the last day and the **/m** parameter with the wildcard character (*) to indicate that the program runs every month.

```
schtasks /create /tn "My App" /tr c:\apps\myapp.exe /sc monthly /mo lastday /m *
```

To schedule a task at 6:00 P.M. on the last days of February and March

The following command schedules the MyApp program to run on the last day of February and the last day of March at 6:00 P.M. It uses the **/mo** parameter to specify the last day, the **/m** parameter to specify the months, and the **/st** parameter to specify the start time.

```
schtasks /create /tn "My App" /tr c:\apps\myapp.exe /sc monthly /mo lastday /m FEB,MAR /st 18:00
```

To schedule a task that runs once

Syntax

```
schtasks /create /tn <TaskName> /tr <TaskRun> /sc once /st <HH:MM> [/sd <StartDate>] [/it] [/ru {[<Domain>\]<User> [/rp <Password>] | System}] [/s <Computer> [/u [<Domain>\]<User> [/p <Password>]]]
```

Remarks

In the run-once schedule type, the **/sc once** parameter is required. The **/st** parameter, which specifies the time that the task runs, is required. The **/sd** parameter, which specifies the date that the task runs, is optional. The **/mo** (modifier) and **/ed** (end date) parameters are not valid for this schedule type.

Schtasks does not permit you to schedule a task to run once if the date and time specified are in the past, based on the time of the local computer. To schedule a task that runs once on a remote computer in a different time zone, you must schedule it before that date and time occurs on the local computer.

Examples

To schedule a task that runs one time

The following command schedules the MyApp program to run at midnight on January 1, 2003. It uses the **/sc** parameter to specify the schedule type and the **/sd** and **st** to specify the date and time.

Because the local computer uses the **English (United States)** option in **Regional and Language Options** in **Control Panel**, the format for the start date is MM/DD/YYYY.

```
schtasks /create /tn "My App" /tr c:\apps\myapp.exe /sc once /sd 01/01/2003 /st 00:00
```

To schedule a task that runs every time the system starts

Syntax

```
schtasks /create /tn <TaskName> /tr <TaskRun> /sc onstart [/sd <StartDate>] [/it] [/ru {[<Domain>\]<User> [/rp <Password>] | System}] [/s <Computer> [/u [<Domain>\]<User> [/p <Password>]]]
```

Remarks

In the on-start schedule type, the **/sc onstart** parameter is required. The **/sd** (start date) parameter is optional and the default is the current date.

Examples

To schedule a task that runs when the system starts

The following command schedules the MyApp program to run every time the system starts, beginning on March 15, 2001:

Because the local computer is uses the **English (United States)** option in **Regional and Language Options** in **Control Panel**, the format for the start date is MM/DD/YYYY.

```
schtasks /create /tn "My App" /tr c:\apps\myapp.exe /sc onstart /sd 03/15/2001
```

To schedule a task that runs when a user logs on

Syntax

```
schtasks /create /tn <TaskName> /tr <TaskRun> /sc onlogon [/sd <StartDate>] [/it] [/ru {[<Domain>\]<User> [/rp <Password>] | System}] [/s <Computer> [/u [<Domain>\]<User> [/p <Password>]]]
```

Remarks

The "on logon" schedule type schedules a task that runs whenever any user logs on to the computer. In the "on logon" schedule type, the **/sc onlogon** parameter is required. The **/sd** (start date) parameter is optional and the default is the current date.

Examples

To schedule a task that runs when a user logs on to a remote computer

The following command schedules a batch file to run every time a user (any user) logs on to the remote computer. It uses the **/s** parameter to specify the remote computer. Because the command is remote, all paths in the command, including the path to the batch file, refer to a path on the remote computer.

```
schtasks /create /tn "Start Web Site" /tr c:\myiis\webstart.bat /sc onlogon /s Server23
```

To schedule a task that runs when the system is idle

Syntax

```
schtasks /create /tn <TaskName> /tr <TaskRun> /sc onidle /i {1 - 999} [/sd <StartDate>] [/it] [/ru {[<Domain>\]  
<User> [/rp <Password>] | System}] [/s <Computer> [/u [<Domain>\]<User> [/p <Password>]]]
```

Remarks

The "on idle" schedule type schedules a task that runs whenever there is no user activity during the time specified by the **/i** parameter. In the "on idle" schedule type, the **/sc onidle** parameter and the **/i** parameter are required. The **/sd** (start date) is optional and the default is the current date.

Examples

To schedule a task that runs whenever the computer is idle

The following command schedules the MyApp program to run whenever the computer is idle. It uses the required **/i** parameter to specify that the computer must remain idle for ten minutes before the task starts.

```
schtasks /create /tn "My App" /tr c:\apps\myapp.exe /sc onidle /i 10
```

To schedule a task that runs now

Schtasks does not have a "run now" option, but you can simulate that option by creating a task that runs once and starts in a few minutes.

Syntax

```
schtasks /create /tn <TaskName> /tr <TaskRun> /sc once [/st <HH:MM>] /sd <MM/DD/YYYY> [/it] [/ru {[<Domain>\]  
<User> [/rp <Password>] | System}] [/s <Computer> [/u [<Domain>\]<User> [/p <Password>]]]
```

Examples

To schedule a task that runs a few minutes from now.

The following command schedules a task to run once, on November 13, 2002 at 2:18 P.M. local time.

Because the local computer is uses the **English (United States)** option in **Regional and Language Options** in **Control Panel**, the format for the start date is MM/DD/YYYY.

```
schtasks /create /tn "My App" /tr c:\apps\myapp.exe /sc once /st 14:18 /sd 11/13/2002
```

To schedule a task that runs with different permissions

You can schedule tasks of all types to run with permissions of an alternate account on both the local and a remote computer. In addition to the parameters required for the particular schedule type, the **/ru** parameter is required and the **/rp** parameter is optional.

Examples

To run a task with Administrator permissions on the local computer

The following command schedules the MyApp program to run on the local computer. It uses the **/ru** to specify that the task should run with the permissions of the user's Administrator account (Admin06). In this example, the task is scheduled to run every Tuesday, but you can use any schedule type for a task run with alternate permissions.

```
schtasks /create /tn "My App" /tr myapp.exe /sc weekly /d TUE /ru Admin06
```

In response, **SchTasks.exe** prompts for the "run as" password for the Admin06 account and then displays a

success message.

```
Please enter the run as password for Admin06: *****
SUCCESS: The scheduled task "My App" has successfully been created.
```

To run a task with alternate permissions on a remote computer

The following command schedules the MyApp program to run on the Marketing computer every four days.

The command uses the **/sc** parameter to specify a daily schedule and **/mo** parameter to specify an interval of four days.

The command uses the **/s** parameter to provide the name of the remote computer and the **/u** parameter to specify an account with permission to schedule a task on the remote computer (Admin01 on the Marketing computer). It also uses the **/ru** parameter to specify that the task should run with the permissions of the user's non-Administrator account (User01 in the Reskits domain). Without the **/ru** parameter, the task would run with the permissions of the account specified by **/u**.

```
schtasks /create /tn "My App" /tr myapp.exe /sc daily /mo 4 /s Marketing /u Marketing\Admin01 /ru
Reskits\User01
```

Schtasks first requests the password of the user named by the **/u** parameter (to run the command) and then requests the password of the user named by the **/ru** parameter (to run the task). After authenticating the passwords, **schtasks** displays a message indicating that the task is scheduled.

```
Type the password for Marketing\Admin01:*****

Please enter the run as password for Reskits\User01: *****

SUCCESS: The scheduled task "My App" has successfully been created.
```

To run a task only when a particular user is logged on

The following command schedules the AdminCheck.exe program to run on the Public computer every Friday at 4:00 A.M., but only if the administrator of the computer is logged on.

The command uses the **/sc** parameter to specify a weekly schedule, the **/d** parameter to specify the day, and the **/st** parameter to specify the start time.

The command uses the **/s** parameter to provide the name of the remote computer and the **/u** parameter to specify an account with permission to schedule a task on the remote computer. It also uses the **/ru** parameter to configure the task to run with the permissions of the administrator of the Public computer (Public\Admin01) and the **/it** parameter to indicate that the task runs only when the Public\Admin01 account is logged on.

```
schtasks /create /tn "Check Admin" /tr AdminCheck.exe /sc weekly /d FRI /st 04:00 /s Public /u Domain3\Admin06
/ru Public\Admin01 /it
```

Note

- To identify tasks with the interactive-only (**/it**) property, use a verbose query (**/query /v**). In a verbose query display of a task with **/it**, the **Logon Mode** field has a value of **Interactive only**.

To schedule a task that runs with system permissions

Tasks of all types can run with permissions of the System account on both the local and a remote computer. In addition to the parameters required for the particular schedule type, the **/ru system** (or **/ru ""**) parameter is required and the **/rp** parameter is not valid.

Important

- The System account does not have interactive logon rights. Users cannot see or interact with programs or tasks run with system permissions.
- The **/ru** parameter determines the permissions under which the task runs, not the permissions used to schedule the task. Only Administrators can schedule tasks, regardless of the value of the **/ru** parameter.

Note

To identify tasks that run with system permissions, use a verbose query (**/query /v**). In a verbose query display of a system-run task, the **Run As User** field has a value of **NT AUTHORITY\SYSTEM** and the **Logon Mode** field has a value of **Background only**.

Examples

To run a task with system permissions

The following command schedules the MyApp program to run on the local computer with permissions of the System account. In this example, the task is scheduled to run on the fifteenth day of every month, but you can use any schedule type for a task run with system permissions.

The command uses the **/ru System** parameter to specify the system security context. Because system tasks do not use a password, the **/rp** parameter is omitted.

```
schtasks /create /tn "My App" /tr c:\apps\myapp.exe /sc monthly /d 15 /ru System
```

In response, **SchTasks.exe** displays an informational message and a success message. It does not prompt for a password.

```
INFO: The task will be created under user name ("NT AUTHORITY\SYSTEM").  
SUCCESS: The Scheduled task "My App" has successfully been created.
```

To run a task with system permissions on a remote computer

The following command schedules the MyApp program to run on the Finance01 computer every morning at 4:00 A.M. with system permissions.

The command uses the **/tn** parameter to name the task and the **/tr** parameter to specify the remote copy of the MyApp program. It uses the **/sc** parameter to specify a daily schedule, but omits the **/mo** parameter because 1 (every day) is the default. It uses the **/st** parameter to specify the start time, which is also the time the task will run each day.

The command uses the **/s** parameter to provide the name of the remote computer and the **/u** parameter to specify an account with permission to schedule a task on the remote computer. It also uses the **/ru** parameter to specify that the task should run under the System account. Without the **/ru** parameter, the task would run with the permissions of the account specified by **/u**.

```
schtasks /create /tn "My App" /tr myapp.exe /sc daily /st 04:00 /s Finance01 /u Admin01 /ru System
```

Schtasks requests the password of the user named by the **/u** parameter and, after authenticating the password, displays a message indicating that the task is created and that it will run with permissions of the System account.

```
Type the password for Admin01:*****
```

```
INFO: The Schedule Task "My App" will be created under user name ("NT AUTHORITY\SYSTEM").  
SUCCESS: The scheduled task "My App" has successfully been created.
```

To schedule a task that runs more than one program

Each task runs only one program. However, you can create a batch file that runs multiple programs and then schedule a task to run the batch file. The following procedure demonstrates this method:

1. Create a batch file that starts the programs you want to run.

In this example, you create a batch file that starts Event Viewer (Eventvwr.exe) and System Monitor (Perfmon.exe).

- Open a text editor, such as Notepad.
- Type the name and fully qualified path to the executable file for each program. In this case, the file includes the following statements.

```
C:\Windows\System32\Eventvwr.exe C:\Windows\System32\Perfmon.exe
```

- Save the file as MyApps.bat.

2. Use **Schtasks.exe** to create a task that runs MyApps.bat.

The following command creates the Monitor task, which runs whenever anyone logs on. It uses the **/tn** parameter to name the task, and the **/tr** parameter to run MyApps.bat. It uses the **/sc** parameter to indicate the OnLogon schedule type and the **/ru** parameter to run the task with the permissions of the user's Administrator account.

```
schtasks /create /tn Monitor /tr C:\MyApps.bat /sc onlogon /ru Reskit\Administrator
```

As a result of this command, whenever a user logs on to the computer, the task starts both Event Viewer and System Monitor.

To schedule a task that runs on a remote computer

To schedule a task to run on a remote computer, you must add the task to the remote computer's schedule. Tasks of all types can be scheduled on a remote computer, but the following conditions must be met.

- You must have permission to schedule the task. As such, you must be logged on to the local computer with an account that is a member of the Administrators group on the remote computer, or you must use the **/u** parameter to provide the credentials of an Administrator of the remote computer.
- You can use the **/u** parameter only when the local and remote computers are in the same domain or the local computer is in a domain that the remote computer domain trusts. Otherwise, the remote computer cannot authenticate the user account specified and it cannot verify that the account is a member of the Administrators group.
- The task must have sufficient permission to run on the remote computer. The permissions required vary with the task. By default, the task runs with the permission of the current user of the local computer or, if the **/u** parameter is used, the task runs with the permission of the account specified by the **/u** parameter. However, you can use the **/ru** parameter to run the task with permissions of a different user account or with system permissions.

Examples

An Administrator schedules a task on a remote computer

The following command schedules the MyApp program to run on the SRV01 remote computer every ten days starting immediately. The command uses the **/s** parameter to provide the name of the remote computer. Because the local current user is an Administrator of the remote computer, the **/u** parameter, which provides alternate permissions for scheduling the task, is not necessary.

Please note that when scheduling tasks on a remote computer, all parameters refer to the remote computer. Therefore, the executable file specified by the **/tr** parameter refers to the copy of MyApp.exe on the remote computer.

```
schtasks /create /s SRV01 /tn "My App" /tr "c:\program files\corpapps\myapp.exe" /sc daily /mo 10
```

In response, **schtasks** displays a success message indicating that the task is scheduled.

A user schedules a command on a remote computer (Case 1)

The following command schedules the MyApp program to run on the SRV06 remote computer every three hours. Because Administrator permissions are required to schedule a task, the command uses the **/u** and **/p** parameters to provide the credentials of the user's Administrator account (Admin01 in the Reskits domain). By default, these permissions are also used to run the task. However, because the task does not need Administrator permissions to run, the command includes the **/u** and **/rp** parameters to override the default and run the task with permission of the user's non-Administrator account on the remote computer.

```
schtasks /create /s SRV06 /tn "My App" /tr "c:\program files\corpapps\myapp.exe" /sc hourly /mo 3 /u  
reskits\admin01 /p R43253@4$ /ru SRV06\user03 /rp MyFav!!Pswd
```

In response, **schtasks** displays a success message indicating that the task is scheduled.

A user schedules a command on a remote computer (Case 2)

The following command schedules the MyApp program to run on the SRV02 remote computer on the last day of every month. Because the local current user (user03) is not an Administrator of the remote computer, the command uses the **/u** parameter to provide the credentials of the user's Administrator account (Admin01 in the Reskits domain). The Administrator account permissions will be used to schedule the task and to run the task.

```
schtasks /create /s SRV02 /tn "My App" /tr "c:\program files\corpapps\myapp.exe" /sc monthly /mo LASTDAY /m *  
/u reskits\admin01
```

Because the command did not include the **/p** (password) parameter, **schtasks** prompts for the password. Then it displays a success message and, in this case, a warning.

```
Type the password for reskits\admin01:*****  
  
SUCCESS: The scheduled task "My App" has successfully been created.  
  
WARNING: The Scheduled task "My App" has been created, but may not run because  
the account information could not be set.
```

This warning indicates that the remote domain could not authenticate the account specified by the **/u** parameter. In this case, the remote domain could not authenticate the user account because the local computer is not a member of a domain that the remote computer domain trusts. When this occurs, the task job appears in the list of scheduled tasks, but the task is actually empty and it will not run.

The following display from a verbose query exposes the problem with the task. In the display, note that the value of **Next Run Time** is **Never** and that the value of **Run As User** is **Could not be retrieved from the task scheduler database**.

Had this computer been a member of the same domain or a trusted domain, the task would have been successfully scheduled and would have run as specified.

```

HostName: SRV44
TaskName: My App
Next Run Time: Never
Status:
Logon mode: Interactive/Background
Last Run Time: Never
Last Result: 0
Creator: user03
Schedule: At 3:52 PM on day 31 of every month, start
starting 12/14/2001
Task To Run: c:\program files\corpapps\myapp.exe
Start In: myapp.exe
Comment: N/A
Scheduled Task State: Disabled
Scheduled Type: Monthly
Start Time: 3:52:00 PM
Start Date: 12/14/2001
End Date: N/A
Days: 31
Months: JAN,FEB,MAR,APR,MAY,JUN,JUL,AUG,SEP,OCT,NO
V,DEC
Run As User: Could not be retrieved from the task sched
uler database
Delete Task If Not Rescheduled: Enabled
Stop Task If Runs X Hours and X Mins: 72:0
Repeat: Every: Disabled
Repeat: Until: Time: Disabled
Repeat: Until: Duration: Disabled
Repeat: Stop If Still Running: Disabled
Idle Time: Disabled
Power Management: Disabled

```

Remarks

- To run a **/create** command with the permissions of a different user, use the **/u** parameter. The **/u** parameter is valid only for scheduling tasks on remote computers.
- To view more **schtasks /create** examples, type **schtasks /create /?** at a command prompt.
- To schedule a task that runs with permissions of a different user, use the **/ru** parameter. The **/ru** parameter is valid for tasks on local and remote computers.
- To use the **/u** parameter, the local computer must be in the same domain as the remote computer or must be in a domain that the remote computer domain trusts. Otherwise, either the task is not created, or the task job is empty and the task does not run.
- **Schtasks** always prompts for a password unless you provide one, even when you schedule a task on the local computer using the current user account. This is normal behavior for **schtasks**.
- **Schtasks** does not verify program file locations or user account passwords. If you do not enter the correct file location or the correct password for the user account, the task is created, but it does not run. Also, if the password for an account changes or expires, and you do not change the password saved in the task, then the task does not run.
- The System account does not have interactive logon rights. Users do not see and cannot interact with programs run with system permissions.
- Each task runs only one program. However, you can create a batch file that starts multiple tasks, and then schedule a task that runs the batch file.
- You can test a task as soon as you create it. Use the **run** operation to test the task and then check the SchedLgU.txt file (*SystemRoot\SchedLgU.txt*) for errors.

schtasks change

Changes one or more of the following properties of a task.

- The program that the task runs (**/tr**).
- The user account under which the task runs (**/ru**).
- The password for the user account (**/rp**).
- Adds the interactive-only property to the task (**/it**).

Syntax

```
schtasks /change /tn <TaskName> [/s <Computer> [/u [<Domain>\]<User> [/p <Password>]]] [/ru { [<Domain>\]<User> | System}] [/rp <Password>] [/tr <TaskRun>] [/st <StartTime>] [/ri <Interval>] [{/et <EndTime> | /du <Duration>}] [/k] [/sd <StartDate>] [/ed <EndDate>] [{/ENABLE | DISABLE}] [/it] [/z]
```

Parameters

| TERM | DEFINITION |
|------------------|---|
| /tn <TaskName> | Identifies the task to be changed. Enter the task name. |
| /s <Computer> | Specifies the name or IP address of a remote computer (with or without backslashes). The default is the local computer. |
| /u [<Domain>] | Runs this command with the permissions of the specified user account. The default is the permissions of the current user of the local computer. The specified user account must be a member of the Administrators group on the remote computer. The /u and /p parameters are valid only for changing a task on a remote computer (/s). |
| /p <Password> | Specifies the password of the user account specified in the /u parameter. If you use the /u parameter, but omit the /p parameter or the password argument, schtasks prompts you for a password. The /u and /p parameters are valid only when you use /s . |
| /ru { [<Domain>] | System} |
| /rp <Password> | Specifies a new password for the existing user account, or the user account specified by the /ru parameter. This parameter is ignored with used with the local System account. |
| /tr <TaskRun> | Changes the program that the task runs. Enter the fully qualified path and file name of an executable file, script file, or batch file. If you omit the path, schtasks assumes that the file is in the <systemroot>\System32 directory. The specified program replaces the original program run by the task. |
| /st <Starttime> | Specifies the start time for the task, using the 24-hour time format, HH:mm. For example, a value of 14:30 is equivalent to the 12-hour time of 2:30 PM. |
| /ri <Interval> | Specifies the repetition interval for the scheduled task, in minutes. Valid range is 1 - 599940 (599940 minutes = 9999 hours). |
| /et <EndTime> | Specifies the end time for the task, using the 24-hour time format, HH:mm. For example, a value of 14:30 is equivalent to the 12-hour time of 2:30 PM. |

| TERM | DEFINITION |
|-----------------|--|
| /du <Duration> | Specifies to close the task at the <EndTime> or , if specified. |
| /k | Stops the program that the task runs at the time specified by /et or /du . Without /k , schtasks does not start the program again after it reaches the time specified by /et or /du , but it does not stop the program if it is still running. This parameter is optional and valid only with a MINUTE or HOURLY schedule. |
| /sd <StartDate> | Specifies the first date on which the task should be run. The date format is MM/DD/YYYY. |
| /ed <EndDate> | Specifies the last date on which the task should be run. The format is MM/DD/YYYY. |
| /ENABLE | Specifies to enable the scheduled task. |
| /DISABLE | Specifies to disable the scheduled task. |
| /it | Specifies to run the scheduled task only when the "run as" user (the user account under which the task runs) is logged on to the computer. This parameter has no effect on tasks that run with system permissions or tasks that already have the interactive-only property set. You cannot use a change command to remove the interactive-only property from a task. By default, the "run as" user is the current user of the local computer when the task is scheduled or the account specified by the /u parameter, if one is used. However, if the command includes the /ru parameter, then the "run as" user is the account specified by the /ru parameter. |
| /z | Specifies to delete the task upon the completion of its schedule. |
| /? | Displays help at the command prompt. |

Remarks

- The **/tn** and **/s** parameters identify the task. The **/tr**, **/ru**, and **/rp** parameters specify properties of the task that you can change.
- The **/ru**, and **/rp** parameters specify the permissions under which the task runs. The **/u** and **/p** parameters specify the permissions used to change the task.
- To change tasks on a remote computer, the user must be logged on to the local computer with an account that is a member of the Administrators group on the remote computer.
- To run a **/change** command with the permissions of a different user (**/u**, **/p**), the local computer must be in the same domain as the remote computer or must be in a domain that the remote computer domain trusts.
- The System account does not have interactive logon rights. Users do not see and cannot interact with programs run with system permissions.
- To identify tasks with the **/it** property, use a verbose query (**/query /v**). In a verbose query display of a task with **/it**, the **Logon Mode** field has a value of **Interactive only**.

Examples

To change the program that a task runs

The following command changes the program that the Virus Check task runs from VirusCheck.exe to

VirusCheck2.exe. This command uses the **/tn** parameter to identify the task and the **/tr** parameter to specify the new program for the task. (You cannot change the task name.)

```
schtasks /change /tn "Virus Check" /tr C:\VirusCheck2.exe
```

In response, **SchTasks.exe** displays the following success message:

```
SUCCESS: The parameters of the scheduled task "Virus Check" have been changed.
```

As a result of this command, the Virus Check task now runs VirusCheck2.exe.

To change the password for a remote task

The following command changes the password of the user account for the RemindMe task on the remote computer, Svr01. The command uses the **/tn** parameter to identify the task and the **/s** parameter to specify the remote computer. It uses the **/rp** parameter to specify the new password, p@ssWord3.

This procedure is required whenever the password for a user account expires or changes. If the password saved in a task is no longer valid, then the task does not run.

```
schtasks /change /tn RemindMe /s Svr01 /rp p@ssWord3
```

In response, **SchTasks.exe** displays the following success message:

```
SUCCESS: The parameters of the scheduled task "RemindMe" have been changed.
```

As a result of this command, the RemindMe task now runs under its original user account, but with a new password.

To change the program and user account for a task

The following command changes the program that a task runs and changes the user account under which the task runs. Essentially, it uses an old schedule for a new task. This command changes the ChkNews task, which starts Notepad.exe every morning at 9:00 A.M., to start Internet Explorer instead.

The command uses the **/tn** parameter to identify the task. It uses the **/tr** parameter to change the program that the task runs and the **/ru** parameter to change the user account under which the task runs.

The **/ru**, and **/rp** parameter, which provides the password for the user account, is omitted. You must provide a password for the account, but you can use the **/ru**, and **/rp** parameter and type the password in clear text, or wait for **SchTasks.exe** to prompt you for a password, and then enter the password in obscured text.

```
schtasks /change /tn ChkNews /tr "c:\program files\Internet Explorer\iexplore.exe" /ru DomainX\Admin01
```

In response, **SchTasks.exe** requests the password for the user account. It obscures the text you type, so the password is not visible.

```
Please enter the password for DomainX\Admin01:
```

Note that the **/tn** parameter identifies the task and that the **/tr** and **/ru** parameters change the properties of the task. You cannot use another parameter to identify the task and you cannot change the task name.

In response, **SchTasks.exe** displays the following success message:

```
SUCCESS: The parameters of the scheduled task "ChkNews" have been changed.
```

As a result of this command, the ChkNews task now runs Internet Explorer with the permissions of an Administrator account.

To change a program to the System account

The following command changes the SecurityScript task so that it runs with permissions of the System account. It uses the **/ru ""** parameter to indicate the System account.

```
schtasks /change /tn SecurityScript /ru ""
```

In response, **SchTasks.exe** displays the following success message:

```
INFO: The run as user name for the scheduled task "SecurityScript" will be changed to "NT AUTHORITY\SYSTEM".  
SUCCESS: The parameters of the scheduled task "SecurityScript" have been changed.
```

Because tasks run with System account permissions do not require a password, **SchTasks.exe** does not prompt for one.

To run a program only when I am logged on

The following command adds the interactive-only property to MyApp, an existing task. This property assures that the task runs only when the "run as" user, that is, the user account under which the task runs, is logged on to the computer.

The command uses the **/tn** parameter to identify the task and the **/it** parameter to add the interactive-only property to the task. Because the task already runs with the permissions of my user account, I do not need to change the **/ru** parameter for the task.

```
schtasks /change /tn MyApp /it
```

In response, **SchTasks.exe** displays the following success message.

```
SUCCESS: The parameters of the scheduled task "MyApp" have been changed.
```

schtasks run

Starts a scheduled task immediately. The **run** operation ignores the schedule, but uses the program file location, user account, and password saved in the task to run the task immediately.

Syntax

```
schtasks /run /tn <TaskName> [/s <Computer> [/u [<Domain>\]<User> [/p <Password>]]]
```

Parameters

| TERM | DEFINITION |
|----------------|---|
| /tn <TaskName> | Required. Identifies the task. |
| /s <Computer> | Specifies the name or IP address of a remote computer (with or without backslashes). The default is the local computer. |

| TERM | DEFINITION |
|---------------|--|
| /u [<Domain>] | Runs this command with the permissions of the specified user account. By default, the command runs with the permissions of the current user of the local computer. The specified user account must be a member of the Administrators group on the remote computer. The /u and /p parameters are valid only when you use /s . |
| /p <Password> | Specifies the password of the user account specified in the /u parameter. If you use the /u parameter, but omit the /p parameter or the password argument, schtasks prompts you for a password. The /u and /p parameters are valid only when you use /s . |
| /? | Displays help at the command prompt. |

Remarks

- Use this operation to test your tasks. If a task does not run, check the Task Scheduler Service transaction log, <Systemroot>\SchedLgU.txt, for errors.
- Running a task does not affect the task schedule and does not change the next run time scheduled for the task.
- To run a task remotely, the task must be scheduled on the remote computer. When you run it, the task runs only on the remote computer. To verify that a task is running on a remote computer, use Task Manager or the Task Scheduler transaction log, <Systemroot>\SchedLgU.txt.

Examples

To run a task on the local computer

The following command starts the "Security Script" task.

```
schtasks /run /tn "Security Script"
```

In response, **SchTasks.exe** starts the script associated with the task and displays the following message:

```
SUCCESS: Attempted to run the scheduled task "Security Script".
```

As the message implies, **schtasks** tries to start the program, but it cannot verify that the program actually started.

To run a task on a remote computer

The following command starts the Update task on a remote computer, Svr01:

```
schtasks /run /tn Update /s Svr01
```

In this case, **SchTasks.exe** displays the following error message:

```
ERROR: Unable to run the scheduled task "Update".
```

To find the cause of the error, look in the Scheduled Tasks transaction log, C:\Windows\SchedLgU.txt on Svr01. In this case, the following entry appears in the log:

```
"Update.job" (update.exe) 3/26/2001 1:15:46 PM ** ERROR **
The attempt to log on to the account associated with the task failed, therefore, the task did not run.
The specific error is:
0x8007052e: Logon failure: unknown user name or bad password.
Verify that the task's Run-as name and password are valid and try again.
```

Apparently, the user name or password in the task is not valid on the system. The following **schtasks /change** command updates the user name and password for the Update task on Svr01:

```
schtasks /change /tn Update /s Svr01 /ru Administrator /rp PassW@rd3
```

After the **change** command completes, the **run** command is repeated. This time, the Update.exe program starts and **SchTasks.exe** displays the following message:

```
SUCCESS: Attempted to run the scheduled task "Update".
```

As the message implies, **schtasks** tries to start the program, but it cannot verify that the program actually started.

schtasks end

Stops a program started by a task.

Syntax

```
schtasks /end /tn <TaskName> [/s <Computer> [/u [<Domain>\]<User> [/p <Password>]]]
```

Parameters

| TERM | DEFINITION |
|----------------|---|
| /tn <TaskName> | Required. Identifies the task that started the program. |
| /s <Computer> | Specifies the name or IP address of a remote computer. The default is the local computer. |
| /u [<Domain>] | Runs this command with the permissions of the specified user account. By default, the command runs with the permissions of the current user of the local computer. The specified user account must be a member of the Administrators group on the remote computer. The /u and /p parameters are valid only when you use /s . |
| /p <Password> | Specifies the password of the user account specified in the /u parameter. If you use the /u parameter, but omit the /p parameter or the password argument, schtasks prompts you for a password. The /u and /p parameters are valid only when you use /s . |
| /? | Displays help. |

Remarks

SchTasks.exe ends only the instances of a program started by a scheduled task. To stop other processes, use TaskKill. For more information, see [Taskkill](#).

Examples

To end a task on a local computer

The following command stops the instance of Notepad.exe that was started by the My Notepad task:

```
schtasks /end /tn "My Notepad"
```

In response, **SchTasks.exe** stops the instance of Notepad.exe that the task started, and it displays the following success message:

```
SUCCESS: The scheduled task "My Notepad" has been terminated successfully.
```

To end a task on a remote computer

The following command stops the instance of Internet Explorer that was started by the InternetOn task on the remote computer, Svr01:

```
schtasks /end /tn InternetOn /s Svr01
```

In response, **SchTasks.exe** stops the instance of Internet Explorer that the task started, and it displays the following success message:

```
SUCCESS: The scheduled task "InternetOn" has been terminated successfully.
```

schtasks delete

Deletes a scheduled task.

Syntax

```
schtasks /delete /tn {<TaskName> | *} [/f] [/s <Computer> [/u [<Domain>\]<User> [/p <Password>]]]
```

Parameters

| TERM | DEFINITION |
|-----------------|---|
| /tn {<TaskName> | * |
| /f | Suppresses the confirmation message. The task is deleted without warning. |
| /s <Computer> | Specifies the name or IP address of a remote computer (with or without backslashes). The default is the local computer. |
| /u [<Domain>] | Runs this command with the permissions of the specified user account. By default, the command runs with the permissions of the current user of the local computer. The specified user account must be a member of the Administrators group on the remote computer. The /u and /p parameters are valid only when you use /s . |

| TERM | DEFINITION |
|---------------|---|
| /p <Password> | Specifies the password of the user account specified in the /u parameter. If you use the /u parameter, but omit the /p parameter or the password argument, schtasks prompts you for a password. The /u and /p parameters are valid only when you use /s . |
| /? | Displays help at the command prompt. |

Remarks

- The **delete** operation deletes the task from the schedule. It does not delete the program that the task runs or interrupt a running program.
- The **delete *** command deletes all tasks scheduled for the computer, not just the tasks scheduled by the current user.

Examples

To delete a task from the schedule of a remote computer

The following command deletes the "Start Mail" task from the schedule of a remote computer. It uses the **/s** parameter to identify the remote computer.

```
schtasks /delete /tn "Start Mail" /s Svr16
```

In response, **SchTasks.exe** displays the following confirmation message. To delete the task, press Y. To cancel the command, type n:

```
WARNING: Are you sure you want to remove the task "Start Mail" (Y/N)?
SUCCESS: The scheduled task "Start Mail" was successfully deleted.
```

To delete all tasks scheduled for the local computer

The following command deletes all tasks from the schedule of the local computer, including tasks scheduled by other users. It uses the **/tn *** parameter to represent all tasks on the computer and the **/f** parameter to suppress the confirmation message.

```
schtasks /delete /tn * /f
```

In response, **SchTasks.exe** displays the following success messages indicating that the only task scheduled, SecureScript, is deleted.

```
SUCCESS: The scheduled task "SecureScript" was successfully deleted.
```

schtasks query

Displays tasks scheduled to run on the computer.

Syntax

```
schtasks [/query] [/fo {TABLE | LIST | CSV}] [/nh] [/v] [/s <Computer> [/u [<Domain>\<User> [/p <Password>]]]
```

Parameters

| TERM | DEFINITION |
|---------------|---|
| /[query] | The operation name is optional. Typing schtasks without any parameters performs a query. |
| /fo {TABLE | LIST |
| /nh | Omits column headings from the table display. This parameter is valid with the TABLE and CSV output formats. |
| /v | Adds advanced properties of the tasks to the display. Queries using /v should be formatted as LIST or CSV . |
| /s <Computer> | Specifies the name or IP address of a remote computer (with or without backslashes). The default is the local computer. |
| /u [<Domain>] | Runs this command with the permissions of the specified user account. By default, the command runs with the permissions of the current user of the local computer. The specified user account must be a member of the Administrators group on the remote computer. The /u and /p parameters are valid only when you use /s . |
| /p <Password> | Specifies the password of the user account specified in the /u parameter. If you use /u , but omit /p or the password argument, schtasks prompts you for a password. The /u and /p parameters are valid only when you use /s . |
| /? | Displays help at the command prompt. |

Remarks

SchTasks.exe ends only the instances of a program started by a scheduled task. To stop other processes, use TaskKill. For more information, see [Taskkill](#).

Examples

To display the scheduled tasks on the local computer

The following commands display all tasks scheduled for the local computer. These commands produce the same result and can be used interchangeably.

```
schtasks
schtasks /query
```

In response, **SchTasks.exe** displays the tasks in the default, simple table format, as shown in the following table:

```
TaskName Next Run Time Status
=====
Microsoft Outlook At logon time
SecureScript 14:42:00 PM , 2/4/2001
```

To display advanced properties of scheduled tasks

The following command requests a detailed display of the tasks on the local computer. It uses the **/v** parameter to request a detailed (verbose) display and the **/fo LIST** parameter to format the display as a list for easy reading. You can use this command to verify that a task you created has the intended recurrence pattern.

schtasks /query /fo LIST /v

In response, **SchTasks.exe** displays a detailed property list for all tasks. The following display shows the task list for a task scheduled to run at 4:00 A.M. on the last Friday of every month:

```
HostName: RESKIT01
TaskName: SecureScript
Next Run Time: 4:00:00 AM , 3/30/2001
Status: Not yet run
Logon mode: Interactive/Background
Last Run Time: Never
Last Result: 0
Creator: user01
Schedule: At 4:00 AM on the last Fri of every month, starting 3/24/2001
Task To Run: C:\WINDOWS\system32\notepad.exe
Start In: notepad.exe
Comment: N/A
Scheduled Task State: Enabled
Scheduled Type: Monthly
Modifier: Last FRIDAY
Start Time: 4:00:00 AM
Start Date: 3/24/2001
End Date: N/A
Days: FRIDAY
Months: JAN,FEB,MAR,APR,MAY,JUN,JUL,AUG,SEP,OCT,NOV,DEC
Run As User: RESKIT\user01
Delete Task If Not Rescheduled: Enabled
Stop Task If Runs X Hours and X Mins: 72:0
Repeat: Until Time: Disabled
Repeat: Duration: Disabled
Repeat: Stop If Still Running: Disabled
Idle: Start Time(For IDLE Scheduled Type): Disabled
Idle: Only Start If Idle for X Minutes: Disabled
Idle: If Not Idle Retry For X Minutes: Disabled
Idle: Stop Task If Idle State End: Disabled
Power Mgmt: No Start On Batteries: Disabled
Power Mgmt: Stop On Battery Mode: Disabled
```

To log tasks scheduled for a remote computer

The following command requests a list of tasks scheduled for a remote computer, and adds the tasks to a comma-separated log file on the local computer. You can use this command format to collect and track tasks that are scheduled for multiple computers.

The command uses the **/s** parameter to identify the remote computer, Reskit16, the **/fo** parameter to specify the format and the **/nh** parameter to suppress the column headings. The **>>** append symbol redirects the output to the task log, p0102.csv, on the local computer, Svr01. Because the command runs on the remote computer, the local computer path must be fully qualified.

```
schtasks /query /s Reskit16 /fo csv /nh >> \\svr01\data\tasklogs\p0102.csv
```

In response, **SchTasks.exe** adds the tasks scheduled for the Reskit16 computer to the p0102.csv file on the local computer, Svr01.

Additional references

[Command-Line Syntax Key](#)

Scwcmd

1/8/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server 2012 R2, Windows Server 2012

The Scwcmd.exe command-line tool included with the Security Configuration Wizard (SCW) can be used to perform the following tasks:

- Configure one or many servers with an SCW-generated policy.
- Analyze one or many servers with an SCW-generated policy.
- View analysis results in HTML format.
- Roll back SCW policies.
- Transform an SCW-generated policy into native files that are supported by Group Policy.
- Register a Security Configuration Database extension with SCW.

When you use **scwcmd** to configure, analyze, or roll back a policy on a remote server, SCW must be installed on the remote server.

Syntax

```
scwcmd <command> [<subcommand>]
```

Parameters

| SUBCOMMAND | DESCRIPTION |
|------------|---|
| /analyze | Determines whether a computer is in compliance with a policy. See Scwcmd: analyze for syntax and options. |
| /configure | Applies an SCW-generated security policy to a computer. See Scwcmd: configure for syntax and options. |
| /register | Extends or customizes the SCW Security Configuration Database by registering a Security Configuration Database file that contains role, task, service, or port definitions. See Scwcmd: register for syntax and options. |
| /rollback | Applies the most recent rollback policy available, and then deletes that rollback policy. See Scwcmd: rollback for syntax and options. |
| /transform | Transforms a security policy file generated by using SCW into a new Group Policy object (GPO) in Active Directory Domain Services. See Scwcmd: transform syntax and options. |
| /view | Renders an .xml file by using a specified .xsl transform. See Scwcmd: view for syntax and options. |

| SUBCOMMAND | DESCRIPTION |
|------------|--------------------------------------|
| /? | Displays help at the command prompt. |

Additional references

- [Command-Line Syntax Key](#)

Scwcmd: analyze

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server 2012 R2, Windows Server 2012

Determines whether a computer is in compliance with a policy. Results are returned in an .xml file. Also accepts a list of computer names as input. To view the results in your browser, use **scwcmd view** and specify **%windir%\security\msscw\TransformFiles\scwanalysis.xml** as the .xsl transform. For examples of how this command can be used, see [Examples](#).

Syntax

```
scwcmd analyze [[/m:<ComputerName> | /ou:<Ou>] /p:<Policy>] [/i:<ComputerList>] [/o:  
<ResultDir>] [/u:<UserName>] [/pw:<Password>] [/t:<Threads>] [/l] [/e]
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------------|--|
| /m:<ComputerName> | Specifies the NetBIOS name, DNS name, or IP address of the computer to analyze. If the /m parameter is specified, then the /p parameter must also be specified. |
| /ou:<OuName> | Specifies the fully qualified domain name (FQDN) of an organizational unit (OU) in Active Directory Domain Services. If the /ou parameter is specified, then the /p parameter must also be specified. All computers in the OU will be analyzed against the given policy. |
| /p:<Policy> | Specifies the path and file name of the .xml policy file to be used to perform the analysis. |
| /i:<ComputerList> | Specifies the path and file name of an .xml file that contains a list of computers along with their expected policy files. All computers in the .xml file will be analyzed against their corresponding policy files. A sample .xml file is %windir%\security\SampleMachineList.xml. |
| /o:<ResultDir> | Specifies the path and directory where the analysis result files should be saved. The default is the current directory. |
| /u:<UserName> | Specifies an alternate user credential to use when performing the analysis on a remote computer. The default is the logged on user. |
| /pw:<Password> | Specifies an alternate user credential to use when performing the analysis on a remote computer. The default is the password of the logged on user. |

| PARAMETER | DESCRIPTION |
|--------------|--|
| /t:<Threads> | Specifies the number of simultaneous outstanding analysis operations that should be maintained during the analysis (DefaultValue=40, MinValue= 1, MaxValue= 1000). |
| /l | Causes the analysis process to be logged. One log file will be generated for each computer being analyzed. The log files will be stored in the same directory as the result files. Use the /o option to specify the directory for the result files. |
| /e | Log an event to the Application Event log if a mismatch is found. |
| /? | Displays help at the command prompt. |

Remarks

Scwcmd.exe is only available on computers running Windows Server 2008 R2, Windows Server 2008, or Windows Server 2003.

Examples

To analyze a security policy against the file webpolicy.xml, type:

```
scwcmd analyze /p:webpolicy.xml
```

To analyze a security policy on the computer named webserver against the file webpolicy.xml by using the credentials of the webadmin account, type:

```
scwcmd analyze /m:webserver /p:webpolicy.xml /u:webadmin
```

To analyze a security policy against the file webpolicy.xml, with a maximum of 100 threads, and output the results to a file named results in the resultserver share, type:

```
scwcmd analyze /i:webpolicy.xml /t:100 /o:\\resultserver\results
```

To analyze a security policy for the WebServers OU against the file webpolicy.xml by using the DomainAdmin credentials, type:

```
scwcmd analyze /ou:OU=WebServers,DC=Marketing,DC=ABCCompany,DC=com /p:webpolicy.xml /u:DomainAdmin
```

Additional references

- [Command-Line Syntax Key](#)

Scwcmd: configure

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server 2012 R2, Windows Server 2012

Applies a Security Configuration Wizard (SCW)-generated security policy to a computer. This command-line tool also accepts a list of computer names as input.

Syntax

```
scwcmd configure [[/m:<ComputerName> | /ou:<OuName>] /p:<Policy>] | /i:<ComputerList>] [/u:<UserName>] [/pw:<Password>] [/t:<Threads>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------------|---|
| /m:<ComputerName> | Specifies the NetBIOS name, DNS name, or IP address of the computer to configure. If the /m parameter is specified, then the /p parameter must also be specified. |
| /ou:<OuName> | Specifies the fully qualified domain name (FQDN) of an organizational unit (OU) in Active Directory Domain Services. If the /ou parameter is specified, then the /p parameter must also be specified. All computers in the OU will be analyzed according to the given policy. |
| /p:<Policy> | Specifies the path and file name of the .xml policy file to be used to perform the configuration. |
| /i:<ComputerList> | Specifies the path and file name of an .xml file that contains a list of computers along with their expected policy files. All computers in the .xml file will be configured according to their corresponding policy files. A sample .xml file is %windir%\security\SampleMachineList.xml. |
| /u:<UserName> | Specifies an alternate user credential to use when configuring a remote computer. The default is the logged on user. |
| /pw:<Password> | Specifies an alternate user credential to use when configuring a remote computer. The default is the password of the logged on user. |
| /t:<Threads> | Specifies the number of simultaneous outstanding configuration operations that should be maintained during the configuration process (DefaultValue=40, MinValue=1, MaxValue=1000). |
| /? | Displays help at the command prompt. |

Remarks

Scwcmd.exe is only available on computers running Windows Server 2008 R2, Windows Server 2008, or Windows Server 2003.

Examples

To configure a security policy against the file webpolicy.xml, type:

```
scwcmd configure /p:webpolicy.xml
```

To configure a security policy for the computer at 172.16.0.0 against the file webpolicy.xml by using the webadmin account credentials, type:

```
scwcmd configure /m:172.16.0.0 /p:webpolicy.xml /u:webadmin
```

To configure a security policy on all computers on the list campusmachines.xml with a maximum of 100 threads, type:

```
scwcmd configure /i:campusmachines.xml /t:100
```

To configure a security policy on all computers in the WebServers OU against the file webpolicy.xml by using the credentials of the DomainAdmin account, type:

```
scwcmd configure /ou:OU=WebServers,DC=Marketing,DC=ABCCompany,DC=com /p:webpolicy.xml /u:DomainAdmin
```

Additional references

- [Command-Line Syntax Key](#)

Scwcmd: register

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server 2012 R2, Windows Server 2012

Extends or customizes the Security Configuration Wizard (SCW) Security Configuration Database by registering a Security Configuration Database file that contains role, task, service, or port definitions.

Syntax

```
scwcmd register /kbname:<MyApp> [/kbfile:<kb.xml>] [/kb:<path>] [/d]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------------|--|
| /kbname:<MyApp> | Specifies the name under which the Security Configuration Database extension will be registered. This parameter must be specified. |
| /kbfile:<Kb.xml> | Specifies the path and file name of the Security Configuration Database file that will be used to extend or customize the base Security Configuration Database. To validate that the Security Configuration Database file is compliant with the SCW schema, use the %windir%\security\KBRegistrationInfo.xsd schema definition file. This option must be provided unless the /d parameter is specified. |
| /kb:<Path> | Specifies the path to the directory that contains the SCW Security Configuration Database files to be updated. If this option is not specified, %windir%\security\msscwin\kbs is used. |
| /d | Unregisters a Security Configuration Database extension from the Security Configuration Database. The extension to unregister is specified by the /kbname parameter. (The /kbfile parameter should not be specified.) The Security Configuration Database to unregister the extension from is specified by the /kb parameter. |
| /? | Displays help at the command prompt. |

Remarks

Scwcmd.exe is only available on computers running Windows Server 2008 R2, Windows Server 2008, or Windows Server 2003.

Examples

To register the Security Configuration Database file named SCWKbForMyApp.xml under the name MyApp in the location \\kbserver\kb, type:

```
scwcmd register /kbfile:d:\SCWKBForMyApp.xml /kbname:MyApp /kb:\\kbserver\kb
```

To unregister the Security Configuration Database MyApp located at \\kbserver\kb, type:

```
scwcmd register /d /kbname:MyApp /kb:\\kbserver\kb
```

Additional references

- [Command-Line Syntax Key](#)

secedit

4/13/2018 • 2 minutes to read • [Edit Online](#)

Configures and analyzes system security by comparing your current configuration to specified security templates.

Syntax

```
secedit
[/analyze /db <database file name> /cfg <configuration file name> [/overwrite] /log <log file name> [/quiet]]
[/configure /db <database file name> [/cfg <configuration filename>] [/overwrite] [/areas [securitypolicy |
group_mgmt | user_rights | regkeys | filestore | services]] [/log <log file name>] [/quiet]]
[/export /db <database file name> [/mergedpolicy] /cfg <configuration file name> [/areas [securitypolicy |
group_mgmt | user_rights | regkeys | filestore | services]] [/log <log file name>]]
[/generaterollback /db <database file name> /cfg <configuration file name> /rbk <rollback file name> [/log
<log file name>] [/quiet]]
[/import /db <database file name> /cfg <configuration file name> [/overwrite] [/areas [securitypolicy |
group_mgmt | user_rights | regkeys | filestore | services]] [/log <log file name>] [/quiet]]
[/validate <configuration file name>]
```

Parameters

| PARAMETER | DESCRIPTION |
|--|--|
| Secedit:analyze | Allows you to analyze current systems settings against baseline settings that are stored in a database. The analysis results are stored in a separate area of the database and can be viewed in the Security Configuration and Analysis snap-in. |
| Secedit:configure | Allows you to configure a system with security settings stored in a database. |
| Secedit:export | Allows you to export security settings stored in a database. |
| Secedit:generaterollback | Allows you to generate a rollback template with respect to a configuration template. |
| Secedit:import | Allows you to import a security template into a database so that the settings specified in the template can be applied to a system or analyzed against a system. |
| Secedit:validate | Allows you to validate the syntax of a security template. |

Remarks

For all filenames, the current directory is used if no path is specified.

When a security template is created using the Security Template snap-in and the Security Configuration and Analysis snap-in is run, the following files are created:

| FILE | DESCRIPTION |
|------|-------------|
|------|-------------|

| FILE | DESCRIPTION |
|------------------------|--|
| Scesrv.log | <p>Location: %windir%\security\logs</p> <p>Created by: operating system</p> <p>File type: text</p> <p>Refresh rate: Overwritten when secedit /analyze, /configure, /export or /import are run.</p> <p>Content: Contains the results of the analysis grouped by policy type.</p> |
| User-selected name.sdb | <p>Location: %windir%*user account\Documents\Security\Database</p> <p>Created by: running the Security Configuration and Analysis snap-in</p> <p>File type: proprietary</p> <p>Refresh rate: Updated whenever a new security template is created.</p> <p>Content*: Local security policies and user-created security templates.</p> |
| User-selected name.log | <p>Location: User-defined but defaults to %windir%*user account\Documents\Security\Logs</p> <p>Created by: Running the /analyze and /configure subcommands (or using the Security Configuration and Analysis snap-in)</p> <p>File type: text</p> <p>Refresh rate: Running the /analyze and /configure subcommands (or using the Security Configuration and Analysis snap-in); overwritten.</p> <p>Content*:</p> <ol style="list-style-type: none"> 1. Log file name 2. Date and time 3. Results of analysis or investigation. |
| User-selected name.inf | <p>Location: %windir%*user account\Documents\Security\Templates</p> <p>Created by: running the Security Template snap-in</p> <p>File type: text</p> <p>Refresh rate: each time the security template is updated</p> <p>Content*: Contains the set up information for the template for each policy selected using the snap-in.</p> |

NOTE

The Microsoft Management Console (MMC) and the Security Configuration and Analysis snap-in are not available on Server Core.

Additional references

For examples of how this command can be used, see the examples section in any of the subcommand files.

- [Command-Line Syntax Key](#)

secedit:analyze

4/13/2018 • 2 minutes to read • [Edit Online](#)

Allows you to analyze current systems settings against baseline settings that are stored in a database. For examples of how this command can be used, see [Examples](#).

Syntax

```
Secedit /analyze /db <database file name> [/cfg <configuration file name>] [/overwrite] [/log <log file name>] [/quiet]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| db | <p>Required.</p> <p>Specifies the path and file name of a database that contains the stored configuration against which the analysis will be performed.</p> <p>If file name specifies a database that has not had a security template (as represented by the configuration file) associated with it, the <code>/cfg \<configuration file name></code> command-line option must also be specified.</p> |
| cfg | <p>Optional.</p> <p>Specifies the path and file name for the security template that will be imported into the database for analysis.</p> <p>This /cfg option is only valid when used with the <code>/db \<database file name></code> parameter. If this is not specified, the analysis is performed against any configuration already stored in the database.</p> |
| overwrite | <p>Optional.</p> <p>Specifies whether the security template in the /cfg parameter should overwrite any template or composite template that is stored in the database instead of appending the results to the stored template.</p> <p>This command-line option is only valid when the <code>/cfg \<configuration file name></code> parameter is also used.</p> <p>If this is not specified, the template in the /cfg parameter is appended to the stored template.</p> |
| log | <p>Optional.</p> <p>Specifies the path and file name of the log file to be used in the process.</p> |
| quiet | <p>Optional.</p> <p>Suppresses screen output. You can still view analysis results by using the Security Configuration and Analysis snap-in to the Microsoft Management Console (MMC).</p> |

Remarks

The analysis results are stored in a separate area of the database and can be viewed in the Security Configuration and Analysis snap-in to the MMC.

If the path for the log file is not provided, the default log file, (*systemroot*\Documents and Settings*UserAccount\My Documents\Security\Logs*DatabaseName.log) is used.

In Windows Server 2008, `Secedit /refreshpolicy` has been replaced with `gpupdate`. For information on how to refresh security settings, see [Gpupdate](#).

Examples

Perform the analysis for the security parameters on the security database, SecDbContoso.sdb, you created using the Security Configuration and Analysis snap-in. Direct the output to the file SecAnalysisContosoFY11 with prompting so you can verify the command ran correctly.

```
Secedit /analyze /db C:\Security\FY11\SecDbContoso.sdb /log C:\Security\FY11\SecAnalysisContosoFY11.log
```

Let's say that the analysis revealed some inadequacies so the security template, SecContoso.inf, was modified. Run the command again to incorporate the changes, directing the output to the existing file SecAnalysisContosoFY11 with no prompting.

```
Secedit /analyze /db C:\Security\FY11\SecDbContoso.sdb /cfg SecContoso.inf /overwrite /log  
C:\Security\FY11\SecAnalysisContosoFY11.xml /quiet
```

Additional references

- [Secedit](#)
- [Command-Line Syntax Key](#)

secedit:configure

4/13/2018 • 2 minutes to read • [Edit Online](#)

Allows you to configure the current system settings using security settings stored in a database. For examples of how this command can be used, see [Examples](#).

Syntax

```
Secedit /configure /db <database file name> [/cfg <configuration file name>] [/overwrite] [/areas  
SECURITYPOLICY | GROUP_MGMT | USER_RIGHTS | REGKEYS | FILESTORE | SERVICES] [/log <log file name>] [/quiet]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| db | <p>Required.</p> <p>Specifies the path and file name of a database that contains the stored configuration.</p> <p>If file name specifies a database that has not had a security template (as represented by the configuration file) associated with it, the <code>/cfg \<configuration file name></code> command-line option must also be specified.</p> |
| cfg | <p>Optional.</p> <p>Specifies the path and file name for the security template that will be imported into the database for analysis.</p> <p>This /cfg option is only valid when used with the <code>/db \<database file name></code> parameter. If this is not specified, the analysis is performed against any configuration already stored in the database.</p> |
| overwrite | <p>Optional.</p> <p>Specifies whether the security template in the /cfg parameter should overwrite any template or composite template that is stored in the database instead of appending the results to the stored template.</p> <p>This command-line option is only valid when the <code>/cfg \<configuration file name></code> parameter is also used.</p> <p>If this is not specified, the template in the /cfg parameter is appended to the stored template.</p> |

| PARAMETER | DESCRIPTION |
|-----------|---|
| areas | Optional. Specifies the security areas to be applied to the system. If this parameter is not specified, all security settings defined in the database are applied to the system. To configure multiple areas, separate each area by a space. The following security areas are supported: - SecurityPolicy Local policy and domain policy for the system, including account policies, audit policies, security options, and so on. - Group_Mgmt Restricted group settings for any groups specified in the security template. - User_Rights User logon rights and granting of privileges. - RegKeys Security on local registry keys. - FileStore Security on local file storage. - Services Security for all defined services. |
| log | Optional. Specifies the path and file name of the log file for the process. |
| quiet | Optional. Suppresses screen and log output. You can still view analysis results by using the Security Configuration and Analysis snap-in to the Microsoft Management Console (MMC). |

Remarks

If the path for the log file is not provided, the default log file, (*systemroot*\Users *UserAccount\My Documents\Security\Logs*DatabaseName.log) is used.

Beginning with Windows Server 2008, `Secedit /refreshpolicy` has been replaced with `gpupdate`. For information on how to refresh security settings, see [Gpupdate](#).

Examples

Perform the analysis for the security parameters on the security database, SecDbContoso.sdb, you created using the Security Configuration and Analysis snap-in. Direct the output to the file SecAnalysisContosoFY11 with prompting so you can verify the command ran correctly.

```
Secedit /analyze /db C:\Security\FY11\SecDbContoso.sdb /log C:\Security\FY11\SecAnalysisContosoFY11.log
```

Let's say that the analysis revealed some inadequacies so the security template, SecContoso.inf, was modified. Run the command again to incorporate the changes, directing the output to the existing file SecAnalysisContosoFY11 with no prompting.

```
Secedit /configure /db C:\Security\FY11\SecDbContoso.sdb /cfg SecContoso.inf /overwrite /log C:\Security\FY11\SecAnalysisContosoFY11.xml /quiet
```

Additional references

- [Secedit](#)

- [Secedit:analyze](#)
- [Command-Line Syntax Key](#)

secedit:export

4/13/2018 • 2 minutes to read • [Edit Online](#)

Exports security settings stored in a database configured with security templates. For examples of how this command can be used, see [Examples](#).

Syntax

```
Secedit /export /db <database file name> [/mergedpolicy] /cfg <configuration file name> [/areas  
[securitypolicy | group_mgmt | user_rights | regkeys | filestore | services]] [/log <log file name>] [/quiet]
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------|--|
| db | Required. Specifies the path and file name of a database that contains the stored configuration against which the analysis will be performed. If file name specifies a database that has not had a security template (as represented by the configuration file) associated with it, the <code>/cfg \<configuration file name></code> command-line option must also be specified. |
| mergedpolicy | Optional. Merges and exports domain and local policy security settings. |
| cfg | Required. Specifies the path and file name for the security template that will be imported into the database for analysis. This /cfg option is only valid when used with the <code>/db \<database file name></code> parameter. If this is not specified, the analysis is performed against any configuration already stored in the database. |

| PARAMETER | DESCRIPTION |
|-----------|---|
| areas | <p>Optional.</p> <p>Specifies the security areas to be applied to the system. If this parameter is not specified, all security settings defined in the database are applied to the system. To configure multiple areas, separate each area by a space. The following security areas are supported:</p> <ul style="list-style-type: none"> - SecurityPolicy Local policy and domain policy for the system, including account policies, audit policies, security options, and so on. - Group_Mgmt Restricted group settings for any groups specified in the security template. - User_Rights User logon rights and granting of privileges. - RegKeys Security on local registry keys. - FileStore Security on local file storage. - Services Security for all defined services. |
| log | <p>Optional.</p> <p>Specifies the path and file name of the log file for the process.</p> |
| quiet | <p>Optional.</p> <p>Suppresses screen and log output. You can still view analysis results by using the Security Configuration and Analysis snap-in to the Microsoft Management Console (MMC).</p> |

Remarks

You can use this command to backup your security policies on a local computer in addition to importing the settings to another computer.

If the path for the log file is not provided, the default log file, (*systemroot*\Documents and Settings*UserAccount\My Documents\Security\Logs*DatabaseName.log) is used.

In Windows Server 2008, `Secedit /refreshpolicy` has been replaced with `gpupdate`. For information on how to refresh security settings, see [Gpupdate](#).

Examples

Export the security database and the domain security policies to an inf file and then import that file to a different database in order to replicate the security policy settings on another computer.

```
Secedit /export /db C:\Security\FY11\SecDbContoso.sdb /mergedpolicy /cfg SecContoso.inf /log
C:\Security\FY11\SecAnalysisContosoFY11.log /quiet
```

Import that file to a different database on another computer.

```
Secedit /import /db C:\Security\FY12\SecDbContoso.sdb /cfg SecContoso.inf /log
C:\Security\FY11\SecAnalysisContosoFY12.log /quiet
```

Additional references

- [Secedit:import](#)

- [Secedit](#)
- [Command-Line Syntax Key](#)

secedit:generaterollback

4/13/2018 • 2 minutes to read • [Edit Online](#)

Allows you to generate a rollback template for a specified configuration template. For examples of how this command can be used, see [Examples](#).

Syntax

```
Secedit /generaterollback /db <database file name> /cfg <configuration file name> /rbk <rollback template file name> [log <log file name>] [/quiet]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| db | Required. Specifies the path and file name of a database that contains the stored configuration against which the analysis will be performed. If file name specifies a database that has not had a security template (as represented by the configuration file) associated with it, the <code>/cfg \<configuration file name></code> command-line option must also be specified. |
| cfg | Required. Specifies the path and file name for the security template that will be imported into the database for analysis. This /cfg option is only valid when used with the <code>/db \<database file name></code> parameter. If this is not specified, the analysis is performed against any configuration already stored in the database. |
| rbk | Required. Specifies a security template into which the rollback information is written. Security templates are created using the Security Templates snap-in. Rollback files can be created with this command. |
| log | Optional. Specifies the path and file name of the log file for the process. |
| quiet | Optional. Suppresses screen and log output. You can still view analysis results by using the Security Configuration and Analysis snap-in to the Microsoft Management Console (MMC). |

Remarks

If the path for the log file is not provided, the default log file, (*systemroot*\Users *UserAccount\My Documents\Security\Logs*DatabaseName.log) is used.

Beginning with Windows Server 2008, `Secedit /refreshpolicy` has been replaced with `gpupdate`. For information on how to refresh security settings, see [Gpupdate](#).

The successful running of this command will state "The task has completed successfully.❖" and logs only the mismatches between the stated security template and security policy configuration. It lists these mismatches in the scesrv.log.

If an existing rollback template is specified, this command will overwrite it. You can create a new rollback template with this command. No additional parameters are needed for either condition.

Examples

After creating the security template using the Security Configuration and Analysis snap-in, SecTmplContoso.inf, create the rollback configuration file to save the original settings. Write out the action to the FY11 log file.

```
Secedit /generaterollback /db C:\Security\FY11\SecDbContoso.sdb /cfg sectmplcontoso.inf /rbk  
sectmplcontosoRBK.inf /log C:\Security\FY11\SecAnalysisContosoFY11.log
```

Additional references

- [Secedit](#)
- [Command-Line Syntax Key](#)

secedit:import

4/13/2018 • 3 minutes to read • [Edit Online](#)

Imports security settings stored in an inf file previously exported from the database configured with security templates. For examples of how this command can be used, see [Examples](#).

Syntax

```
Secedit /import /db <database file name> /cfg <configuration file name> [/overwrite] [/areas [securitypolicy | group_mgmt | user_rights | regkeys | filestore | services]] [/log <log file name>] [/quiet]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| db | <p>Required.</p> <p>Specifies the path and file name of a database that contains the stored configuration into which the import will be performed.</p> <p>If file name specifies a database that has not had a security template (as represented by the configuration file) associated with it, the <code>/cfg \<configuration file name></code> command-line option must also be specified.</p> |
| overwrite | <p>Optional.</p> <p>Specifies whether the security template in the /cfg parameter should overwrite any template or composite template that is stored in the database instead of appending the results to the stored template.</p> <p>This command-line option is only valid when the <code>/cfg \<configuration file name></code> parameter is also used. If this is not specified, the template in the /cfg parameter is appended to the stored template.</p> |
| cfg | <p>Required.</p> <p>Specifies the path and file name for the security template that will be imported into the database for analysis.</p> <p>This /cfg option is only valid when used with the <code>/db \<database file name></code> parameter. If this is not specified, the analysis is performed against any configuration already stored in the database.</p> |
| overwrite | <p>Optional.</p> <p>Specifies whether the security template in the /cfg parameter should overwrite any template or composite template that is stored in the database instead of appending the results to the stored template.</p> <p>This command-line option is only valid when the <code>/cfg \<configuration file name></code> parameter is also used. If this is not specified, the template in the /cfg parameter is appended to the stored template.</p> |

| PARAMETER | DESCRIPTION |
|-----------|---|
| areas | Optional. Specifies the security areas to be applied to the system. If this parameter is not specified, all security settings defined in the database are applied to the system. To configure multiple areas, separate each area by a space. The following security areas are supported: - SecurityPolicy Local policy and domain policy for the system, including account policies, audit policies, security options, and so on. - Group_Mgmt Restricted group settings for any groups specified in the security template. - User_Rights User logon rights and granting of privileges. - RegKeys Security on local registry keys. - FileStore Security on local file storage. - Services Security for all defined services. |
| log | Optional. Specifies the path and file name of the log file for the process. |
| quiet | Optional. Suppresses screen and log output. You can still view analysis results by using the Security Configuration and Analysis snap-in to the Microsoft Management Console (MMC). |

Remarks

Before importing an .inf file onto another computer, run the command `secedit /generaterollback` on the database on which the import will be performed and `secedit /validate` on the import file to verify its integrity.

If the path for the log file is not provided, the default log file, (*systemroot*\Documents and Settings*UserAccount\My Documents\Security\Logs*DatabaseName.log) is used.

In Windows Server 2008, `Secedit /refreshpolicy` has been replaced with `gpupdate`. For information on how to refresh security settings, see [Gpupdate](#).

Examples

Export the security database and the domain security policies to an inf file and then import that file to a different database in order to replicate the security policy settings on another computer.

```
Secedit /export /db C:\Security\FY11\SecDbContoso.sdb /mergedpolicy /cfg NetworkShare\Policies\SecContoso.inf /log C:\Security\FY11\SecAnalysisContosoFY11.log /quiet
```

Import just the security policies portion of the file to a different database on another computer.

```
Secedit /import /db C:\Security\FY12\SecDbContoso.sdb /cfg NetworkShare\Policies\SecContoso.inf /areas securitypolicy /log C:\Security\FY11\SecAnalysisContosoFY12.log /quiet
```

Additional references

- [Secedit:export](#)

- [Secedit:generaterollback](#)
- [Secedit:validate](#)
- [Secedit](#)
- [Command-Line Syntax Key](#)

secedit:validate

4/13/2018 • 2 minutes to read • [Edit Online](#)

Validates the security settings stored in a security template (.inf file). For examples of how this command can be used, see [Examples](#).

Syntax

```
Secedit /validate <configuration file name>
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------------------|---|
| Configuration file name | Required. Specifies the path and file name for the security template that will be validated. |

Remarks

Validating security templates can help you if one is corrupted or inappropriately set.

An invalid security template will not be applied.

The log file will not be updated.

In Windows Server 2008, `Secedit /refreshpolicy` has been replaced with `gpupdate`. For information on how to refresh security settings, see [Gpupdate](#).

Examples

After a rollback is performed on a security template, you want to verify that the rollback inf file, secRBKcontoso.inf, is valid.

```
Secedit /validate secRBKcontoso.inf
```

Additional references

- [Secedit:generaterollback](#)
- [Secedit](#)
- [Command-Line Syntax Key](#)

serverceipoptin

4/13/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Allows you to participate in the Customer Experience Improvement Program (CEIP).

Syntax

```
serverceipoptin [/query] [/enable] [/disable]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------------|
| /query | verifies the current setting. |
| /enable | Enables participation. |
| /disable | Disables participation. |
| /? | Displays help at the command prompt. |

Examples

To verify the current settings, type:

```
serverceipoptin /query
```

To enable participation, type:

```
serverceipoptin /enable
```

To disable participation, type:

```
serverceipoptin /disable
```

additional references

- [Command-Line Syntax Key](#)

Servermanagercmd

7/11/2018 • 5 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

IMPORTANT

This command is available only on servers that are running Windows Server 2008 or Windows Server 2008 R2 .

Servermanagercmd.exe has been deprecated, and is not available in Windows Server 2012. For information about how to install or remove roles, role services, and features in Windows Server 2012 , see [Install or uninstall roles, role services, and features](#) on Microsoft TechNet.

Installs and removes roles, role services, and features. Also displays the list of all roles, role services, and features available, and shows which are installed on this computer. For additional information about the roles, role services, and features that you can specify by using this tool, see the [Server Manager help](#). For examples of how to use this command, see [Examples](#).

Syntax

```
servermanagercmd -query [[<Drive>:]<path>]<query.xml> [-logpath [[<Drive>:]<path>]<log.txt>]
servermanagercmd -inputpath [[<Drive>:]<path>]<answer.xml> [-resultpath <result.xml> [-restart] | -whatif] [-logpath [[<Drive>:]<path>]<log.txt>]
servermanagercmd -install <Id> [-allSubFeatures] [-resultpath [[<Drive>:]<path>]<result.xml> [-restart] | -whatif] [-logpath [[<Drive>:]<path>]<log.txt>]
servermanagercmd -remove <Id> [-resultpath <result.xml> [-restart] | -whatif] [-logpath [[<Drive>:]<path>]<log.txt>]
servermanagercmd [-help | -?]
servermanagercmd -version
```

Parameters

| PARAMETER | DESCRIPTION |
|--|--|
| -query [[<Drive>:]<path>]<query.xml> | Displays a list of all roles, role services, and features installed and available for installation on the server. You can also use the short form of this parameter, -q . If you want the query results saved to an XML file, specify an XML file to replace <i>query.xml</i> . |
| -inputpath <[[<Drive>:]<path>]<answer.xml> | Installs or removes the roles, role services, and features specified in an XML answer file represented by <i>answer.xml</i> . You can also use the short form of this parameter, -p . |

| PARAMETER | DESCRIPTION |
|---------------|---|
| -install <Id> | <p>Installs the role, role service, or feature specified by <i>Id</i>. The identifiers are case-insensitive. Multiple roles, role services, and features must be separated by spaces. The following optional parameters are used with the -install parameter.</p> <ul style="list-style-type: none"> - -setting <SettingName> = <SettingValue> Specifies required settings for the installation. - -allSubFeatures Specifies the installation of all subordinate services and features along with the parent role, role service, or feature named in the <i>Id</i> value. Note: Some role containers do not have a command line identifier to allow installation of all role services. This is the case when role services cannot be installed in the same instance of the Server Manager command. For example, the Federation Service role service of active directory Federation Services and the Federation Service Proxy role service cannot be installed by using the same Server Manager command instance. - -resultpath <result.xml> Saves installation results to an XML file represented by <i>*result.xml</i>. You can also use the short form of this parameter, -r. Note: You cannot run servermanagercmd with both the -resultpath parameter and the -whatif parameter specified. - -restart Restarts the computer automatically when installation is complete (if restarting is required by the roles or features installed). - -whatif Displays any operations specified for the -install parameter. You can also use the short form of the -whatif parameter, -w. You cannot run servermanagercmd with both the -resultpath parameter and the -whatif parameter specified. - -logpath <[[<Drive>:]<path>]>log.txt Specifies a name and location for the log file, other than the default, %windir%\temp\servermanager.log. |
| -remove <Id> | <p>Removes the role, role service, or feature specified by <i>Id</i>. The identifiers are case-insensitive. Multiple roles, role services, and features must be separated by spaces. The following optional parameters are used with the -remove parameter.</p> <ul style="list-style-type: none"> - -resultpath <[[<Drive>:]<path>]>result.xml Saves removal results to an XML file represented by <i>result.xml</i>. You can also use the short form of this parameter, -r. Note: You cannot run servermanagercmd with both the -resultpath parameter and the -whatif parameter specified. - -restart Restarts the computer automatically when removal is complete (if restarting is required by remaining roles or features). - -whatif Displays any operations specified for the -remove parameter. You can also use the short form of the -whatif parameter, -w. You cannot run servermanagercmd with both the -resultpath parameter and the -whatif parameter specified. - -logpath<[[<Drive>:]<path>]>log.txt Specifies a name and location for the log file, other than the default, %windir%\temp\servermanager.log. |
| -help | <p>Displays help in the Command prompt window. You can also use the short form, -?.</p> |

| PARAMETER | DESCRIPTION |
|-----------|--|
| -version | Displays the Server Manager version number. You can also use the short form, -v . |

Remarks

Servermanagercmd is deprecated, and is not guaranteed to be supported in future releases of Windows. We recommend that if you are running Server Manager on computers that are running Windows Server 2008 R2 , you use the Windows PowerShell cmdlets that are available for Server Manager. For more information, see [Server Manager cmdlets](#). Servermanagercmd can be run from any directory on the server's local drives. You must be a member of the Administrators group on the server on which you want to install or remove software.

IMPORTANT

Because of security restrictions imposed by User Account Control in Windows Server 2008 R2 , you must run **Servermanagercmd** in a Command prompt window opened with elevated permissions. To do this, right-click the Command prompt executable, or the **Command prompt** object on the **start** menu, and then click **Run as administrator**.

Examples

The following example shows how to use **servermanagercmd** to display a list of all roles, role services, and features available, and which roles, role services, and features are installed on the computer.

```
servermanagercmd -query
```

The following example shows how to use **servermanagercmd** to install the Web Server (IIS) role, and save the installation results to an XML file represented by *installResult.xml*.

```
servermanagercmd -install Web-Server -resultpath installResult.xml
```

The following example shows how to use the **** whatif**** parameter with **servermanagercmd** to display detailed information about the roles, role services, and features that would be installed or removed, based upon instructions that are specified in an XML answer file represented by *install.xml*.

```
servermanagercmd -inputpath install.xml -whatif
```

Additional references

- for a complete list of the role, role service, or feature identifiers you can specify for the *Id* parameter, or more information about using an XML answer file with **Servermanagercmd**, see the [Server Manager help](https://go.microsoft.com/fwlink/?LinkID=137387). (<https://go.microsoft.com/fwlink/?LinkID=137387>).
- See [Server Manager cmdlets](#) for a listing of Windows PowerShell cmdlets that are available for Server Manager.
- [Command-Line Syntax Key](#)

set

8/21/2018 • 4 minutes to read • [Edit Online](#)

Displays, sets, or removes CMD.EXE environment variables. If used without parameters, **set** displays the current environment variable settings.

For examples of how to use this command, see [Examples](#).

Syntax

```
set [<Variable>]=[<String>]]
set [/p] <Variable>=[<PromptString>]
set /a <Variable>=<Expression>
```

Parameters

| PARAMETER | DESCRIPTION |
|----------------|--|
| <Variable> | Specifies the environment variable to set or modify. |
| <String> | Specifies the string to associate with the specified environment variable. |
| /p | Sets the value of <i>Variable</i> to a line of input entered by the user. |
| <PromptString> | Optional. Specifies a message to prompt the user for input. This parameter is used with the /p command-line option. |
| /a | Sets <i>String</i> to a numerical expression that is evaluated. |
| <Expression> | Specifies a numerical expression. See Remarks for valid operators that can be used in <i>Expression</i> . |
| /? | Displays help at the command prompt. |

Remarks

- Using **set** with command extensions enabled

When command extensions are enabled (the default) and you run **set** with a value, it displays all of the variables that begin with that value.

- Using special characters

The characters **<**, **>**, **|**, **&**, **^** are special command shell characters, and they must be preceded by the escape character (**^**) or enclosed in quotation marks when used in *String* (for example, **"StringContaining&Symbol"**). If you use quotation marks to enclose a string that contains one of the special characters, the quotation marks are set as part of the environment variable value.

- Using environment variables

Use environment variables to control the behavior of some batch files and programs and to control the way Windows and the MS-DOS subsystem appears and works. The **set** command is often used in the Autoexec.nt file to set environment variables.

- Displaying the current environment settings

When you type the **set** command alone, the current environment settings are displayed. These settings usually include the COMSPEC and PATH environment variables, which are used to help find programs on disk. Two other environment variables used by Windows are PROMPT and DIRCMD.

- Using parameters

When you specify values for *Variable* and *String*, the specified *variable* value is added to the environment and *String* is associated with that variable. If the variable already exists in the environment, the new string value replaces the old string value.

If you specify only a variable and an equal sign (without *String*) for the **set** command, the *String* value associated with the variable is cleared (as if the variable is not there).

- Using **/a**

The following table lists the operators supported for **/a** in descending order of precedence.

| OPERATOR | OPERATION PERFORMED |
|------------------------|----------------------|
| () | Grouping |
| ! ~ - | Unary |
| * / % | Arithmetic |
| + - | Arithmetic |
| << >> | Logical shift |
| & | Bitwise AND |
| ^ | Bitwise exclusive OR |
| | |
| = *= /= %= += -= &= ^= | = <<= >>= |
| , | Expression separator |

If you use logical (**&&** or **||**) or modulus (%) operators, enclose the expression string in quotation marks. Any non-numeric strings in the expression are considered environment variable names, and their values are converted to numbers before they are processed. If you specify an environment variable name that is not defined in the current environment, a value of zero is allotted, which allows you to perform arithmetic with environment variable values without using the % to retrieve a value.

If you run **set /a** from the command line outside of a command script, it displays the final value of the expression.

Numeric values are decimal numbers unless prefixed by 0× for hexadecimal numbers or 0 for octal numbers. Therefore, 0×12 is the same as 18, which is the same as 022.

- Supporting delayed environment variable expansion

Delayed environment variable expansion support is disabled by default, but you can enable or disable it by using **cmd /v**.

- Working with command extensions

When command extensions are enabled (the default) and you run **set** alone, it displays all current environment variables. If you run **set** with a value, it displays the variables that match that value.

- Using **set** in batch files

When creating batch files, you can use **set** to create variables, and then use them in the same way that you would use the numbered variables **%0** through **%9**. You can also use the variables **%0** through **%9** as input for **set**.

- Calling a **set** variable from a batch file

When you call a variable value from a batch file, enclose the value with percent signs (%). For example, if your batch program creates an environment variable named BAUD, you can use the string associated with BAUD as a replaceable parameter by typing **%baud%** at the command prompt.

- Using **set** at the Recovery Console

The **set** command, with different parameters, is available from the Recovery Console.

Examples

To set an environment variable named TEST^1, type:

```
set testVar=test^1
```

NOTE

The **set** command assigns everything that follows the equal sign (=) to the value of the variable. If you type:

```
set testVar="test^1"
```

You get the following result:

```
testVar="test^1"
```

To set an environment variable named TEST&1, type:

```
set testVar=test^&1
```

To set an environment variable named INCLUDE so that the string C:\Inc (the \Inc directory on drive C) is associated with it, type:

```
set include=c:\inc
```

You can then use the string C:\Inc in batch files by enclosing the name INCLUDE with percent signs (%). For example, you might include the following command in a batch file so that you can display the contents of the directory that is associated with the INCLUDE environment variable:

```
dir %include%
```

When this command is processed, the string C:\Inc replaces **%include%**.

You can also use **set** in a batch program that adds a new directory to the PATH environment variable. For example:

```
@echo off
rem ADDPATH.BAT adds a new directory
rem to the path environment variable.
set path=%1;%path%
set
```

To display a list of all of the environment variables that begin with the letter P, type:

```
set p
```

NOTE

This command requires command extensions, which are enabled by default.

Additional references

[Command-Line Syntax Key](#)

setlocal

4/13/2018 • 2 minutes to read • [Edit Online](#)

Starts localization of environment variables in a batch file. Localization continues until a matching **endlocal** command is encountered or the end of the batch file is reached.

For examples of how to use this command, see [Examples](#).

Syntax

```
setlocal [enableextensions | disableextensions] [enabledelayedexpansion | disabledelayedexpansion]
```

Arguments

| ARGUMENT | DESCRIPTION |
|-------------------------|--|
| enableextensions | Enables the command extensions until the matching endlocal command is encountered, regardless of the setting before the setlocal command was run. |
| disableextensions | Disables the command extensions until the matching endlocal command is encountered, regardless of the setting before the setlocal command was run. |
| enabledelayedexpansion | Enables the delayed environment variable expansion until the matching endlocal command is encountered, regardless of the setting before the setlocal command was run. |
| disabledelayedexpansion | Disables the delayed environment variable expansion until the matching endlocal command is encountered, regardless of the setting before the setlocal command was run. |
| /? | Displays help at the command prompt. |

Remarks

- Using **setlocal**

When you use **setlocal** outside of a script or batch file, it has no effect.

- Changing environmental variables

Use **setlocal** to change environment variables when you run a batch file. Environment changes made after you run **setlocal** are local to the batch file. The Cmd.exe program restores previous settings when it encounters an **endlocal** command or reaches the end of the batch file.

- Nesting commands

You can have more than one **setlocal** or **endlocal** command in a batch program (that is, nested commands).

- Testing for command extensions in batch files

The **setlocal** command sets the ERRORLEVEL variable. If you pass {**enableextensions** | **disableextensions**} or {**enabledelayedexpansion** | **disabledelayedexpansion**}, the ERRORLEVEL variable is set to **0** (zero). Otherwise, it is set to **1**. You can use this information in batch scripts to determine whether the extensions are available, as shown in the following example:

```
setlocal enableextensions
verify other 2>nul
if errorlevel 1 echo Unable to enable extensions
```

Because **cmd** does not set the ERRORLEVEL variable when command extensions are disabled, the **verify** command initializes the ERRORLEVEL variable to a nonzero value when you use it with an invalid argument. Also, if you use the **setlocal** command with arguments {**enableextensions** | **disableextensions**} or {**enabledelayedexpansion** | **disabledelayedexpansion**} and it does not set the ERRORLEVEL variable to **1**, command extensions are not available.

Examples

You can localize environment variables in a batch file, as shown in the following sample script:

```
rem *****Begin Comment*****
rem This program starts the superapp batch program on the network,
rem directs the output to a file, and displays the file
rem in Notepad.
rem *****End Comment*****
@echo off
setlocal
path=g:\programs\superapp;%path%
call superapp>c:\superapp.out
endlocal
start notepad c:\superapp.out
```

Additional references

[Command-Line Syntax Key](#)

setx

4/13/2018 • 4 minutes to read • [Edit Online](#)

Creates or modifies environment variables in the user or system environment, without requiring programming or scripting. The **Setx** command also retrieves the values of registry keys and writes them to text files.

For examples of how to use this command, see [Examples](#).

Syntax

```
setx [/s <Computer> [/u [<Domain>]\<User name> [/p [<Password>]]]] <Variable> <Value> [/m]
setx [/s <Computer> [/u [<Domain>]\<User name> [/p [<Password>]]]] [<Variable>] /k <Path> [/m]
setx [/s <Computer> [/u [<Domain>]\<User name> [/p [<Password>]]]] /f <FileName> {[<Variable>] [/a <X>,<Y> | /r <X>,<Y> "<String>"]
[/m] | /x} [/d <Delimiters>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------|---|
| /s <Computer> | Specifies the name or IP address of a remote computer. Do not use backslashes. The default value is the name of the local computer. |
| /u [<Domain>] | Runs the script with the credentials of the specified user account. The default value is the system permissions. |
| /p [<Password>] | Specifies the password of the user account that is specified in the /u parameter. |
| <Variable> | Specifies the name of the environment variable that you want to set. |
| <Value> | Specifies the value to which you want to set the environment variable. |
| /k <Path> | Specifies that the variable is set based on information from a registry key. The <i>path</i> uses the following syntax: <code>\\<HIVE>\<KEY>\...<Value></code> For example, you might specify the following path: <code>HKEY_LOCAL_MACHINE\System\CurrentControlSet\Control\TimeZoneInformation\S</code> |
| /f <File name> | Specifies the file that you want to use. |
| /a <X>,<Y> | Specifies absolute coordinates and offset as search parameters. |
| /r <X>,<Y> "" | Specifies relative coordinates and offset from String as search parameters. |
| /m | Specifies to set the variable in the system environment. The default setting is the local environment. |
| /x | Displays file coordinates, ignoring the /a , /r , and /d command-line options. |
| /d <Delimiters> | Specifies delimiters such as "," or "\" to be used in addition to the four built-in delimiters — SPACE, TAB, ENTER, and LINEFEED. Valid delimiters include any ASCII character. The maximum number of delimiters is 15, including built-in delimiters. |
| /? | Displays help at the command prompt. |

Remarks

- The **Setx** command is similar to the UNIX utility SETENV.
- **Setx** provides the only command-line or programmatic way to directly and permanently set system environment values. System

environment variables are manually configurable through **Control Panel** or through a registry editor. The **set** command, which is internal to the command interpreter (Cmd.exe), sets user environment variables for the current console window only.

- You can use the **setx** command to set values for user and system environment variables from one of three sources (modes): Command Line Mode, Registry Mode, or File Mode.
- **Setx** writes variables to the master environment in the registry. Variables set with **setx** variables are available in future command windows only, not in the current command window.
- **HKEY_CURRENT_USER** and **HKEY_LOCAL_MACHINE** are the only supported hives. REG_DWORD, REG_EXPAND_SZ, REG_SZ, and REG_MULTI_SZ are the valid **RegKey** data types.
- When you gain access to **REG_MULTI_SZ** values in the registry, only the first item is extracted and used.
- You cannot use the **setx** command to remove values that have been added to the local or system environments. You can use **set** with a variable name and no value to remove a corresponding value from the local environment.
- REG_DWORD registry values are extracted and used in hexadecimal mode.
- File mode supports the parsing of carriage return and line feed (CRLF) text files only.

Examples

To set the MACHINE environment variable in the local environment to the value Brand1, type:

```
setx MACHINE Brand1
```

To set the MACHINE environment variable in the system environment to the value Brand1 Computer, type:

```
setx MACHINE "Brand1 Computer" /m
```

To set the MYPATH environment variable in the local environment to use the search path defined in the PATH environment variable, type:

```
setx MYPATH %PATH%
```

To set the MYPATH environment variable in the local environment to use the search path defined in the PATH environment variable after replacing ~ with %, type:

```
setx MYPATH ~PATH~
```

To set the MACHINE environment variable in the local environment to Brand1 on a remote computer named Computer1, type:

```
setx /s computer1 /u maindom\hiropln /p p@ssw23 MACHINE Brand1
```

To set the MYPATH environment variable in the local environment to use the search path defined in the PATH environment variable on a remote computer named Computer1, type:

```
setx /s computer1 /u maindom\hiropln /p p@ssw23 MYPATH %PATH%
```

To set the TZONE environment variable in the local environment to the value found in the **HKEY_LOCAL_MACHINE\System\CurrentControlSet\Control\TimeZoneInformation\StandardName** registry key, type:

```
setx TZONE /k HKEY_LOCAL_MACHINE\System\CurrentControlSet\Control\TimeZoneInformation\StandardName
```

To set the TZONE environment variable in the local environment of a remote computer named Computer1 to the value found in the **HKEY_LOCAL_MACHINE\System\CurrentControlSet\Control\TimeZoneInformation\StandardName** registry key, type:

```
setx /s computer1 /u maindom\hiropln /p p@ssw23 TZONE /k  
HKEY_LOCAL_MACHINE\System\CurrentControlSet\Control\TimeZoneInformation\StandardName
```

To set the BUILD environment variable in the system environment to the value found in the **HKEY_LOCAL_MACHINE\Software\Microsoft\WindowsNT\CurrentVersion\CurrentBuildNumber** registry key, type:

```
setx BUILD /k "HKEY_LOCAL_MACHINE\Software\Microsoft\WindowsNT\CurrentVersion\CurrentBuildNumber" /m
```

To set the BUILD environment variable in the system environment of a remote computer named Computer1 to the value found in the **HKEY_LOCAL_MACHINE\Software\Microsoft\WindowsNT\CurrentVersion\CurrentBuildNumber** registry key, type:

```
setx /s computer1 /u maindom\hiropln /p p@ssw23 BUILD /k "HKEY_LOCAL_MACHINE\Software\Microsoft\WindowsNT\CurrentVersion\CurrentBuildNumber" /m
```

To display the contents of a file named Ipconfig.out, along with the contents' corresponding coordinates, type:

```
setx /f ipconfig.out /x
```

To set the IPADDR environment variable in the local environment to the value found at the coordinate 5,11 in the file Ipconfig.out, type:

```
setx IPADDR /f ipconfig.out /a 5,11
```

To set the OCTET1 environment variable in the local environment to the value found at the coordinate 5,3 in the file Ipconfig.out with delimiters "#\$.", type:

```
setx OCTET1 /f ipconfig.out /a 5,3 /d "#$*."
```

To set the IPGATEWAY environment variable in the local environment to the value found at the coordinate 0,7 with respect to the coordinate of "Gateway" in the file Ipconfig.out, type:

```
setx IPGATEWAY /f ipconfig.out /r 0,7 Gateway
```

To display the contents of a file named Ipconfig.out — along with the contents' corresponding coordinates — on a computer named Computer1, type:

```
setx /s computer1 /u maindom\hiropln /p p@ssw23 /f ipconfig.out /x
```

Additional references

[Command-Line Syntax Key](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Scans and verifies the integrity of all protected system files and replaces incorrect versions with correct versions. for examples of how to use this command, see [Examples](#).

Syntax

```
sfc [/scannow] [/verifyonly] [/scanfile=<file>] [/verifyfile=<file>] [/offwindir=<offline windows directory> /offbootdir=<offline boot directory>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------|---|
| /scannow | Scans the integrity of all protected system files and repairs files with problems when possible. |
| /verifyonly | Scans integrity of all protected system files. No repair operation is performed. |
| /scanfile | Scans integrity of the specified file and repairs the file if problems are detected, when possible. |
| <file> | Specified full path and filename |
| /verifyfile | verifies the integrity of the specified file. No repair operation is performed. |
| /offwindir | Specifies the location of the offline windows directory, for offline repair. |
| /offbootdir | Specifies the location of the offline boot directory for offline |
| /? | Displays help at the command prompt. |

Remarks

- You must be logged on as a member of the Administrators group to run **sfc.exe**.
- if **sfc** discovers that a protected file has been overwritten, it retrieves the correct version of the file from the **systemroot\system32\dllcache** folder, and then replaces the incorrect file.
- There are functional differences between **sfc** on Windows Server 2003, Windows Server 2008 , and Windows Server 2008 R2 :
 - for more information about **sfc** on Windows Server 2003, see [article 310747](#) in the Microsoft Knowledge Base.
 - for more information about **sfc** on Windows Server 2008 , and Windows Server 2008 R2 , see [System File Checker](#).

Examples

To verify the **kernel32.dll** file, type:

```
sfc /verifyfile=c:\windows\system32\kernel32.dll
```

To setup offline repair of the **kernel32.dll** file with an offline boot directory set to **d:** and offline windows directory set to **d:\windows**, type:

```
sfc /scanfile=d:\windows\system32\kernel32.dll /offbootdir=d:\ /offwindir=d:\windows
```

Additional references

- [Command-Line Syntax Key](#)

shadow

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Enables you to remotely control an active session of another user on a Remote Desktop Session Host (rd Session Host) server. for examples of how to use this command, see [Examples](#).

Syntax

```
shadow {<SessionName> | <SessionID>} [/server:<ServerName>] [/v]
```

Parameters

| PARAMETER | DESCRIPTION |
|----------------------|---|
| <SessionName> | Specifies the name of the session that you want to remotely control. |
| <SessionID> | Specifies the ID of the session that you want to remotely control. Use query user to display the list of sessions and their session IDs. |
| /server:<ServerName> | Specifies the rd Session Host server containing the session that you want to remotely control. By default, the current rd Session Host4 server is used. |
| /v | Displays information about the actions being performed. |
| /? | Displays help at the command prompt. |

Remarks

- You can either view or actively control the session. If you choose to actively control a user's session, you will be able to input keyboard and mouse actions to the session.
- You can always remotely control your own sessions (except the current session), but you must have Full Control permission or remote Control special access permission to remotely control another session.
- You can also initiate remote control by using Remote Desktop Services Manager.
- Before monitoring begins, the server warns the user that the session is about to be remotely controlled, unless this warning is disabled. Your session might appear to be frozen for a few seconds while it waits for a response from the user. To configure remote control for users and sessions, use the Remote Desktop Services Configuration tool or the Remote Desktop Services extensions to Local Users and Groups and active directory Users and computers.
- Your session must be capable of supporting the video resolution used at the session that you are remotely controlling or the operation fails.
- The console session can neither remotely control another session nor can it be remotely controlled by another session.

- When you want to end remote control (shadowing), press CTRL+* (by using * from the numeric keypad only).

Examples

- To shadow session 93, type: `shadow 93`
- To shadow the session ACCTG01, type: `shadow ACCTG01`

additional references

[Command-Line Syntax Key Remote Desktop Services \(Terminal Services\) Command Reference](#)

shift

4/13/2018 • 2 minutes to read • [Edit Online](#)

Changes the position of batch parameters in a batch file.

For examples of how to use this command, see [Examples](#).

Syntax

```
shift [/n <N>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| /n <N> | Specifies to start shifting at the <i>N</i> th argument, where <i>N</i> is any value from 0 to 8. Requires command extensions, which are enabled by default. |
| /? | Displays help at the command prompt. |

Remarks

- The **shift** command changes the values of the batch parameters **%0** through **%9** by copying each parameter into the previous one—the value of **%1** is copied to **%0**, the value of **%2** is copied to **%1**, and so on. This is useful for writing a batch file that performs the same operation on any number of parameters.
- If command extensions are enabled, the **shift** command supports the **/n** command-line option. The **/n** option specifies to start shifting at the *N*th argument, where **N** is any value from 0 to 8. For example, **SHIFT /2** would shift **%3** to **%2**, **%4** to **%3**, and so on, and leave **%0** and **%1** unaffected. Command extensions are enabled by default.
- You can use the **shift** command to create a batch file that can accept more than 10 batch parameters. If you specify more than 10 parameters on the command line, those that appear after the tenth (**%9**) will be shifted one at a time into **%9**.
- The **shift** command has no effect on the **%*** batch parameter.
- There is no backward **shift** command. After you implement the **shift** command, you cannot recover the batch parameter (**%0**) that existed before the shift.

Examples

The following lines from a sample batch file called `Mycopy.bat` demonstrate how to use **shift** with any number of batch parameters. In this example, `Mycopy.bat` copies a list of files to a specific directory. The batch parameters are represented by the directory and file name arguments.

```
@echo off
rem MYCOPY.BAT copies any number of files
rem to a directory.
rem The command uses the following syntax:
rem mycopy dir file1 file2 ...
set todir=%1
:getfile
shift
if "%1"==" " goto end
copy %1 %todir%
goto getfile
:end
set todir=
echo All done
```

Additional references

[Command-Line Syntax Key](#)

showmount

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

You can use **showmount** to display mounted directories.

Syntax

```
showmount {-e|-a|-d} <Server>
```

Description

The **showmount** command-line utility displays information about mounted file systems exported by Server for NFS on the computer specified by *Server*. If *Server* is not provided, **showmount** displays information about the computer on which the **showmount** command is run.

You must provide one of the following options:

- **-e** - Displays all file systems exported on the server.
- **-a** - Displays all Network File System (NFS) clients and the directories on the server each has mounted.
- **-d** - Displays all directories on the server that are currently mounted by NFS clients.

See Also

[Services for Network File System Command Reference](#)

shutdown

4/13/2018 • 3 minutes to read • [Edit Online](#)

Enables you to shut down or restart local or remote computers one at a time.

For examples of how to use this command, see [Examples](#).

Syntax

```
shutdown [/i | /l | /s | /r | /a | /p | /h | /e] [/f] [/m \\<ComputerName>] [/t <XXX>] [/d [p|u:]<XX>:<YY>] [/c "comment"]]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------------|--|
| /i | Displays the Remote Shutdown Dialog box. The /i option must be the first parameter following the command. If /i is specified, all other options are ignored. |
| /l | Logs off the current user immediately, with no time-out period. You cannot use /l with /m or /t . |
| /s | Shuts down the computer. |
| /r | Restarts the computer after shutdown. |
| /a | Aborts a system shutdown. Effective only during the timeout period. To use /a , you must also use the /m option. |
| /p | Turns off the local computer only (not a remote computer)—with no time-out period or warning. You can use /p only with /d or /f . If your computer does not support power-off functionality, it will shut down when you use /p , but the power to the computer will remain on. |
| /h | Puts the local computer into hibernation, if hibernation is enabled. You can use /h only with /f . |
| /e | Enables you to document the reason for the unexpected shutdown on the target computer. |
| /f | Forces running applications to close without warning users. Caution: Using the /f option might result in loss of unsaved data. |
| /m \\<ComputerName> | Specifies the target computer. Cannot be used with the /l option. |

| PARAMETER | DESCRIPTION |
|---|--|
| <code>/t <XXX></code> | Sets the time-out period or delay to <i>XXX</i> seconds before a restart or shutdown. This causes a warning to display on the local console. You can specify 0-600 seconds. If you do not use /t , the time-out period is 30 seconds by default. |
| <code>/d [p u:]<XX>:<YY></code> | Lists the reason for the system restart or shutdown. The following are the parameter values: p Indicates that the restart or shutdown is planned. u Indicates that the reason is user defined. Note: If p or u are not specified, the restart or shutdown is unplanned. <i>XX</i> Specifies the major reason number (positive integer less than 256). <i>YY</i> Specifies the minor reason number (positive integer less than 65536). |
| <code>/c "<Comment>"</code> | Enables you to comment in detail about the reason for the shutdown. You must first provide a reason by using the /d option. You must enclose comments in quotation marks. You can use a maximum of 511 characters. |
| <code>/?</code> | Displays help at the command prompt, including a list of the major and minor reasons that are defined on your local computer. |

Remarks

- Users must be assigned the **Shut down the system** user right to shut down a local or remotely administered computer that is using the **shutdown** command.
- Users must be members of the Administrators group to annotate an unexpected shutdown of a local or remotely administered computer. If the target computer is joined to a domain, members of the Domain Admins group might be able to perform this procedure. For more information, see:
 - [Default local groups](#)
 - [Default groups](#)
- If you want to shut down more than one computer at a time, you can call **shutdown** for each computer by using a script, or you can use **shutdown /i** to display the Remote Shutdown Dialog box.
- If you specify major and minor reason codes, you must first define these reason codes on each computer where you plan to use the reasons. If the reason codes are not defined on the target computer, Shutdown Event Tracker cannot log the correct reason text.
- Remember to indicate that a shutdown is planned by using the **p:** parameter. Omitting **p:** indicates that a shutdown is unplanned. If you type **p:** followed by the reason code for an unplanned shutdown, the command will not carry out the shutdown. Conversely, if you omit **p:** and type in the reason code for a planned shutdown, the command will not carry out the shutdown.

Examples

To force applications to close and restart the local computer after a one-minute delay with the reason "Application: Maintenance (Planned)" and the comment "Reconfiguring myapp.exe" type:

```
shutdown /r /t 60 /c "Reconfiguring myapp.exe" /f /d p:4:1
```

To restart the remote computer \\ServerName with the same parameters, type:

```
shutdown /r /m \\servername /t 60 /c "Reconfiguring myapp.exe" /f /d p:4:1
```

Additional references

[Command-Line Syntax Key](#)

sort

7/11/2018 • 4 minutes to read • [Edit Online](#)

Reads input, sorts data, and writes the results to the screen, to a file, or to another device.

For examples of how to use this command, see [Examples](#).

Syntax

```
sort [/r] [/+<N>] [/m <Kilobytes>] [/l <Locale>] [/rec <Characters>] [[<Drive1>:][<Path1>]<FileName1>] [/t  
[<Drive2>:][<Path2>]] [/o [<Drive3>:][<Path3>]<FileName3>]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------------------------------|--|
| /r | Reverses the sort order (that is, sorts from Z to A and from 9 to 0). |
| /+ <N> | Specifies the character position number where sort will begin each comparison. <i>N</i> can be any valid integer. |
| /m <Kilobytes> | Specifies the amount of main memory to use for the sort in kilobytes (KB). |
| /l <Locale> | Overrides the sort order of characters that are defined by the system default locale (that is, the language and Country/Region selected during installation). |
| /rec <Characters> | Specifies the maximum number of characters in a record or a line of the input file (the default value is 4,096 and the maximum is 65,535). |
| [<Drive1>:][<Path1>]<FileName1> | Specifies the file to be sorted. If no file name is specified, the standard input is sorted. Specifying the input file is faster than redirecting the same file as standard input. |
| /t [<Drive2>:][<Path2>] | Specifies the path of the directory to hold the sort command's working storage if the data does not fit in the main memory. By default, the system temporary directory is used. |
| /o [<Drive3>:][<Path3>]<FileName3> | Specifies the file where the sorted input is to be stored. If not specified, the data is written to the standard output. Specifying the output file is faster than redirecting standard output to the same file. |
| /? | Displays help at the command prompt. |

Remarks

- Using the **/+** command-line option

By default, comparisons start at the first character of each line. The **/+** command-line option starts comparisons at the character that is specified by *N*. For example, **/+3** indicates that each comparison should begin at the third character of each line. Lines with fewer than *N* characters collate before other lines.

- Using the **/m** command-line option

The memory used is always a minimum of 160 KB. If the memory size is specified, the exact specified amount is used for the sort (must be at least 160 KB), regardless of how much main memory is available.

The default maximum memory size when no size is specified is 90 percent of the available main memory if both the input and output are files, or 45 percent of main memory otherwise. The default setting usually gives the best performance.

- Using the **/l** command-line option

Currently, the only alternative to the default locale is the "C" locale, which is faster than natural language sorting (it sorts characters according to their binary encodings).

- Using redirection symbols with the **sort** command

You can use the pipe symbol (**|**) to direct input data to the **sort** command from another command or to direct sorted output to another command. You can specify input and output files by using redirection symbols (**<** or **>**). It can be faster and more efficient (especially with large files) to specify the input file directly (as defined by *FileName1* in the command syntax), and then specify the output file using the **/o** parameter.

- Case sensitivity

The **sort** command does not distinguish between uppercase and lowercase letters.

- Limits on file size

The **sort** command has no limit on file size.

- Collating sequence

The sort program uses the collating-sequence table that corresponds to the Country/Region code and code-page settings. Characters greater than ASCII code 127 are sorted based on information in the Country.sys file or in an alternate file specified by the **country** command in your Config.nt file.

- Memory usage

If the sort fits within the maximum memory size (as set by default or as specified by the **/m** parameter), the sort is performed in a single pass. Otherwise, the sort is performed in two separate sort and merge passes, and the amounts of memory used for both passes are equal. When two passes are performed, the partially sorted data is stored in a temporary file on disk. If there is not enough memory to perform the sort in two passes, a run-time error is issued. If the **/m** command-line option is used to specify more memory than is truly available, performance degradation or a run-time error can occur.

Examples

Sorting a file

To sort and display in reverse order the lines in a file named Expenses.txt, type:

```
sort /r expenses.txt
```

Sorting the output from a command

To search a large file named Maillist.txt for the text "Jones," and to sort the results of the search, use the pipe (**|**) to

direct the output of a **find** command to the **sort** command, as follows:

```
find "Jones" maillist.txt | sort
```

The command produces a sorted list of lines that contain the specified text.

Sorting keyboard input

To sort keyboard input and display the results alphabetically on the screen, you can first use the **sort** command with no parameters, as follows:

```
sort
```

Then type the text that you want sorted, and press ENTER at the end of each line. When you have finished typing text, press CTRL+Z, and then press ENTER. The **sort** command displays the text you typed, sorted alphabetically.

Additional references

[Command-Line Syntax Key](#)

start

9/28/2018 • 2 minutes to read • [Edit Online](#)

Starts a separate Command Prompt window to run a specified program or command.

For examples of how to use this command, see [Examples](#).

Syntax

```
start ["<Title>"] [/d <Path>] [/i] [{/min | /max}] [{/separate | /shared}] [{/low | /normal | /high | /realtime | /abovenormal | belownormal}] [/affinity <HexAffinity>] [/wait] [/b {<Command> | <Program>} [<Parameters>]]
```

Parameters

| PARAMETER | DESCRIPTION |
|--|---|
| "<Title>" | Specifies the title to display in the Command Prompt window title bar. |
| /d <Path> | Specifies the startup directory. |
| /i | Passes the Cmd.exe startup environment to the new Command Prompt window. If /i is not specified, the current environment is used. |
| /min /max | Specifies to minimize (/min) or maximize (/max) the new Command Prompt window. |
| /separate /shared | Starts 16-bit programs in a separate memory space (/separate) or shared memory space (/shared). These options are not supported on 64-bit platforms. |
| /low /normal /high /realtime /abovenormal /belownormal | Starts an application in the specified priority class. Valid priority class values are /low , /normal , /high , /realtime , /abovenormal , and /belownormal . |
| /affinity <HexAffinity> | Applies the specified processor affinity mask (expressed as a hexadecimal number) to the new application. |
| /wait | Starts an application and waits for it to end. |
| /b | Starts an application without opening a new Command Prompt window. CTRL+C handling is ignored unless the application enables CTRL+C processing. Use CTRL+BREAK to interrupt the application. |
| /b <Command> <Program> | Specifies the command or program to start. |
| <Parameters> | Specifies parameters to pass to the command or program. |

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------------|
| /? | Displays help at the command prompt. |

Remarks

- You can run nonexecutable files through their file association by typing the name of the file as a command.
- When you run a command that contains the string "CMD" as the first token without an extension or path qualifier, "CMD" is replaced with the value of the COMSPEC variable. This prevents users from picking up **cmd** from the current directory.
- When you run a 32-bit graphical user interface (GUI) application, **cmd** does not wait for the application to quit before returning to the command prompt. This behavior does not occur if you run the application from a command script.
- When you run a command that uses a first token that does not contain an extension, Cmd.exe uses the value of the PATHEXT environment variable to determine which extensions to look for and in what order. The default value for the PATHEXT variable is:

```
.COM;.EXE;.BAT;.CMD
```

Note that the syntax is the same as the PATH variable, with semicolons separating each extension.

- When it searches for an executable file, if there is no match on any extension, **start** checks to see if the name matches a directory name. If it does, **start** opens Explorer.exe on that path.

Examples

To start the MyApp program at the command prompt and retain use of the current Command Prompt window, type:

```
start myapp
```

To view the **start** command-line help topic in a separate maximized Command Prompt window, type:

```
start /max start /?
```

Additional references

[Command-Line Syntax Key](#)

subst

7/11/2018 • 2 minutes to read • [Edit Online](#)

Associates a path with a drive letter. If used without parameters, **subst** displays the names of the virtual drives in effect.

For examples of how to use this command, see [Examples](#).

Syntax

```
subst [<Drive1>: [<Drive2>:]<Path>]
subst <Drive1>: /d
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------------|---|
| <Drive1>: | Specifies the virtual drive to which you want to assign a path. |
| [<Drive2>:]<Path> | Specifies the physical drive and path that you want to assign to a virtual drive. |
| /d | Deletes a substituted (virtual) drive. |
| /? | Displays help at the command prompt. |

Remarks

- The following commands do not work and should not be used on drives that are specified in the **subst** command:

chkdsk

diskcomp

diskcopy

format

label

recover

- The *Drive1* parameter must be within the range that is specified by the **lastdrive** command. If not, **subst** displays the following error message:

```
Invalid parameter - drive1:
```

Examples

To create a virtual drive Z for the path B:\User\Betty\Forms, type:

```
subst z: b:\user\betty\forms
```

Instead of typing the full path, you can reach this directory by typing the letter of the virtual drive followed by a colon as follows:

```
z:
```

Additional references

[Command-Line Syntax Key](#)

sxstrace

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Diagnoses side-by-side problems.

Syntax

```
sxstrace [{[trace /logfile:<FileName> [/nostop]]|[parse /logfile:<FileName> /outfile:<ParsedFile> [/filter:<AppName>]]}]
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------|---|
| trace | Enables tracing for sxs (side-by-side) |
| /logfile | Specifies the raw log file. |
| <FileName> | Saves tracing log to <i>FileName</i> . |
| /nostop | Specifies no prompt to stop tracing. |
| parse | Translates the raw trace file. |
| /outfile | Specifies the output filename. |
| <ParsedFile> | Specifies the filename of the parsed file. |
| /filter | Allows the output to be filtered. |
| <AppName> | Specifies the name of the application. |
| stoptrace | Stop the trace if it is not stopped before. |
| /? | Displays help at the command prompt. |

Examples

Enable tracing and save trace file to **sxstrace.etl**:

```
sxstrace trace /logfile:sxstrace.etl
```

Translate the raw trace file into a human readable format and save the result to **sxstrace.txt**:

```
sxstrace parse /logfile:sxstrace.etl /outfile:sxstrace.txt
```

Additional references

- [Command-Line Syntax Key](#)

sysocmgr

4/13/2018 • 2 minutes to read • [Edit Online](#)

Sysocmgr is deprecated, and is not guaranteed to be supported in future releases of Windows.

This tool is included in Windows Server 2003. For more information see [Sysocmgr](#).

systeminfo

4/13/2018 • 2 minutes to read • [Edit Online](#)

Displays detailed configuration information about a computer and its operating system, including operating system configuration, security information, product ID, and hardware properties (such as RAM, disk space, and network cards).

For examples of how to use this command, see [Examples](#).

Syntax

```
Systeminfo [/s <Computer> [/u <Domain>\<UserName> [/p <Password>]] [/fo {TABLE | LIST | CSV}] [/nh]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------------------|---|
| /s <Computer> | Specifies the name or IP address of a remote computer (do not use backslashes). The default is the local computer. |
| /u <Domain> <UserName> | Runs the command with the account permissions of the specified user account. If /u is not specified, this command uses the permissions of the user who is currently logged on to the computer that is issuing the command. |
| /p <Password> | Specifies the password of the user account that is specified in the /u parameter. |
| /fo <Format> | Specifies the output format with one of the following values: TABLE: Displays output in a table. LIST: Displays output in a list. CSV: Displays output in Comma Separated Values format. |
| /nh | Suppresses column headers in the output. Valid when the /fo parameter is set to TABLE or CSV. |
| /? | Displays help at the command prompt. |

Examples

To view configuration information for a computer named Srvmain, type:

```
systeminfo /s srvmain
```

To remotely view configuration information for a computer named Srvmain2 that is located on the Maindom domain, type:

```
systeminfo /s srvmain2 /u maindom\hiropln
```

To remotely view configuration information (in list format) for a computer named Srvmain2 that is located on the Maindom domain, type:

systeminfo /s srvmain2 /u maindom\hiropln /p p@ssW23 /fo list

Additional references

[Command-Line Syntax Key](#)

takeown

7/11/2018 • 2 minutes to read • [Edit Online](#)

Enables an administrator to recover access to a file that previously was denied, by making the administrator the owner of the file.

For examples of how to use this command, see [Examples](#).

Syntax

```
takeown [/s <Computer> [/u [<Domain>\]<User name> [/p [<Password>]]] /f <File name> [/a] [/r [/d {Y|N}]]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------|---|
| /s <Computer> | Specifies the name or IP address of a remote computer (do not use backslashes). The default value is the local computer. This parameter applies to all of the files and folders specified in the command. |
| /u [<Domain>] | Runs the script with the permissions of the specified user account. The default value is system permissions. |
| /p [<Password>] | Specifies the password of the user account that is specified in the /u parameter. |
| /f <File name> | Specifies the file name or directory name pattern. You can use the wildcard character * when specifying the pattern. You can also use the syntax <i>ShareName*FileName*</i> . |
| /a | Gives ownership to the Administrators group instead of the current user. |
| /r | Performs a recursive operation on all files in the specified directory and subdirectories. |
| /d {Y N} | Suppresses the confirmation prompt that is displayed when the current user does not have the "List Folder" permission on a specified directory, and instead uses the specified default value. Valid values for the /d option are as follows: <ul style="list-style-type: none">- Y: Take ownership of the directory.- N: Skip the directory. Note that you must use this option in conjunction with the /r option. |
| /? | Displays help at the command prompt. |

Remarks

- This command is typically used in batch files.
- If the **/a** parameter is not specified, file ownership is given to the user who is currently logged on to the

computer.

- Mixed patterns using (?) and *) are not supported by **takeown** command.
- After deleting the lock with **takeown**, you might have to use Windows Explorer or the **cacls** command to give yourself full permissions to the files and directories before you can delete them. For more information about **cacls**, see "Additional references" at the end of this topic.

Examples

To take ownership of a file named Lostfile, type:

```
takeown /f lostfile
```

Additional references

[Command-Line Syntax Key](#)

tapicfg

7/11/2018 • 4 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Creates, removes, or displays a TAPI application directory partition, or sets a default TAPI application directory partition. TAPI 3.1 clients can use the information in this application directory partition with the directory service locator service to find and communicate with TAPI directories. You can also use **tapicfg** to create or remove service connection points, which enable TAPI clients to efficiently locate TAPI application directory partitions in a domain. For more information, see remarks. To view the command syntax, click a command.

- [tapicfg install](#)
- [tapicfg remove](#)
- [tapicfg publishscp](#)
- [tapicfg removescp](#)
- [tapicfg show](#)
- [tapicfg makedefault](#)

tapicfg install

Creates a TAPI application directory partition.

Syntax

```
tapicfg install /directory:<PartitionName> [/server:<DCName>] [/forcedefault]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------------------------------|---|
| install /directory:<PartitionName> | Required. Specifies the DNS name of the TAPI application directory partition to be created. This name must be a fully qualified domain name. |
| /server: <DCName> | Specifies the DNS name of the domain controller on which the TAPI application directory partition is created. If the domain controller name is not specified, the name of the local computer is used. |
| /forcedefault | Specifies that this directory is the default TAPI application directory partition for the domain. There can be multiple TAPI application directory partitions in a domain. if this directory is the first TAPI application directory partition created on the domain, it is automatically set as the default, regardless of whether you use the /forcedefault option. |
| /? | Displays help at the command prompt. |

tapicfg remove

Removes a TAPI application directory partition.

Syntax

```
tapicfg remove /directory:<PartitionName>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------------------------|--|
| remove /directory:<PartitionName> | Required. Specifies the DNS name of the TAPI application directory partition to be removed. Note that this name must be a fully qualified domain name. |
| /? | Displays help at the command prompt. |

tapicfg publishscp

Creates a service connection point to publish a TAPI application directory partition.

Syntax

```
tapicfg publishscp /directory:<PartitionName> [/domain:<DomainName>] [/forcedefault]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------------------------------|--|
| publishscp /directory:<PartitionName> | Required. Specifies the DNS name of the TAPI application directory partition that the service connection point will publish. |
| /domain:<DomainName> | Specifies the DNS name of the domain in which the service connection point is created. If the domain name is not specified, the name of the local domain is used. |
| /forcedefault | Specifies that this directory is the default TAPI application directory partition for the domain. There can be multiple TAPI application directory partitions in a domain. |
| /? | Displays help at the command prompt. |

tapicfg removescp

Removes a service connection point for a TAPI application directory partition.

Syntax

```
tapicfg removescp /directory:<PartitionName> [/domain:<DomainName>]
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------------------------------|---|
| removescp /directory:<PartitionName> | Required. Specifies the DNS name of the TAPI application directory partition for which a service connection point is removed. |
| /domain: <DomainName> | Specifies the DNS name of the domain from which the service connection point is removed. If the domain name is not specified, the name of the local domain is used. |
| /? | Displays help at the command prompt. |

tapicfg show

Displays the names and locations of the TAPI application directory partitions in the domain.

Syntax

```
tapicfg show [/defaultonly][ /domain:<DomainName>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------------|--|
| /defaultonly | Displays the names and locations of only the default TAPI application directory partition in the domain. |
| /domain: <DomainName> | Specifies the DNS name of the domain for which the TAPI application directory partitions are displayed. If the domain name is not specified, the name of the local domain is used. |
| /? | Displays help at the command prompt. |

tapicfg makedefault

Sets the default TAPI application directory partition for the domain.

Syntax

```
tapicfg makedefault /directory:<PartitionName> [/domain:<DomainName>]
```

Parameters

| PARAMETER | DESCRIPTION |
|--|--|
| makedefault /directory:<PartitionName> | Required. Specifies the DNS name of the TAPI application directory partition set as the default partition for the domain. Note that this name must be a fully qualified domain name. Specifies the DNS name of the domain for which the TAPI application directory partition is set as the default. If the domain name is not specified, the name of the local domain is used. |
| /? | Displays help at the command prompt. |

Remarks

You must be a member of the Enterprise Admins group in active directory to run either **tapicfg install** (to create a TAPI application directory partition) or **tapicfg remove** (to remove a TAPI application directory partition).

This command-line tool can be run on any computer that is a member of the domain.

User-supplied text (such as the names of TAPI application directory partitions, servers, and domains) with International or Unicode characters are only displayed correctly if appropriate fonts and language support are installed.

You can still use Internet Locator Service (ILS) servers in your organization, if ILS is needed to support certain applications, because TAPI clients running Windows XP or a Windows Server 2003 operating system can query either ILS servers or TAPI application directory partitions.

You can use **tapicfg** to create or remove service connection points. If the TAPI application directory partition is renamed for any reason (for example, if you rename the domain in which it resides), you must remove the existing service connection point and create a new one that contains the new DNS name of the TAPI application directory partition to be published. Otherwise, TAPI clients are unable to locate and access the TAPI application directory partition. You can also remove a service connection point for maintenance or security purposes (for example, if you do not want to expose TAPI data on a specific TAPI application directory partition).

Examples

To create a TAPI application directory partition named `tapifiction.testdom.microsoft.com` on a server named `testdc.testdom.microsoft.com` and then set it as the default TAPI application directory partition for the new domain, type:

```
tapicfg install /directory:tapifiction.testdom.microsoft.com /server:testdc.testdom.microsoft.com /forcedefault
```

To display the name of the default TAPI application directory partition for the new domain, type:

```
tapicfg show /defaultonly
```

Additional references

- [Command-Line Syntax Key](#)

taskkill

7/11/2018 • 3 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Ends one or more tasks or processes. Processes can be ended by process ID or image name. **taskkill** replaces the **kill** tool. for examples of how to use this command, see [Examples](#).

Syntax

```
taskkill [/s <computer> [/u [<Domain>\<UserName> [/p [<Password>]]]] {[/fi <Filter>] [...] [/pid <ProcessID> | /im <ImageName>]} [/f] [/t]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------------------|---|
| /s <computer> | Specifies the name or IP address of a remote computer (do not use backslashes). The default is the local computer. |
| /u <Domain>\<UserName> | Runs the command with the account permissions of the user who is specified by <i>UserName</i> or <i>Domain\UserName</i> . /u can be specified only if /s is specified. The default is the permissions of the user who is currently logged on to the computer that is issuing the command. |
| /p <Password> | Specifies the password of the user account that is specified in the /u parameter. |
| /fi <Filter> | Applies a filter to select a set of tasks. You can use more than one filter or use the wildcard character (*) to specify all tasks or image names. See the following table for valid filter names, operators, and values . |
| /pid <ProcessID> | Specifies the process ID of the process to be terminated. |
| /im <ImageName> | Specifies the image name of the process to be terminated. Use the wildcard character (*) to specify all image names. |
| /f | Specifies that processes be forcefully terminated. This parameter is ignored for remote processes; all remote processes are forcefully terminated. |
| /t | Terminates the specified process and any child processes started by it. |

Filter names, operators, and values

| FILTER NAME | VALID OPERATORS | VALID VALUE(S) |
|-------------|------------------------|--|
| StatUS | eq, ne | RUNNING NOT RESPONDING UNKNOWN |
| IMAGENAME | eq, ne | Image name |
| PID | eq, ne, gt, lt, ge, le | PID value |
| SESSION | eq, ne, gt, lt, ge, le | Session number |
| CPUtime | eq, ne, gt, lt, ge, le | CPU time in the format <i>HH:MM:SS</i> , where <i>MM</i> and <i>SS</i> are between 0 and 59 and <i>HH</i> is any unsigned number |
| MEMUSAGE | eq, ne, gt, lt, ge, le | Memory usage in KB |
| USERNAME | eq, ne | Any valid user name (<i>User</i> or <i>Domain\User</i>) |
| SERVICES | eq, ne | Service name |
| WINDOWTITLE | eq, ne | Window title |
| MODULES | eq, ne | DLL name |

Remarks

- The WINDOWTITLE and StatUS filters are not supported when a remote system is specified.
- The wildcard character (\) is accepted for the ***/im** option only when a filter is applied.
- Termination of remote processes is always carried out forcefully, regardless of whether the */f* option is specified.
- Supplying a computer name to the hostname filter causes a shutdown and all processes are stopped.
- You can use **tasklist** to determine the process ID (PID) for the process to be terminated.

Examples

To end the processes with process IDs 1230, 1241, and 1253, type:

```
taskkill /pid 1230 /pid 1241 /pid 1253
```

To forcefully end the process "Notepad.exe" if it was started by the system, type:

```
taskkill /f /fi "USERNAME eq NT AUTHORITY\SYSTEM" /im notepad.exe
```

To end all processes on the remote computer "Srvmain" with an image name beginning with "note," while using the credentials for the user account Hiropln, type:

```
taskkill /s srvmain /u maindom\hiropln /p p@ssw23 /fi "IMAGENAME eq note*" /im *
```

To end the process with the process ID 2134 and any child processes that it started, but only if those processes

were started by the Administrator account, type:

```
taskkill /pid 2134 /t /fi "username eq administrator"
```

To end all processes that have a process ID greater than or equal to 1000, regardless of their image names, type:

```
taskkill /f /fi "PID ge 1000" /im *
```

additional references

[Command-Line Syntax Key](#)

tasklist

7/11/2018 • 2 minutes to read • [Edit Online](#)

Displays a list of currently running processes on the local computer or on a remote computer. **Tasklist** replaces the **tlist** tool.

For examples of how to use this command, see [Examples](#).

Syntax

```
tasklist [/s <Computer> [/u [<Domain>\]<UserName> [/p <Password>]]] [{/m <Module> | /svc | /v}] [/fo {table | list | csv}] [/nh] [/fi <Filter> [/fi <Filter> [ ... ]]]
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------------------|--|
| /s <Computer> | Specifies the name or IP address of a remote computer (do not use backslashes). The default is the local computer. |
| /u [<Domain>\]<UserName> | Runs the command with the account permissions of the user who is specified by <i>UserName</i> or <i>Domain*UserName</i> . **/u* can be specified only if /s is specified. The default is the permissions of the user who is currently logged on to the computer that is issuing the command. |
| /p <Password> | Specifies the password of the user account that is specified in the /u parameter. |
| /m <Module> | Lists all tasks with DLL modules loaded that match the given pattern name. If the module name is not specified, this option displays all modules loaded by each task. |
| /svc | Lists all the service information for each process without truncation. Valid when the /fo parameter is set to table . |
| /v | Displays verbose task information in the output. For complete verbose output without truncation, use /v and /svc together. |
| /fo {table list csv} | Specifies the format to use for the output. Valid values are table , list , and csv . The default format for output is table . |
| /nh | Suppresses column headers in the output. Valid when the /fo parameter is set to table or csv . |
| /fi <Filter> | Specifies the types of processes to include in or exclude from the query. See the following table for valid filter names, operators, and values. |
| /? | Displays help at the command prompt. |

Filter names, operators, and values

| FILTER NAME | VALID OPERATORS | VALID VALUES |
|-------------|------------------------|--|
| STATUS | eq, ne | RUNNING |
| IMAGENAME | eq, ne | Image name |
| PID | eq, ne, gt, lt, ge, le | PID value |
| SESSION | eq, ne, gt, lt, ge, le | Session number |
| SESSIONNAME | eq, ne | Session name |
| CPUTIME | eq, ne, gt, lt, ge, le | CPU time in the format <i>HH:MM:SS</i> , where <i>MM</i> and <i>SS</i> are between 0 and 59 and <i>HH</i> is any unsigned number |
| MEMUSAGE | eq, ne, gt, lt, ge, le | Memory usage in KB |
| USERNAME | eq, ne | Any valid user name |
| SERVICES | eq, ne | Service name |
| WINDOWTITLE | eq, ne | Window title |
| MODULES | eq, ne | DLL name |

Remarks

The WINDOWTITLE and STATUS filters are not supported when a remote system is specified.

Examples

To list all tasks with a process ID greater than 1000, and display them in CSV format, type:

```
tasklist /v /fi "PID gt 1000" /fo csv
```

To list the system processes that are currently running, type:

```
tasklist /fi "USERNAME ne NT AUTHORITY\SYSTEM" /fi "STATUS eq running"
```

To list detailed information for all processes that are currently running, type:

```
tasklist /v /fi "STATUS eq running"
```

To list all the service information for processes on the remote computer "Srvmain" that have a DLL name beginning with "ntdll," type:

```
tasklist /s srvmain /svc /fi "MODULES eq ntdll*"
```

To list the processes on the remote computer "Srvmain," using the credentials of your currently logged-on user

account, type:

```
tasklist /s srvmain
```

To list the processes on the remote computer "Srvmain," using the credentials of the user account Hiropln, type:

```
tasklist /s srvmain /u maindom\hiropln /p p@ssw23
```

Additional references

[Command-Line Syntax Key](#)

tcmsetup

7/11/2018 • 2 minutes to read • [Edit Online](#)

Sets up or disables the TAPI client.

Syntax

```
tcmsetup [/q] [/x] /c <Server1> [<Server2> ...]  
tcmsetup [/q] /c /d
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------|---|
| /q | Prevents the display of message boxes. |
| /x | Specifies that connection-oriented callbacks will be used for heavy traffic networks where packet loss is high. When this parameter is omitted, connectionless callbacks will be used. |
| /c | Required. Specifies client setup. |
| <Server1> | Required. Specifies the name of the remote server that has the TAPI service providers that the client will use. The client will use the service providers' lines and phones. The client must be in the same domain as the server or in a domain that has a two-way trust relationship with the domain that contains the server. |
| <Server2>... | Specifies any additional server or servers that will be available to this client. If you specify a list of servers is, use a space to separate the server names. |
| /d | Clears the list of remote servers. Disables the TAPI client by preventing it from using the TAPI service providers that are on the remote servers. |
| /? | Displays help at the command prompt. |

Remarks

- To perform this procedure, you must be a member of the Administrators group on the local computer, or you must have been delegated the appropriate authority. If the computer is joined to a domain, members of the Domain Admins group might be able to perform this procedure. As a security best practice, consider using **Run as** to perform this procedure.
- In order for TAPI to function correctly, you must run **tcmsetup** to specify the remote servers that will be used by TAPI clients.
- Before a client user can use a phone or line on a TAPI server, the telephony server administrator must assign the user to the phone or line.
- The list of telephony servers that is created by this command replaces any existing list of telephony servers

available to the client. You cannot use this command to add to the existing list.

Additional references

[Command-Line Syntax Key](#)

[Command shell overview](#)

[Specify telephony servers on a client computer](#)

[Assign a telephony user to a line or phone](#)

telnet

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Communicates with a computer running the telnet Server service.

Syntax

```
telnet [/a] [/e <EscapeChar>] [/f <FileName>] [/l <UserName>] [/t {vt100 | vt52 | ansi | vtnt}] [<Host> [<Port>]] [/?]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------------------------|---|
| /a | attempt automatic logon. Same as /l option except uses the currently logged on user s name. |
| /e <EscapeChar> | Escape character used to enter the telnet client prompt. |
| /f <FileName> | File name used for client side logging. |
| /l <UserName> | Specifies the user name to log on with on the remote computer. |
| /t {vt100 vt52 ansi vtnt} | Specifies the terminal type. Supported terminal types are vt100, vt52, ansi, and vtnt. |
| <Host> [<Port>] | Specifies the hostname or IP address of the remote computer to connect to, and optionally the TCP port to use (default is TCP port 23). |
| /? | Displays help at the command prompt. Alternatively, you can type /h. |

Remarks

- You must install the telnet client software before you can run this command. For more information, see [Installing telnet](#).
- You can run telnet without parameters to enter the telnet context, indicated by the telnet prompt (**Microsoft telnet>**). From the telnet prompt, you can use telnet commands to manage the computer running the telnet client.

Examples

Use telnet to connect to the computer running the telnet Server Service at telnet.microsoft.com.

```
telnet telnet.microsoft.com
```

Use telnet to connect to the computer running the telnet Server Service at telnet.microsoft.com on TCP port 44 and log the session activity in a local file called telnetlog.txt

```
telnet /f telnetlog.txt telnet.microsoft.com 44
```

additional references

- [Installing telnet](#)
- [telnet Technical Reference](#)
- [Command-Line Syntax Key](#)

tftp

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Transfers files to and from a remote computer, typically a computer running UNIX, that is running the Trivial File Transfer Protocol (tftp) service or daemon. tftp is typically used by embedded devices or systems that retrieve firmware, configuration information, or a system image during the boot process from a tftp server.

Syntax

```
tftp [-i] [<Host>] [{get | put}] <Source> [<Destination>]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------|---|
| -i | Specifies binary image transfer mode (also called octet mode). In binary image mode, the file is transferred in one-byte units. Use this mode when transferring binary files. If -i is omitted, the file is transferred in ASCII mode. This is the default transfer mode. This mode converts the end-of-line (EOL) characters to an appropriate format for the specified computer. Use this mode when transferring text files. If a file transfer is successful, the data transfer rate is displayed. |
| <Host> | Specifies the local or remote computer. |
| put | Transfers the file <i>Source</i> on the local computer to the file <i>Destination</i> on the remote computer. Because the tftp protocol does not support user authentication, the user must be logged onto the remote computer, and the files must be writable on the remote computer. |
| get | Transfers the file <i>Destination</i> on the remote computer to the file <i>Source</i> on the local computer. |
| <Source> | Specifies the file to transfer. |
| <Destination> | Specifies where to transfer the file. |

Remarks

- You can install the tftp client using the add Features Wizard.
- The tftp protocol does not support any authentication or encryption mechanism, and as such can introduce a security risk when present. Installing the tftp client is not recommended for systems connected to the Internet.
- The tftp client is optional software, and marked as deprecated on Windows Vista and later versions of the Windows operating system. A tftp server service is no longer provided by Microsoft for security reasons.

Examples

Copy the file **boot.img** from the remote computer **Host1**.

```
tftp -i Host1 get boot.img
```

Additional references

- [Command-Line Syntax Key](#)

time

7/11/2018 • 2 minutes to read • [Edit Online](#)

Displays or sets the system time. If used without parameters, **time** displays the current system time and prompts you to enter a new time.

For examples of how to use this command, see [Examples](#).

Syntax

```
time [/t | [<HH>[:<MM>[:<SS>]] [<am|pm>]]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------------------------|--|
| <HH>[:<MM>[:<SS>[:<NN>]]] [am pm] | Sets the system time to the new time specified, where <i>HH</i> is in hours (required), <i>MM</i> is in minutes, and <i>SS</i> is in seconds. <i>NN</i> can be used to specify hundredths of a second. If am or pm is not specified, time uses the 24-hour format by default. |
| /t | Displays the current time without prompting you for a new time. |
| /? | Displays help at the command prompt. |

Remarks

- To change the current time, you must have administrative credentials.
- You must separate values for *HH*, *MM*, and *SS* with colons (:). *SS* and *NN* must be separated with a period (.).
- Valid *HH* values are 0 through 24.
- Valid *MM* and *SS* values are 0 through 59.

Examples

If command extensions are enabled, to display the current system time, type:

```
time /t
```

To change the current system time to 5:30 P.M., type either of the following:

```
time 17:30:00
time 5:30 pm
```

To display the current system time, followed by a prompt to enter a new time, type:

```
The current time is: 17:33:31.35
Enter the new time:
```

To keep the current time and return to the command prompt, press ENTER. To change the current time, type the new time and then press ENTER.

Additional references

[Command-Line Syntax Key](#)

timeout

4/13/2018 • 2 minutes to read • [Edit Online](#)

Pauses the command processor for the specified number of seconds.

For examples of how to use this command, see [Examples](#).

Syntax

```
timeout /t <TimeoutInSeconds> [/nobreak]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------------|--|
| /t <TimeoutInSeconds> | Specifies the decimal number of seconds (between -1 and 99999) to wait before the command processor continues processing. The value -1 causes the computer to wait indefinitely for a keystroke. |
| /nobreak | Specifies to ignore user key strokes. |
| /? | Displays help at the command prompt. |

Remarks

- The **timeout** command is typically used in batch files.
- A user keystroke resumes the command processor execution immediately, even if the timeout period has not expired.
- When used in conjunction with the **sleep** command, **timeout** is similar to the **pause** command.

Examples

To pause the command processor for ten seconds, type:

```
timeout /t 10
```

To pause the command processor for 100 seconds and ignore any keystroke, type:

```
timeout /t 100 /nobreak
```

To pause the command processor indefinitely until a key is pressed, type:

```
timeout /t -1
```

Additional references

[Command-Line Syntax Key](#)

title

4/13/2018 • 2 minutes to read • [Edit Online](#)

Creates a title for the Command Prompt window.

For examples of how to use this command, see [Examples](#).

Syntax

```
title [<String>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| <String> | Specifies the title of the Command Prompt window. |
| /? | Displays help at the command prompt. |

Remarks

- To create window title for batch programs, include the **title** command at the beginning of a batch program.
- After a window title is set, you can reset it only by using the **title** command.

Examples

In the following sample script, the title of the Command Prompt window is changed to "Updating Files" while the batch file executes the **copy** command. After the command is executed, the text `Files Updated` is displayed, and the title of the Command Prompt window is changed back to "Command Prompt."

```
@echo off
title Updating Files
copy \\server\share\*.xls c:\users\common\*.xls
echo Files Updated.
title Command Prompt
```

Additional references

[Command-Line Syntax Key](#)

tlntadmn

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Administers a local or remote computer that is running the telnet Server Service.

Syntax

```
tlntadmn [<computerName>] [-u <UserName>] [-p <Password>] [{start | stop | pause | continue}] [-s {<SessionID> | all}] [-k {<SessionID> | all}] [-m {<SessionID> | all} <Message>] [config [dom = <Domain>] [ctrlakeymap = {yes | no}] [timeout = <hh>:<mm>:<ss>] [timeoutactive = {yes | no}] [maxfail = <attempts>] [maxconn = <Connections>] [port = <Number>] [sec {+ | -}NTLM {+ | -}passwd] [mode = {console | stream}]] [-?]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------------------------|--|
| <computerName> | Specifies the name of the server to connect to. The default is the local computer. |
| -u <UserName> -p <Password> | Specifies administrative credentials for a remote server that you want to administer. This parameter is required if you want to administer a remote server to which you are not logged on with administrative credentials. |
| start | starts the telnet Server Service. |
| stop | Stops the telnet Server Service |
| pause | pauses the telnet Server Service. No new connections will be accepted. |
| continue | Resumes the telnet Server Service. |
| -s {<SessionID> all} | Displays active telnet sessions. |
| -k {<SessionID> all} | Ends telnet sessions. type the Session ID to end a specific session, or type all to end all the sessions. |
| -m {<SessionID> all} | Sends a message to one or more sessions. type the session ID to send a message to a specific session, or type all to send a message to all sessions. type the message that you want to send between quotation marks. |
| config dom = <Domain> | Configures the default domain for the server. |
| config ctrlakeymap = {yes no} | Specifies if you want the telnet server to interpret CTRL+A as ALT. type yes to map the shortcut key, or type no to prevent the mapping. |

| PARAMETER | DESCRIPTION |
|--------------------------------------|--|
| config timeout = <hh>:<mm>:<ss> | Sets the time-out period in hours, minutes, and seconds. |
| config timeoutactive = {yes no} | Enables the idle session timeout. |
| config maxfail = <attempts> | Sets the maximum number of failed logon attempts before disconnecting. |
| config maxconn = <Connections> | Sets the maximum number of connections. |
| config port = <\Number> | Sets the telnet port. You must specify the port with an integer smaller than 1024. |
| config sec {+ -}NTLM {+ -}passwd | Specifies whether you want to use NTLM, a password, or both to authenticate logon attempts. To use a particular type of authentication, type a plus sign (+) before that type of authentication. To prevent using a particular type of authentication, type a minus sign (-) before that type of authentication. |
| config mode = {console stream} | Specifies the mode of operation. |
| -? | Displays help at the command prompt. |

Remarks

- To display the server settings, type **tlntadm** without any parameters.
- To use the **tlntadm** command, you must log on to the local computer with administrative credentials. To administer a remote computer, you must also provide administrative credentials for the remote computer. You can do so by logging on to the local computer with an account that has administrative credentials for both the local computer and the remote computer. If you cannot use this method, you can use the **-u** and **-p** parameters to provide administrative credentials for the remote computer.

Examples

Configure the idle session timeout to 30 minutes.

```
tlntadm config timeout=0:30:0
```

Display active telnet sessions.

```
tlntadm -s
```

Additional references

- [telnet Operations Guide](#)
- [Command-Line Syntax Key](#)

tpmvscmgr

7/11/2018 • 3 minutes to read • [Edit Online](#)

The Tpmvscmgr command-line tool allows users with Administrative credentials to create and delete TPM virtual smart cards on a computer. For examples of how this command can be used, see [Examples](#).

Syntax

```
Tpmvscmgr create [/name] [/AdminKey DEFAULT | PROMPT | RANDOM] [/PIN DEFAULT | PROMPT] [/PUK DEFAULT | PROMPT] [/generate] [/machine] [/?]
```

```
Tpmvscmgr destroy [/instance <instance ID>] [/?]
```

Parameters for Create command

The Create command sets up new virtual smart cards on the user's system. It returns the instance ID of the newly created card for later reference if deletion is required. The instance ID is in the format

ROOT\SMARTCARDREADER\000n where **n** starts from 0 and is increased by 1 each time you create a new virtual smart card.

| PARAMETER | DESCRIPTION |
|-----------|--|
| /name | Required. Indicates the name of the new virtual smart card. |
| /AdminKey | Indicates the desired administrator key that can be used to reset the PIN of the card if the user forgets the PIN. DEFAULT Specifies the default value of 010203040506070801020304050607080102030405060708. PROMPT Prompts the user to enter a value for the administrator key. RANDOM Results in a random setting for the administrator key for a card that is not returned to the user. This creates a card that might not be manageable by using smart card management tools. When generated with RANDOM, the administrator key must be entered as 48 hexadecimal characters. |
| /PIN | Indicates desired user PIN value. DEFAULT Specifies the default PIN of 12345678. PROMPT Prompts the user to enter a PIN at the command line. The PIN must be a minimum of eight characters, and it can contain numerals, characters, and special characters. |
| /PUK | Indicates the desired PIN Unlock Key (PUK) value. The PUK value must be a minimum of eight characters, and it can contain numerals, characters, and special characters. If the parameter is omitted, the card is created without a PUK. DEFAULT Specifies the default PUK of 12345678. PROMPT Prompts to the user to enter a PUK at the command line. |

| PARAMETER | DESCRIPTION |
|-----------|---|
| /generate | Generates the files in storage that are necessary for the virtual smart card to function. If the /generate parameter is omitted, it is equivalent to creating a card without this file system. A card without a file system can be managed only by a smart card management system such as Microsoft Configuration Manager. |
| /machine | Allows you to specify the name of a remote computer on which the virtual smart card can be created. This can be used in a domain environment only, and it relies on DCOM. For the command to succeed in creating a virtual smart card on a different computer, the user running this command must be a member in the local administrators group on the remote computer. |
| /? | Displays Help for this command. |

Parameters for Destroy command

The Destroy command securely deletes a virtual smart card from the user's computer.

WARNING

When a virtual smart card is deleted, it cannot be recovered.

| PARAMETER | DESCRIPTION |
|-----------|--|
| /instance | Specifies the instance ID of the virtual smart card to be removed. The instanceID was generated as output by Tpmvscmgr.exe when the card was created. The /instance parameter is a required field for the Destroy command. |
| /? | Displays Help for this command. |

Remarks

Membership in the **Administrators** group (or equivalent) on the target computer is the minimum required to run all the parameters of this command.

For alphanumeric inputs, the full 127 character ASCII set is allowed.

Examples

The following command shows how to create a virtual smart card that can be later managed by a smart card management tool launched from another computer.

```
tpmvscmgr.exe create /name "VirtualSmartCardForCorpAccess" /AdminKey DEFAULT /PIN PROMPT
```

Alternatively, instead of using a default administrator key, you can create an administrator key at the command line. The following command shows how to create an administrator key.

```
tpmvscmgr.exe create /name "VirtualSmartCardForCorpAccess" /AdminKey PROMPT /PIN PROMPT
```

The following command will create the unmanaged virtual smart card that can be used to enroll certificates.

```
tpmvscmgr.exe create /name "VirtualSmartCardForCorpAccess" /AdminKey RANDOM /PIN PROMPT /generate
```

The following command will create a virtual smart card with a randomized administrator key. The key is automatically discarded after the card is created. This means that if the user forgets the PIN or wants to change the PIN, the user needs to delete the card and create it again. To delete the card, the user can run the following command.

```
tpmvscmgr.exe destroy /instance <instance ID>
```

where <instance ID> is the value printed on the screen when the user created the card. Specifically, for the first card created, the instance ID is ROOT\SMARTCARDREADER\0000.

Additional references

- [Command-Line Syntax Key](#)

tracertp

4/13/2018 • 2 minutes to read • [Edit Online](#)

The **tracertp** command can be used to parse Event Trace Logs, log files generated by Performance Monitor, and real-time Event Trace providers. It generates dump files, report files, and report schemas.

For examples of how to use **tracertp**, see [Examples](#).

Syntax

```
tracertp <[-l] <value [value [...]]>|-rt <session_name [session_name [...]]>> [options]
```

Options

| OPTION FLAG | DESCRIPTION |
|--------------------|---|
| -? | Displays context sensitive help. |
| -config <filename> | Load a settings file containing command options. |
| -y | Answer yes to all questions without prompting. |
| -f <XML | HTML> |
| -of <CSV | EVTX |
| -df <filename> | Create a Microsoft-specific counting/reporting schema file. |
| -int <filename> | Dump the interpreted event structure to the specified file. |
| -rts | Report raw timestamp in the event trace header. Can only be used with -o, not -report or -summary. |
| -tmf <filename> | Specify a Trace Message Format definition file. |
| -tp <value> | Specify the TMF file search path. Multiple paths may be used, separated by a semicolon (;). |
| -i <value> | Specify the provider image path. The matching PDB will be located in the Symbol Server. Multiple paths can be used, separated by a semicolon (;). |
| -pdb <value> | Specify the symbol server path. Multiple paths can be used, separated by a semicolon (;). |
| -gmt | Convert WPP payload timestamps to Greenwich Mean Time. |
| -rl <value> | Define System Report Level from 1 to 5. Default is 1. |

| OPTION FLAG | DESCRIPTION |
|---|--|
| -summary [filename] | Generate a summary report text file. Filename if not specified is summary.txt. |
| -o [filename] | Generate a text output file. Filename if not specified is dumpfile.xml. |
| -report [filename] | Generate a text output report file. Filename if not specified is workload.xml. |
| -lr | Specify "less restrictive." This uses best efforts for events that do not match the events schema. |
| -export [filename] | Generate an Event Schema export file. Filename if not specified is schema.man. |
| [-l] <value [value [...]]> | Specify the Event Trace log file to process. |
| -rt <session_name [session_name [...]]> | Specify Real-time Event Trace Session data sources. |

Examples

- This example creates a report based on the two event logs **logfile1.etl** and **logfile2.etl** and creates the dump file **logdump.xml** in XML format.

```
tracertpt logfile1.etl logfile2.etl -o logdump.xml -of XML
```

- This example creates a report based on the event log **logfile.etl**, creates the dump file **logdmp.xml** in XML format, uses best efforts to identify events not in the schema, produces a summary report file **logdump.txt**, and produces the report file **logrpt.xml**.

```
tracertpt logfile.etl -o logdmp.xml -of XML -lr -summary logdmp.txt -report logrpt.xml
```

- This example uses the two event logs **logfile1.etl** and **logfile2.etl** to produce a dump file and report file with the default filenames.

```
tracertpt logfile1.etl logfile2.etl -o -report
```

- This example uses the event log **logfile.etl** and the performance log **counterfile.blg** to produce the report file **logrpt.xml** and the Microsoft-specific XML schema file **schema.xml**.

```
tracertpt logfile.etl counterfile.blg -report logrpt.xml -df schema.xml
```

- This example reads the real-time Event Trace Session "NT Kernel Logger" and produces the dump file **logfile.csv** in CSV format.

```
tracertpt -rt "NT Kernel Logger" -o logfile.csv -of CSV
```

tracert

7/11/2018 • 3 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Determines the path taken to a destination by sending Internet Control Message Protocol (ICMP) echo Request or ICMPv6 messages to the destination with incrementally increasing time to Live (TTL) field values. The path displayed is the list of near/side router interfaces of the routers in the path between a source host and a destination. The near/side interface is the interface of the router that is closest to the sending host in the path. Used without parameters, tracert displays help.

Syntax

```
tracert [/d] [/h <MaximumHops>] [/j <Hostlist>] [/w <timeout>] [/R] [/S <Srcaddr>] [/4]/[6] <TargetName>
```

Parameters

| PARAMETER | DESCRIPTION |
|------------------|--|
| /d | Prevents tracert from attempting to resolve the IP addresses of intermediate routers to their names. This can speed up the display of tracert results. |
| /h <MaximumHops> | Specifies the maximum number of hops in the path to search for the target (destination). The default is 30 hops. |
| /j <Hostlist> | Specifies that echo Request messages use the Loose Source Route option in the IP header with the set of intermediate destinations specified in <i>Hostlist</i> . With loose source routing, successive intermediate destinations can be separated by one or multiple routers. The maximum number of addresses or names in the host list is 9. The <i>Hostlist</i> is a series of IP addresses (in dotted decimal notation) separated by spaces. Use this parameter only when tracing IPv4 addresses. |
| /w <timeout> | Specifies the amount of time in milliseconds to wait for the ICMP time Exceeded or echo Reply message corresponding to a given echo Request message to be received. If not received within the time-out, an asterisk (*) is displayed. The default time-out is 4000 (4 seconds). |
| /R | Specifies that the IPv6 Routing extension header be used to send an echo Request message to the local host, using the destination as an intermediate destination and testing the reverse route. |
| /S <Srcaddr> | Specifies the source address to use in the echo Request messages. Use this parameter only when tracing IPv6 addresses. |

| PARAMETER | DESCRIPTION |
|--------------|--|
| /4 | Specifies that tracert.exe can use only IPv4 for this trace. |
| /6 | Specifies that tracert.exe can use only IPv6 for this trace. |
| <TargetName> | Specifies the destination, identified either by IP address or host name. |
| /? | Displays help at the command prompt. |

Remarks

- This diagnostic tool determines the path taken to a destination by sending ICMP echo Request messages with varying time to Live (TTL) values to the destination. Each router along the path is required to decrement the TTL in an IP packet by at least 1 before forwarding it. Effectively, the TTL is a maximum link counter. When the TTL on a packet reaches 0, the router is expected to return an ICMP time Exceeded message to the source computer. tracert determines the path by sending the first echo Request message with a TTL of 1 and incrementing the TTL by 1 on each subsequent transmission until the target responds or the maximum number of hops is reached. The maximum number of hops is 30 by default and can be specified using the **/h** parameter. The path is determined by examining the ICMP time Exceeded messages returned by intermediate routers and the echo Reply message returned by the destination. However, some routers do not return time Exceeded messages for packets with expired TTL values and are invisible to the tracert command. In this case, a row of asterisks (*) is displayed for that hop.
- To trace a path and provide network latency and packet loss for each router and link in the path, use the **pathping** command.
- This command is available only if the Internet Protocol (TCP/IP) protocol is installed as a component in the properties of a network adapter in Network Connections.

Examples

To trace the path to the host named corp7.microsoft.com, type:

```
tracert corp7.microsoft.com
```

To trace the path to the host named corp7.microsoft.com and prevent the resolution of each IP address to its name, type:

```
tracert /d corp7.microsoft.com
```

To trace the path to the host named corp7.microsoft.com and use the loose source route 10.12.0.1/10.29.3.1/10.1.44.1, type:

```
tracert /j 10.12.0.1 10.29.3.1 10.1.44.1 corp7.microsoft.com
```

Additional references

- [Command-Line Syntax Key](#)

tree

4/13/2018 • 2 minutes to read • [Edit Online](#)

Displays the directory structure of a path or of the disk in a drive graphically.

For examples of how to use this command, see [Examples](#).

Syntax

```
tree [<Drive>:][<Path>] [/f] [/a]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| <Drive>: | Specifies the drive that contains the disk for which you want to display the directory structure. |
| <Path> | Specifies the directory for which you want to display the directory structure. |
| /f | Displays the names of the files in each directory. |
| /a | Specifies that tree is to use text characters instead of graphic characters to show the lines that link subdirectories. |
| /? | Displays help at the command prompt. |

Remarks

The structure displayed by **tree** depends upon the parameters that you specify at the command prompt. If you do not specify a drive or path, **tree** displays the tree structure beginning with the current directory of the current drive.

Examples

To display the names of all the subdirectories on the disk in your current drive, type:

```
tree \
```

To display, one screen at a time, the files in all the directories on drive C, type:

```
tree c:\ /f | more
```

To print a list of all the directories on drive C, type:

```
tree c:\ /f prn
```

Additional references

[Command-Line Syntax Key](#)

tscon

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Connects to another session on a Remote Desktop Session Host (rd Session Host) server.
for examples of how to use this command, see [Examples](#).

NOTE

In Windows Server 2008 R2, Terminal Services was renamed Remote Desktop Services. To find out what's new in the latest version, see [What's New in Remote Desktop Services in Windows Server 2012](#) in the Windows Server TechNet Library.

Syntax

```
tscon {<SessionID> | <SessionName>} [/dest:<SessionName>] [/password:<pw> | /password:*] [/v]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------------|---|
| <SessionID> | Specifies the ID of the session to which you want to connect. If you use the optional /dest:<SessionName> parameter, this is the ID of the session to which you want to connect. |
| <SessionName> | Specifies the name of the session to which you want to connect. |
| /dest:<SessionName> | Specifies the name of the current session. This session will disconnect when you connect to the new session. |
| /password:<pw> | Specifies the password of the user who owns the session to which you want to connect. This password is required when the connecting user does not own the session. |
| /password:* | prompts for the password of the user who owns the session to which you want to connect. |
| /v | Displays information about the actions being performed. |
| /? | Displays help at the command prompt. |

Remarks

- You must have Full Control access permission or Connect special access permission to connect to another session.
- The **/dest:<SessionName>** parameter allows you to connect the session of another user to a different session.

- if you do not specify a password in the *<Password>* parameter, and the target session belongs to a user other than the current one, **tscon** fails.
- You cannot connect to the console session.

Examples

- To connect to session 12 on the current rd Session Host server and disconnect the current session, type:

```
tscon 12
```

- To connect to session 23 on the current rd Session Host server, by using the password mypass, and disconnect the current session, type:

```
tscon 23 /password:mypass
```

- To connect the session named TERM03 to the session named TERM05, and then disconnect session TERM05, if it is connected, type:

```
tscon TERM03 /v /dest:TERM05
```

Additional references

[Command-Line Syntax Key](#)

[Remote Desktop Services \(Terminal Services\) Command Reference](#)

tsdiscon

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Disconnects a session from a Remote Desktop Session Host (rd Session Host) server. For examples of how to use this command, see [Examples](#).

NOTE

In Windows Server 2008 R2, Terminal Services was renamed Remote Desktop Services. To find out what's new in the latest version, see [What's New in Remote Desktop Services in Windows Server 2012](#) in the Windows Server TechNet Library.

Syntax

```
tsdiscon [<SessionID> | <SessionName>] [/server:<ServerName>] [/v]
```

Parameters

| PARAMETER | DESCRIPTION |
|----------------------|---|
| <SessionId> | Specifies the ID of the session to disconnect. |
| <SessionName> | Specifies the name of the session to disconnect. |
| /server:<ServerName> | Specifies the terminal server that contains the session that you want to disconnect. Otherwise, the current rd Session Host server is used. |
| /v | Displays information about the actions being performed. |
| /? | Displays help at the command prompt. |

Remarks

- You must have Full Control permission or Disconnect special access permission to disconnect another user from a session.
- If no session ID or session name is specified, **tsdiscon** disconnects the current session.
- Any applications that were running when you disconnected the session are automatically running when you reconnect to that session with no loss of data. Use **reset session** to end the running applications of the disconnected session, but be aware that this might result in loss of data at the session.
- The **/server** parameter is required only if you use **tsdiscon** from a remote server.
- The console session cannot be disconnected.

Examples

- To disconnect the current session, type: `tsdiscon`
- To disconnect session 10, type: `tsdiscon 10`
- To disconnect the session named TERM04, type: `tsdiscon TERM04` ##### Additional references [Command-Line Syntax Key Remote Desktop Services \(Terminal Services\) Command Reference](#)

tsecimp

7/11/2018 • 3 minutes to read • [Edit Online](#)

Imports assignment information from an Extensible Markup Language (XML) file into the TAPI server security file (Tsec.ini). You can also use this command to display the list of TAPI providers and the lines devices associated with each of them, validate the structure of the XML file without importing the contents, and check domain membership.

Syntax

```
tsecimp /f <Filename> [{/v | /u}]  
tsecimp /d
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------|---|
| /f <Filename> | Required. Specifies the name of the XML file that contains the assignment information that you want to import. |
| /v | Validates the structure of the XML file without importing the information into the Tsec.ini file. |
| /u | Checks whether each user is a member of the domain specified in the XML file. The computer on which you use this parameter must be connected to the network. This parameter might significantly slow performance if you are processing a large amount of user assignment information. |
| /d | Displays a list of installed telephony providers. For each telephony provider, the associated line devices are listed, as well as the addresses and users associated with each line device. |
| /? | Displays help at the command prompt. |

Remarks

- The XML file from which you want to import assignment information must follow the structure described below.

- **UserList** element

The **UserList** is the top element of the XML file.

- **User** element

Each **User** element contains information about a user who is a member of a domain. Each user might be assigned one or more line devices.

Additionally, each **User** element might have an attribute named **NoMerge**. When this attribute is specified, all current line device assignments for the user are removed before new ones are made. You

can use this attribute to easily remove unwanted user assignments. By default, this attribute is not set.

The **User** element must contain a single **DomainUserName** element, which specifies the domain and user name of the user. The **User** element might also contain one **FriendlyName** element, which specifies a friendly name for the user.

The **User** element might contain one **LineList** element. If a **LineList** element is not present, all line devices for this user are removed.

- **LineList** element

The **LineList** element contains information about each line or device that might be assigned to the user. Each **LineList** element can contain more than one **Line** element.

- **Line** element

Each **Line** element specifies a line device. You must identify each line device by adding either an **Address** element or a **PermanentID** element under the **Line** element.

For each **Line** element, you can set the **Remove** attribute. If you set this attribute, the user is no longer assigned that line device. If this attribute is not set, the user gains access to that line device. No error is given if the line device is not available to the user.

Examples

- The following sample XML code segments illustrate correct usage of the elements defined above.

- The following code removes all line devices assigned to User1.

```
<UserList> <User NoMerge="1"> <DomainUser>domain1\user1</DomainUser> </User> </UserList>
```

- The following code removes all line devices assigned to User1 before assigning one line with address 99999. User1 will have no other lines devices assigned, regardless of whether any line devices were assigned previously.

```
<UserList>
  <User NoMerge="1">
    <DomainUser>domain1\user1</DomainUser>
    <FriendlyName>User1</FriendlyName>
    <LineList>
      <Line>
        <Address>99999</Address>
      </Line>
    </LineList>
  </User>
</UserList>
```

- The following code adds one line device for User1 without deleting any previously assigned line devices.

```
<UserList>
  <User>
    <DomainUser>domain1\user1</DomainUser>
    <FriendlyName>User1</FriendlyName>
    <LineList>
      <Line>
        <Address>99999</Address>
      </Line>
    </LineList>
  </User>
</UserList>
```

- The following code adds line address 99999 and removes line address 88888 from User1's access.

```
<UserList>
  <User>
    <DomainUser>domain1\user1</DomainUser>
    <FriendlyName>User1</FriendlyName>
    <LineList>
      <Line>
        <Address>99999</Address>
      </Line>
      <Line Remove="1">
        <Address>88888</Address>
      </Line>
    </LineList>
  </User>
</UserList>
```

- The following code adds permanent device 1000 and removes line 88888 from User1's access.

```
"" domain1\user1 User1 1000
88888
```

```
...
```

- The following sample output appears after the **/d** command-line option is specified to display the current TAPI configuration. For each telephony provider, the associated line devices are listed, as well as the addresses and users associated with each line device.

```
NDIS Proxy TAPI Service Provider
  Line: "WAN Miniport (L2TP)"
    Permanent ID: 12345678910

NDIS Proxy TAPI Service Provider
  Line: "LPT1DOMAIN1\User1"
    Permanent ID: 12345678910

Microsoft H.323 Telephony Service Provider
  Line: "H323 Line"
    Permanent ID: 123456
  Addresses:
    BLDG1-TAPI32
```

Additional references

[Command-Line Syntax Key](#)

[Command shell overview](#)

tskill

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Ends a process running in a session on a Remote Desktop Session Host (rd Session Host) server. for examples of how to use this command, see [Examples](#).

NOTE

In Windows Server 2008 R2, Terminal Services was renamed Remote Desktop Services. To find out what's new in the latest version, see [What's New in Remote Desktop Services in Windows Server 2012](#) in the Windows Server TechNet Library.

Syntax

```
tskill {<ProcessID> | <ProcessName>} [/server:<ServerName>] [/id:<SessionID> | /a] [/v]
```

Parameters

| PARAMETER | DESCRIPTION |
|----------------------|---|
| <ProcessID> | Specifies the ID of the process that you want to end. |
| <ProcessName> | Specifies the name of the process that you want to end. This parameter can include wildcard characters. |
| /server:<ServerName> | Specifies the terminal server that contains the process that you want to end. If /server is not specified, the current RD Session Host server is used. |
| /id:<SessionID> | Ends the process that is running in the specified session. |
| /a | Ends the process that is running in all sessions. |
| /v | Displays information about the actions being performed. |
| /? | Displays help at the command prompt. |

Remarks

- You can use **tskill** to end only those processes that belong to you, unless you are an administrator. Administrators have full access to all **tskill** functions and can end processes that are running in other user sessions.
- When all processes that are running in a session end, the session also ends.
- if you use the *ProcessName* and the **/server:ServerName** parameters, you must also specify either the **/id:SessionID** or the **/a** parameter.

Examples

- To end process 6543, type: `tskill 6543`
- To end the process "explorer" running on session 5, type: `tskill explorer /id:5` ##### Additional references
[Command-Line Syntax Key Remote Desktop Services \(Terminal Services\) Command Reference](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Copies the Remote Desktop Services user configuration information from one user to another. The Remote Desktop Services user configuration information is displayed in the Remote Desktop Services extensions to Local Users and Groups and active directory Users and computers.

tsprof can also set the profile path for a user.

For examples of how to use this command, see [Examples](#).

NOTE

In Windows Server 2008 R2, Terminal Services was renamed Remote Desktop Services. To find out what's new in the latest version, see [What's New in Remote Desktop Services in Windows Server 2012](#) in the Windows Server TechNet Library.

Syntax

```
tsprof /update {/domain:<DomainName> | /local} /profile:<path> <UserName>
tsprof /copy {/domain:<DomainName> | /local} [/profile:<path>] <Src_usr> <Dest_usr>
tsprof /q {/domain:<DomainName> | /local} <UserName>
```

Parameters

| PARAMETER | DESCRIPTION |
|----------------------|---|
| /update | Updates profile path information for <UserName> in domain <DomainName> to <Profilepath>. |
| /domain:<DomainName> | Specifies the name of the domain in which the operation is applied. |
| /local | Applies the operation only to local user accounts. |
| /profile:<path> | Specifies the profile path as displayed in the Remote Desktop Services extensions in Local Users and Groups and active directory Users and computers. |
| <UserName> | Specifies the name of the user for whom you want to update or query the server profile path. |
| /copy | Copies user configuration information from <SourceUser> to <DestinationUser> and updates the profile path information for <DestinationUser> to <Profilepath>. Both <SourceUser> and <DestinationUser> must either be local or must be in domain <DomainName>. |

| PARAMETER | DESCRIPTION |
|------------|---|
| <Src_usr> | Specifies the name of the user from whom you want to copy the user configuration information. |
| <Dest_usr> | Specifies the name of the user to whom you want to copy the user configuration information. |
| /q | Displays the current profile path of the user for whom you want to query the server profile path. |
| /? | Displays help at the command prompt. |

Remarks

- The **tsprof** command is only available when you have installed the Terminal Server role service on a computer running Windows Server 2008 or RD Session Host role service on a computer running Windows Server 2008 R2.

Examples

- To copy user configuration information from LocalUser1 to LocalUser2, type:

```
tsprof /copy /local LocalUser1 LocalUser2
```
- To set the Remote Desktop Services profile path for LocalUser1 to a directory called "c:\profiles," type:

```
tsprof /update /local /profile:c:\profiles LocalUser1
```

Additional references

[Command-Line Syntax Key Remote Desktop Services \(Terminal Services\) Command Reference](#)

type

10/24/2018 • 2 minutes to read • [Edit Online](#)

In the Windows Command shell, **type** is a built in command which displays the contents of a text file. Use the **type** command to view a text file without modifying it.

In PowerShell, **type** is a built-in alias to the **Get-Content** cmdlet which also displays the contents of a file, but with a different syntax.

For examples of how to use this command within the Windows Command shell (Cmd.exe), see [Examples](#).

Syntax

```
type [<Drive>:][<Path>]<FileName>
```

Parameters

| PARAMETER | DESCRIPTION |
|------------------------------|---|
| [<Drive>:][<Path>]<FileName> | Specifies the location and name of the file or files that you want to view. Separate multiple file names with spaces. |
| /? | Displays help at the command prompt. |

Remarks

- If *FileName* contains spaces, enclose it in quotation marks (for example, "File Name Containing Spaces.txt").
- If you display a binary file or a file that is created by a program, you may see strange characters on the screen, including formfeed characters and escape-sequence symbols. These characters represent control codes that are used in the binary file. In general, avoid using the **type** command to display binary files.

Examples

To display the contents of a file named Holiday.mar, type:

```
type holiday.mar
```

To display the contents of a lengthy file named Holiday.mar one screen at a time, type:

```
type holiday.mar | more
```

Additional references

[Command-Line Syntax Key](#)

typeperf

7/12/2018 • 2 minutes to read • [Edit Online](#)

The **typeperf** command writes performance data to the command window or to a log file. To stop **typeperf**, press CTRL+C.

For examples of how to use **typeperf**, see [Examples](#).

Syntax

```
typeperf <counter [counter ...]> [options]
typeperf -cf <filename> [options]
typeperf -q [object] [options]
typeperf -qx [object] [options]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------------------|--|
| <counter [counter [...]]> | Specifies performance counters to monitor. |

NOTE

<counter> is the full name of a performance counter in *\\Computer\Object(Instance)\Counter* format, such as **\\Server1\Processor(0)% User Time**.

Options

| OPTION | DESCRIPTION |
|----------------------|--|
| -? | Displays context-sensitive help. |
| -f <CSV TSV BIN SQL> | Specifies the output file format. The default is CSV. |
| -cf <filename> | Specifies a file containing a list of performance counters to monitor, with one counter per line. |
| -si <[[hh:]mm:]ss> | Specifies the sample interval. The default is one second. |
| -o <filename> | Specifies the path for the output file, or the SQL database. The default is STDOUT (written to the command window). |
| -q [object] | Display a list of installed counters (no instances). To list counters for one object, include the object name. ***EXAMPLE |
| -qx [object] | Display a list of installed counters with instances. To list counters for one object, include the object name. |

| OPTION | DESCRIPTION |
|--------------------|---|
| -sc <samples> | Specifies the number of samples to collect. The default is to collect data until CTRL+C is pressed. |
| -config <filename> | Specifies a settings file containing command options. |
| -s <computer_name> | Specifies a remote computer to monitor if no computer is specified in the counter path. |
| -y | Answer yes to all questions without prompting. |

Examples

- The following example writes the values for the local computer's performance counter **\\Processor(_Total)% Processor Time** to the command window at a default sample interval of 1 second until CTRL+C is pressed.

```
typeperf "\\Processor(_Total)\\% Processor Time"
```

- The following example writes the values for the list of counters in the file **counters.txt** to the tab-delimited file **domain2.tsv** at a sample interval of 5 seconds until 50 samples have been collected.

```
typeperf -cf counters.txt -si 5 -sc 50 -f TSV -o domain2.tsv
```

- The following example queries installed counters with instances for the counter object **PhysicalDisk** and writes the resulting list to the file **counters.txt**.

```
typeperf -qx PhysicalDisk -o counters.txt
```

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Displays the Windows time Zone Utility.

Syntax

```
tzutil [/?] [/g] [/s <timeZoneID>[_dstoff]] [/l]
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------------------|--|
| /? | Displays help at the command prompt. |
| /g | Displays the current time zone ID. |
| /s <timeZoneID>[_dstoff] | Sets the current time zone using the specified time zone ID. The _dstoff suffix disables Daylight Saving time adjustments for the time zone (where applicable). |
| /l | lists all valid time zone IDs and display names. The output will be: - <display name> - <time zone ID> |

Remarks

An exit code of **0** indicates the command completed successfully.

Examples

To display the current time zone ID, type:

```
tzutil /g
```

To set the current time zone to Pacific Standard time, type:

```
tzutil /s Pacific Standard time
```

To set the current time zone to Pacific Standard time and disable Daylight Saving time adjustments, type:

```
tzutil /s Pacific Standard time_dstoff
```

additional references

- [Command-Line Syntax Key](#)

Vssadmin

6/20/2018 • 2 minutes to read • [Edit Online](#)

Applies to: Windows 10, Windows 8.1, Windows Server 2016, Windows Server 2012 R2, Windows Server 2012, Windows Server 2008 R2, Windows Server 2008

Displays current volume shadow copy backups and all installed shadow copy writers and providers. Select a command name in the following table view its command syntax.

| COMMAND | DESCRIPTION |
|---|---|
| Vssadmin add shadowstorage | Adds a volume shadow copy storage association. |
| Vssadmin create shadow | Creates a new volume shadow copy. |
| Vssadmin delete shadows | Deletes volume shadow copies. |
| Vssadmin delete shadowstorage | Deletes volume shadow copy storage associations. |
| Vssadmin list providers | Lists registered volume shadow copy providers. |
| Vssadmin list shadows | Lists existing volume shadow copies. |
| Vssadmin list shadowstorage | Lists all shadow copy storage associations on the system. |
| Vssadmin list volumes | Lists volumes that are eligible for shadow copies. |
| Vssadmin list writers | Lists all subscribed volume shadow copy writers on the system. |
| Vssadmin resize shadowstorage | Resizes the maximum size for a shadow copy storage association. |

Vssadmin delete shadows

6/20/2018 • 2 minutes to read • [Edit Online](#)

Applies to: Windows 10, Windows 8.1, Windows Server 2016, Windows Server 2012 R2, Windows Server 2012, Windows Server 2008 R2, Windows Server 2008

Deletes a specified volume's shadow copies.

Syntax

```
vssadmin delete shadows /for=<ForVolumeSpec> [/oldest | /all | /shadow=<ShadowID>] [/quiet]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------------|--|
| /for= <ForVolumeSpec> | Specifies which volume's shadow copy will be deleted. |
| /oldest | Deletes only the oldest shadow copy. |
| /all | Deletes all of the specified volume's shadow copies. |
| /shadow= <ShadowID> | Deletes the shadow copy specified by ShadowID. To get the shadow copy ID, use the vssadmin list shadows command. When you enter a shadow copy ID, use the following format, where each <i>X</i> represents a hexadecimal character: XXXXXXXX-XXXX-XXXX-XXXX-XXXXXXXXXXXX |
| /quiet | Specifies that the command will not display messages while running. |

Remarks

You can only delete shadow copies with the client-accessible type.

Examples

To delete the oldest shadow copy of volume C, enter this command:

```
vssadmin delete shadows /for=c: /oldest
```

Additional references

- [Command-line syntax key](#)
- [Vssadmin](#)
- [Vssadmin list shadows](#)

Vssadmin list shadows

6/20/2018 • 2 minutes to read • [Edit Online](#)

Applies to: Windows 10, Windows 8.1, Windows Server 2016, Windows Server 2012 R2, Windows Server 2012, Windows Server 2008 R2, Windows Server 2008

Lists all existing shadow copies of a specified volume. If you use this command without parameters, it displays all volume shadow copies on the computer in the order dictated by **Shadow Copy Set**.

Syntax

```
vssadmin list shadows [/for=<ForVolumeSpec>] [/shadow=<ShadowID>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------------|--|
| /for= <ForVolumeSpec> | Specifies which volume the shadow copies will be listed for. |
| /shadow= <ShadowID> | <p>Lists the shadow copy specified by ShadowID. To get the shadow copy ID, use the vssadmin list shadows command. When you type a shadow copy ID, use the following format, where each <i>X</i> represents a hexadecimal character:</p> |
| | XXXXXXXX-XXXX-XXXX-XXXX-XXXXXXXXXXXX |

Additional references

- [Command-line syntax key](#)
- [Vssadmin](#)

Vssadmin list writers

6/20/2018 • 2 minutes to read • [Edit Online](#)

Applies to: Windows 10, Windows 8.1, Windows Server 2016, Windows Server 2012 R2, Windows Server 2012, Windows Server 2008 R2, Windows Server 2008

Lists subscribed volume shadow copy writers.

Syntax

```
vssadmin list writers
```

Parameters

No parameters.

Additional references

- [Command-line syntax key](#)
- [Vssadmin](#)

unlodctr

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Removes Performance counter names and Explain text for a service or device driver from the system registry.

Syntax

```
Unlodctr <DriverName>
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------|---|
| <DriverName> | removes the Performance counter name settings and Explain text for driver or service from the Windows Server 2003 registry. |
| /? | Displays help at the command prompt. |

Remarks

WARNING

Incorrectly editing the registry may severely damage your system. Before making changes to the registry, you should back up any valued data on the computer.

If the information that you supply contains spaces, use quotation marks around the text (for example, "").

Examples

To remove the current Performance registry settings and counter Explain text for the Simple Mail Transfer Protocol (SMTP) service:

```
unlodctr SMTPSVC
```

Additional references

- [Command-Line Syntax Key](#)

ver

10/24/2018 • 2 minutes to read • [Edit Online](#)

Displays the operating system version number.

This command is supported in the Windows Command prompt (Cmd.exe), but not in PowerShell.

For examples of how to use this command, see [Examples](#).

Syntax

```
ver
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--------------------------------------|
| /? | Displays help at the command prompt. |

Examples

To obtain the version number of the operating system from the Command shell (cmd.exe), type:

```
ver
```

The ver command does not work in PowerShell. To obtain the OS version from PowerShell, type:

```
$PSVersionTable.BuildVersion
```

Additional references

[Command-Line Syntax Key](#)

verifier

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Driver verifier manager.

Syntax

```
verifier /standard /driver <name> [<name> ...]
verifier /standard /all
verifier [/flags <flags>] [/faults [<probability> [<tags> [<applications> [<minutes>]]]]] /driver <name>
[<name>...]
verifier [/flags FLAGS] [/faults [<probability> [<tags> [<applications> [<minutes>]]]]] /all
verifier /querysettings
verifier /volatile /flags <flags>
verifier /volatile /adddriver <name> [<name>...]
verifier /volatile /removedriver <name> [<name>...]
verifier /volatile /faults [<probability> [<tags> [<applications>]]
verifier /reset
verifier /query
verifier /log <LogFileName> [/interval <seconds>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| <flags> | <p>Must be a number in decimal or hexadecimal, combination of bits:</p> <ul style="list-style-type: none">- Value: description- bit 0: special pool checking- bit 1: force irql checking- bit 2: low resources simulation- bit 3: pool tracking- bit 4: I/O verification- bit 5: deadlock detection- bit 6: unused- bit 7: DMA verification- bit 8: security checks- bit 9: force pending I/O requests- bit 10: IRP logging- bit 11: miscellaneous checks <p>for example, /flags 27 is equivalent with /flags 0x1B</p> |
| /volatile | <p>Used to change the verifier settings dynamically without restarting the system. Any new settings will be lost when the system is restarted.</p> |

| PARAMETER | DESCRIPTION |
|----------------|---|
| <probability> | <p>Number between 1 and 10,000 specifying the fault injection probability. For example, specifying 100 means a fault injection probability of 1% (100/10,000).</p> <p>if this parameter is not specified then the default probability of 6% will be used.</p> |
| <tags> | <p>Specifies the pool tags that will be injected with faults, separated by space characters. If this parameter is not specified then any pool allocation can be injected with faults.</p> |
| <applications> | <p>Specifies the image file name of the applications that will be injected with faults, separated by space characters. If this parameter is not specified then low resources simulation can take place in any application.</p> |
| <minutes> | <p>A positive number specifying the length of the period after rebooting, in minutes, during which no fault injection will occur. If this parameter is not specified then the default length of 8 minutes will be used.</p> |
| /? | <p>Displays help at the command prompt.</p> |

Additional references

- [Command-Line Syntax Key](#)

verify

7/11/2018 • 2 minutes to read • [Edit Online](#)

Tells **cmd** whether to verify that your files are written correctly to a disk. If used without parameters, **verify** displays the current setting.

For examples of how to use this command, see [Examples](#).

Syntax

```
verify [on | off]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------|---|
| [on off] | Switches the verify setting on or off. |
| /? | Displays help at the command prompt. |

Examples

To display the current **verify** setting, type:

```
verify
```

To turn the **verify** setting on, type:

```
Verify on
```

Additional references

[Command-Line Syntax Key](#)

vol

4/13/2018 • 2 minutes to read • [Edit Online](#)

Displays the disk volume label and serial number, if they exist. If used without parameters, **vol** displays information for the current drive.

Syntax

```
vol [<Drive>:]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| <Drive>: | Specifies the drive that contains the disk for which you want to display the volume label and serial number. |
| /? | Displays help at the command prompt. |

Additional references

[Command-Line Syntax Key](#)

waitfor

4/13/2018 • 2 minutes to read • [Edit Online](#)

Sends or waits for a signal on a system. **Waitfor** is used to synchronize computers across a network.

For examples of how to use this command, see [Examples](#).

Syntax

```
waitfor [/s <Computer> [/u [<Domain>\]<User> [/p [<Password>]]]] /si <SignalName>
waitfor [/t <Timeout>] <SignalName>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------|--|
| /s <Computer> | Specifies the name or IP address of a remote computer (do not use backslashes). The default is the local computer. This parameter applies to all files and folders specified in the command. |
| /u [<Domain>] | Runs the script using the credentials of the specified user account. By default, waitfor uses the current user's credentials. |
| /p [<Password>] | Specifies the password of the user account that is specified in the /u parameter. |
| /si | Sends the specified signal across the network. |
| /t <Timeout> | Specifies the number of seconds to wait for a signal. By default, waitfor waits indefinitely. |
| <SignalName> | Specifies the signal that waitfor waits for or sends. <i>SignalName</i> is not case-sensitive. |
| /? | Displays help at the command prompt. |

Remarks

- Signal names cannot exceed 225 characters. Valid characters include a-z, A-Z, 0-9, and the ASCII extended character set (128-255).
- If you do not use **/s**, the signal is broadcast to all the systems in a domain. If you use **/s**, the signal is sent only to the specified system.
- You can run multiple instances of **waitfor** on a single computer, but each instance of **waitfor** must wait for a different signal. Only one instance of **waitfor** can wait for a given signal on a given computer.
- You can activate a signal manually by using the **/si** command-line option.
- Waitfor** runs only on Windows XP and servers running a Windows Server 2003 operating system, but it can send signals to any computer running a Windows operating system.

- Computers can only receive signals if they are in the same domain as the computer sending the signal.
- You can use **waitfor** when you test software builds. For example, the compiling computer can send a signal to several computers running **waitfor** after the compile has completed successfully. On receipt of the signal, the batch file that includes **waitfor** can instruct the computers to immediately start installing software or running tests on the compiled build.

Examples

To wait until the "espresso\build007" signal is received, type:

```
waitfor espresso\build007
```

By default, **waitfor** waits indefinitely for a signal.

To wait 10 seconds for the "espresso\compile007" signal to be received before timing out, type:

```
waitfor /t 10 espresso\build007
```

To manually activate the "espresso\build007" signal, type:

```
waitfor /si espresso\build007
```

Additional references

[Command-Line Syntax Key](#)

wbadmin

4/13/2018 • 2 minutes to read • [Edit Online](#)

Enables you to back up and restore your operating system, volumes, files, folders, and applications from a command prompt.

To configure a regularly scheduled backup, you must be a member of the **Administrators** group. To perform all other tasks with this command, you must be a member of the **Backup Operators** or the **Administrators** group, or you must have been delegated the appropriate permissions.

You must run **wbadmin** from an elevated command prompt. (To open an elevated command prompt, right-click **Command Prompt**, and then click **Run as administrator**.)

Subcommands

| SUBCOMMAND | DESCRIPTION |
|---|---|
| Wbadmin enable backup | Configures and enables a regularly scheduled backup. |
| Wbadmin disable backup | Disables your daily backups. |
| Wbadmin start backup | Runs a one-time backup. If used with no parameters, uses the settings from the daily backup schedule. |
| Wbadmin stop job | Stops the currently running backup or recovery operation. |
| Wbadmin get versions | Lists details of backups recoverable from the local computer or, if another location is specified, from another computer. |
| Wbadmin get items | Lists the items included in a backup. |
| Wbadmin start recovery | Runs a recovery of the volumes, applications, files, or folders specified. |
| Wbadmin get status | Shows the status of the currently running backup or recovery operation. |
| Wbadmin get disks | Lists disks that are currently online. |
| Wbadmin start systemstaterecovery | Runs a system state recovery. |
| Wbadmin start systemstatebackup | Runs a system state backup. |
| Wbadmin delete systemstatebackup | Deletes one or more system state backups. |
| Wbadmin start sysrecovery | Runs a recovery of the full system (at least all the volumes that contain the operating system's state). This subcommand is only available if you are using the Windows Recovery Environment. |

| SUBCOMMAND | DESCRIPTION |
|---|--|
| Wbadmin restore catalog | Recovers a backup catalog from a specified storage location in the case where the backup catalog on the local computer has been corrupted. |
| Wbadmin delete catalog | Deletes the backup catalog on the local computer. Use this subcommand only if the backup catalog on this computer is corrupted and you have no backups stored at another location that you can use to restore the catalog. |

Additional references

- [Backup and Recovery](#)
- [Windows Server Backup Cmdlets in Windows PowerShell](#)

wbadmin enable backup

4/13/2018 • 7 minutes to read • [Edit Online](#)

Creates and enables a daily backup schedule or modifies an existing backup schedule. With no parameters specified, it displays the currently scheduled backup settings.

To configure or modify a daily backup schedule, you must be a member of either the **Administrators** or **Backup Operators** group. In addition, you must run **wbadmin** from an elevated command prompt. (To open an elevated command prompt right-click **Command Prompt** and then click **Run as administrator**.)

For examples of how to use this subcommand, see [Examples](#).

Syntax

Syntax for Windows Server 2008:

```
wbadmin enable backup  
[-addtarget:<BackupTargetDisk>]  
[-removetarget:<BackupTargetDisk>]  
[-schedule:<TimeToRunBackup>]  
[-include:<VolumesToInclude>]  
[-allCritical]  
[-quiet]
```

Syntax for Windows Server 2008 R2:

```
wbadmin enable backup  
[-addtarget:<BackupTarget>]  
[-removetarget:<BackupTarget>]  
[-schedule:<TimeToRunBackup>]  
[-include:<VolumesToInclude>]  
[-nonRecurseInclude:<ItemsToInclude>]  
[-exclude:<ItemsToExclude>]  
[-nonRecurseExclude:<ItemsToExclude>][-systemState]  
[-allCritical]  
[-vssFull | -vssCopy]  
[-user:<UserName>]  
[-password:<Password>]  
[-quiet]
```

Syntax for Windows Server 2012 and Windows Server 2012 R2:

```
wbadmin enable backup
[-addtarget:<BackupTarget>]
[-removetarget:<BackupTarget>]
[-schedule:<TimeToRunBackup>]
[-include:<VolumesToInclude>]
[-nonRecurseInclude:<ItemsToInclude>]
[-exclude:<ItemsToExclude>]
[-nonRecurseExclude:<ItemsToExclude>][ -systemState]
[-hyperv:<HyperVComponentsToExclude>]
[-allCritical]
[-systemState]
[-vssFull | -vssCopy]
[-user:<UserName>]
[-password:<Password>]
[-quiet]
[-allowDeleteOldBackups]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------|---|
| -addtarget | <p>For Windows Server 2008, specifies the storage location for backups. Requires you to specify a destination for backups as a disk identifier (see Remarks). The disk is formatted before use, and any existing data on it is permanently erased.</p> <p>For Windows Server 2008 R2 and later, Specifies the storage location for backups. Requires you to specify the location as a disk, volume, or Universal Naming Convention (UNC) path to a remote shared folder (\\<servername> <sharename>). By default, the backup will be saved at:</p> <p>\\<sharename>\WindowsImageBackup<ComputerBackedUp>. If you specify a disk, the disk will be formatted before use, and any existing data on it is permanently erased. If you specify a shared folder, you cannot add more locations. You can only specify one shared folder as a storage location at a time.</p> <p>Important: If you save a backup to a remote shared folder, that backup will be overwritten if you use the same folder to back up the same computer again. In addition, if the backup operation fails, you may end up with no backup because the older backup will be overwritten, but the newer backup will not be usable. You can avoid this by creating subfolders in the remote shared folder to organize your backups. If you do this, the subfolders will need twice the space of the parent folder. Only one location can be specified in a single command. Multiple volume and disk backup storage locations can be added by running the command again.</p> |
| -removetarget | <p>Specifies the storage location that you want to remove from the existing backup schedule. Requires you to specify the location as a disk identifier (see Remarks).</p> |
| -schedule | <p>Specifies times of day to create a backup, formatted as HH:MM and comma delimited.</p> |

| PARAMETER | DESCRIPTION |
|--------------------|--|
| -include | <p>For Windows Server 2008, specifies the comma-delimited list of volume drive letters, volume mount points, or GUID-based volume names to include in the backup.</p> <p>For Windows Server 2008 R2 and later, Specifies the comma-delimited list of items to include in the backup. You can include multiple files, folders, or volumes. Volume paths can be specified using volume drive letters, volume mount points, or GUID-based volume names. If you use a GUID-based volume name, it should be terminated with a backslash (). You can use the wildcard character (*) in the file name when specifying a path to a file.</p> |
| -nonRecurseInclude | <p>For Windows Server 2008 R2 and later, specifies the non-recursive, comma-delimited list of items to include in the backup. You can include multiple files, folders, or volumes. Volume paths can be specified using volume drive letters, volume mount points, or GUID-based volume names. If you use a GUID-based volume name, it should be terminated with a backslash (). You can use the wildcard character (*) in the file name when specifying a path to a file. Should be used only when the -backupTarget parameter is used.</p> |
| -exclude | <p>For Windows Server 2008 R2 and later, specifies the comma-delimited list of items to exclude from the backup. You can exclude files, folders, or volumes. Volume paths can be specified using volume drive letters, volume mount points, or GUID-based volume names. If you use a GUID-based volume name, it should be terminated with a backslash (). You can use the wildcard character (*) in the file name when specifying a path to a file.</p> |
| -nonRecurseExclude | <p>For Windows Server 2008 R2 and later, specifies the non-recursive, comma-delimited list of items to exclude from the backup. You can exclude files, folders, or volumes. Volume paths can be specified using volume drive letters, volume mount points, or GUID-based volume names. If you use a GUID-based volume name, it should be terminated with a backslash (). You can use the wildcard character (*) in the file name when specifying a path to a file.</p> |
| -hyperv | <p>Specifies the comma-delimited list of components to be included in backup. The identifier could be a component name or component GUID (with or without braces).</p> |
| -systemState | <p>For Windows®7 and Windows Server 2008 R2 and later, creates a backup that includes the system state in addition to any other items that you specified with the -include parameter. The system state contains boot files (Boot.ini, NTLDR, NTDetect.com), the Windows Registry including COM settings, the SYSVOL (Group Policies and Logon Scripts), the Active Directory and NTDS.DIT on domain controllers and, if the certificates service is installed, the Certificate Store. If your server has the Web server role installed, the IIS Metadirectory will be included. If the server is part of a cluster, Cluster service information is also included.</p> |

| PARAMETER | DESCRIPTION |
|------------------------|--|
| -allCritical | <p>Specifies that all critical volumes (volumes that contain operating system's state) be included in the backups. This parameter is useful if you are creating a backup for full system or system state recovery. It should be used only when -backupTarget is specified, otherwise the command will fail. Can be used with the -include option.</p> <p>Tip: The target volume for a critical-volume backup can be a local drive, but it cannot be any of the volumes that are included in the backup.</p> |
| -vssFull | <p>For Windows Server 2008 R2 and later, Performs a full backup using the Volume Shadow Copy Service (VSS). All files are backed up, each file's history is updated to reflect that it was backed up, and the logs of previous backups may be truncated. If this parameter is not used wadmin start backup makes a copy backup, but the history of files being backed up is not updated.</p> <p>Caution: Do not use this parameter if you are using a product other than Windows Server Backup to back up applications that are on the volumes included in the current backup. Doing so can potentially break the incremental, differential, or other type of backups that the other backup product is creating because the history that they are relying on to determine how much data to backup might be missing and they might perform a full backup unnecessarily.</p> |
| -vssCopy | <p>For Windows Server 2008 R2 and later, performs a copy backup using VSS. All files are backed up but the history of the files being backup up is not updated so you preserve the all the information on which files where changed, deleted, and so on, as well as any application log files. Using this type of backup does not affect the sequence of incremental and differential backups that might happen independent of this copy backup. This is the default value.</p> <p>Warning: A copy backup cannot be used for incremental or differential backups or restores.</p> |
| -user | <p>For Windows Server 2008 R2 and later, specifies the user with write permission to the backup storage destination (if it is a remote shared folder). The user needs to be a member of the Administrators group or Backup Operators group on the computer that is getting backed up.</p> |
| -password | <p>For Windows Server 2008 R2 and later, specifies the password for the user name provided by the parameter -user.</p> |
| -quiet | <p>Runs the subcommand with no prompts to the user.</p> |
| -allowDeleteOldBackups | <p>Overwrites any backups made before the computer was upgraded.</p> |

Remarks

To view the disk identifier value for your disks, type **wadmin get disks**.

Examples

The following examples show how the **wbadmin enable backup** command can be used in different backup scenarios:

Scenario #1

- Schedule backups of hard disk drives e:, d:\mountpoint, and \\?\Volume{cc566d14-44a0-11d9-9d93-806e6f6e6963}\
- Save the files to the disk DiskID
- Run backups daily at 9:00 A.M. and 6:00 P.M.

```
wbadmin enable backup -addtarget:DiskID -schedule:09:00,18:00 -include:e:,d:\mountpoint,\\?\Volume{cc566d14-44a0-11d9-9d93-806e6f6e6963}\
```

Scenario #2

- Schedule backups of the folder d:\documents to the network location \\backupshare\backup1
- Use the network credentials for the backup administrator Aaren Ekelund (aekel), who is a member of the domain CONTOSOEASt to authenticate access to the network share. Aaren's password is \$3hM9^5lp.
- Run backups daily at 12:00 A.M. and 7:00 P.M.

```
wbadmin enable backup -addtarget:\\backupshare\backup1 -include: d:\documents -user:CONTOSOEASt\aeke1 -password:$3hM9^5lp -schedule:00:00,19:00
```

Scenario #3

- Schedule backups of volume t: and folder d:\documents to the drive h:, but exclude the folder d:\documents~tmp
- Perform a full backup using the Volume Shadow Copy Service.
- Run backups daily at 1:00 A.M.

```
wbadmin enable backup -addtarget:H: -include T:,D:\documents -exclude D:\documents\~tmp -vssfull -schedule:01:00
```

Additional references

- [Command-Line Syntax Key](#)
- [Wbadmin](#)

wbadmin disable backup

4/13/2018 • 2 minutes to read • [Edit Online](#)

Stops running the existing scheduled daily backups.

To disable a scheduled daily backup, you must be a member of the **Administrators** group, or you must have been delegated the appropriate permissions. In addition, you must run **wbadmin** from an elevated command prompt. (To open an elevated command prompt right-click **Command Prompt** and then click **Run as administrator**.)

Syntax

```
wbadmin disable backup  
[-quiet]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| -quiet | Runs the subcommand with no prompts to the user. |

Additional references

- [Command-Line Syntax Key](#)
- [Wbadmin](#)

wbadmin start backup

9/28/2018 • 7 minutes to read • [Edit Online](#)

Creates a backup using specified parameters. If no parameters are specified and you have created a scheduled daily backup, this subcommand creates the backup by using the settings for the scheduled backup. If parameters are specified, it creates a Volume Shadow Copy Service (VSS) copy backup and will not update the history of the files that are being backed up.

To create a one-time backup with this subcommand, you must be a member of the **Backup Operators** group or the **Administrators** group, or you must have been delegated the appropriate permissions. In addition, you must run **wbadmin** from an elevated command prompt. (To open an elevated command prompt right-click **Command Prompt** and then click **Run as administrator**.)

For examples of how to use this subcommand, see [Examples](#).

Syntax

Syntax for Windows®Vista and Windows Server 2008:

```
wbadmin start backup
[-backupTarget:{<BackupTargetLocation> | <TargetNetworkShare>}]
[-include:<VolumesToInclude>]
[-allCritical]
[-noVerify]
[-user:<UserName>]
[-password:<Password>]
[-noinheritAcl]
[-vssFull]
[-quiet]
```

Syntax for Windows®7 and Windows Server 2008 R2 and later:

```
Wbadmin start backup
[-backupTarget:{<BackupTargetLocation> | <TargetNetworkShare>}]
[-include:<ItemsToInclude>]
[-nonRecurseInclude:<ItemsToInclude>]
[-exclude:<ItemsToExclude>]
[-nonRecurseExclude:<ItemsToExclude>]
[-allCritical]
[-systemState]
[-noVerify]
[-user:<UserName>]
[-password:<Password>]
[-noInheritAcl]
[-vssFull | -vssCopy]
[-quiet]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|-------------|
|-----------|-------------|

| PARAMETER | DESCRIPTION |
|--------------------|--|
| -backupTarget | <p>Specifies the storage location for this backup. Requires a hard disk drive letter (f.), a volume GUID-based path in the format of \\?\Volume{GUID}, or a Universal Naming Convention (UNC) path to a remote shared folder (\\<servername><sharename>). By default, the backup will be saved at: \\<sharename>*WindowsImageBackup*\. Important: If you save a backup to a remote shared folder, that backup will be overwritten if you use the same folder to back up the same computer again. In addition, if the backup operation fails, you may end up with no backup because the older backup will be overwritten, but the newer backup will not be usable. You can avoid this by creating subfolders in the remote shared folder to organize your backups. If you do this, the subfolders will need twice the space as the parent folder.</p> |
| -include | <p>For Windows®Vista and Windows Server 2008, specifies the comma-delimited list of volume drive letters, volume mount points, or GUID-based volume names to include in the backup. This parameter should be used only when the -backupTarget parameter is used.</p> <p>For Windows®7 and Windows Server 2008 R2 and later, specifies the comma-delimited list of items to include in the backup. You can include multiple files, folders, or volumes. Volume paths can be specified using volume drive letters, volume mount points, or GUID-based volume names. If you use a GUID-based volume name, it should be terminated with a backslash (\). You can use the wildcard character (*) in the file name when specifying a path to a file. Should be used only when the -backupTarget parameter is used.</p> |
| -exclude | <p>For Windows®7 and Windows Server 2008 R2 and later, specifies the comma-delimited list of items to exclude from the backup. You can exclude files, folders, or volumes. Volume paths can be specified using volume drive letters, volume mount points, or GUID-based volume names. If you use a GUID-based volume name, it should be terminated with a backslash (\). You can use the wildcard character (*) in the file name when specifying a path to a file. Should be used only when the -backupTarget parameter is used.</p> |
| -nonRecurseInclude | <p>For Windows®7 and Windows Server 2008 R2 and later, specifies the non-recursive, comma-delimited list of items to include in the backup. You can include multiple files, folders, or volumes. Volume paths can be specified using volume drive letters, volume mount points, or GUID-based volume names. If you use a GUID-based volume name, it should be terminated with a backslash (\). You can use the wildcard character (*) in the file name when specifying a path to a file. Should be used only when the -backupTarget parameter is used.</p> |

| PARAMETER | DESCRIPTION |
|--------------------|--|
| -nonRecurseExclude | For Windows®7 and Windows Server 2008 R2 and later, specifies the non-recursive, comma-delimited list of items to exclude from the backup. You can exclude files, folders, or volumes. Volume paths can be specified using volume drive letters, volume mount points, or GUID-based volume names. If you use a GUID-based volume name, it should be terminated with a backslash (\). You can use the wildcard character (*) in the file name when specifying a path to a file. Should be used only when the -backupTarget parameter is used. |
| -allCritical | Specifies that all critical volumes (volumes that contain operating system's state) be included in the backups. This parameter is useful if you are creating a backup for bare metal recovery. It should be used only when -backupTarget is specified, otherwise the command will fail. Can be used with the -include option. Tip: The target volume for a critical-volume backup can be a local drive, but it cannot be any of the volumes that are included in the backup. |
| -systemState | For Windows®7 and Windows Server 2008 R2 and later, creates a backup that includes the system state in addition to any other items that you specified with the -include parameter. The system state contains boot files (Boot.ini, NTLDR, NTDetect.com), the Windows Registry including COM settings, the SYSVOL (Group Policies and Logon Scripts), the Active Directory and NTDS.DIT on Domain Controllers and, if the certificates service is installed, the Certificate Store. If your server has the Web server role installed, the IIS Metadirectory will be included. If the server is part of a cluster, Cluster Service information will also be included. |
| -noVerify | Specifies that backups saved to removable media (such as a DVD) are not verified for errors. If you do not use this parameter, backups saved to removable media are verified for errors. |
| -user | If the backup is saved to a remote shared folder, specifies the user name with write permission to the folder. |
| -password | Specifies the password for the user name that is provided by the parameter -user . |
| -noInheritAcl | Applies the access control list (ACL) permissions that correspond to the credentials provided by the -user and -password parameters to \\<servername><sharename>\WindowsImageBackup<ComputerBackedUp>\ (the folder that contains the backup). To access the backup later, you must use these credentials or be a member of the Administrators group or the Backup Operators group on the computer with the shared folder. If -noInheritAcl is not used, the ACL permissions from the remote shared folder are applied to the <ComputerBackedUp> folder by default so that anyone with access to the remote shared folder can access the backup. |

| PARAMETER | DESCRIPTION |
|-----------|---|
| -vssFull | <p>Performs a full back up using the Volume Shadow Copy Service (VSS). All files are backed up, each file's history is updated to reflect that it was backed up, and the logs of previous backups may be truncated. If this parameter is not used wbadmin start backup makes a copy backup, but the history of files being backed up is not updated.</p> <p>Caution: Do not use this parameter if you are using a product other than Windows Server Backup to back up applications that are on the volumes included in the current backup. Doing so can potentially break the incremental, differential, or other type of backups that the other backup product is creating because the history that they are relying on to determine how much data to backup might be missing and they might perform a full backup unnecessarily.</p> |
| -vssCopy | <p>For Windows 7 and Windows Server 2008 R2 and later, performs a copy backup using VSS. All files are backed up but the history of the files being backup up is not updated so you preserve the all the information on which files where changed, deleted, and so on, as well as any application log files. Using this type of backup does not affect the sequence of incremental and differential backups that might happen independent of this copy backup. This is the default value.</p> <p>Warning: A copy backup cannot be used for incremental or differential backups or restores.</p> |
| -quiet | Runs the subcommand with no prompts to the user. |

Examples

The following examples show how the **wbadmin start backup** command can be used in different backup scenarios:

Scenario #1

- Create a backup of volumes e:, d:\mountpoint, and \\?\Volume{cc566d14-4410-11d9-9d93-806e6f6e6963}
- Save the backup to volume f:

```
wbadmin start backup -backupTarget:f: -include:e:,d:\mountpoint,\\?\Volume{cc566d14-44a0-11d9-9d93-806e6f6e6963}\
```

Scenario #2

- Perform a one-time backup of *f:\folder1* and *h:\folder2* to volume *d:*.
- Backup the system state
- Make a copy backup so that the normally scheduled differential backup is not impacted.

```
wbadmin start backup -backupTarget:d: -include:g\folder1,h:\folder2 -systemstate -vsscCopy
```

Scenario #3

- Perform a one-time backup of *d:\folder1* that should be backed up non-recursively.
- Backup the folder to the network location \\backupshare\backup1
- Restrict access to the backup to members of the **Administrators** or **Backup Operators** group.

```
wbadmin start backup -backupTarget: \\backupshare\backup1 -noinheritacl -nonrecurseinclude:d:\folder1
```

Additional references

- [Command-Line Syntax Key](#)
- [Wbadmin](#)

wbadmin stop job

4/13/2018 • 2 minutes to read • [Edit Online](#)

Cancels the backup or recovery operation that is currently running. Canceled operations cannot be restarted—you must rerun a canceled backup or recovery operation from the beginning.

To stop a backup or recovery operation with this subcommand, you must be a member of the **Backup Operators** group or the **Administrators** group, or you must have been delegated the appropriate authority. In addition, you must run **wbadmin** from an elevated command prompt. (To open an elevated command prompt right-click **Command Prompt** and then click **Run as administrator**.)

Syntax

```
wbadmin stop job  
[-quiet]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| -quiet | Runs the subcommand with no prompts to the user. |

Additional references

- [Command-Line Syntax Key](#)
- [Wbadmin](#)

wbadmin get versions

4/13/2018 • 2 minutes to read • [Edit Online](#)

Lists details about the available backups that are stored on the local computer or another computer. When this subcommand is used without parameters, it lists all backups of the local computer, even if those backups are not available. The details provided for a backup include the backup time, the backup storage location, the version identifier (needed for the **wbadmin get items** subcommand and to perform recoveries), and the type of recoveries you can perform.

To get details about available backups using this subcommand, you must be a member of the **Backup Operators** group or the **Administrators** group, or you must have been delegated the appropriate permissions. In addition, you must run **wbadmin** from an elevated command prompt. (To open an elevated command prompt **Command Prompt** and then click **Run as administrator**.)

For examples of how to use this subcommand, see [Examples](#).

Syntax

```
wbadmin get versions  
[-backupTarget:{<BackupTargetLocation> | <NetworkSharePath>}]  
[-machine:BackupMachineName]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------|---|
| -backupTarget | Specifies the storage location that contains the backups that you want the details for. Use for listing backups stored at that target location. Backup target locations can be locally attached disk drives, volumes, remote shared folders, removable media such as DVD drives or other optical media. If wbadmin get versions is run on the same computer where the backup was created, this parameter is not needed. However, this parameter is required to get information about a backup created from another computer. |
| -machine | Specifies the computer that you want backup details for. Use when backups of multiple computers are stored in the same location. Should be used when -backupTarget is specified. |

Remarks

To list items available for recovery from a specific backup, use **wbadmin get items**.

Examples

To see a list of available backups that are stored on volume h, type:

```
wbadmin get versions -backupTarget:h:
```

To see a list of available backups that are stored in the remote shared folder \\servername\share for the computer server01, type:

```
wbadmin get versions -backupTarget:\\servername\share -machine:server01
```

Additional references

- [Command-Line Syntax Key](#)
- [Wbadmin](#)
- [Get-WBBackupTarget](#) cmdlet

wbadmin get items

4/13/2018 • 2 minutes to read • [Edit Online](#)

Lists the items included in a specific backup.

To use this subcommand, you must be a member of the **Backup Operators** group or the **Administrators** group, or you must have been delegated the appropriate permissions. In addition, you must run **wbadmin** from an elevated command prompt. (To open an elevated command prompt right-click **Command Prompt** and then click **Run as administrator**.)

For examples of how to use this subcommand, see [Examples](#).

Syntax

```
wbadmin get items
-version:<VersionIdentifier>
[-backupTarget:{<BackupDestinationVolume> | <NetworkSharePath>}]
[-machine:<BackupMachineName>]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------|---|
| -version | Specifies the version of the backup in MM/DD/YYYY-HH:MM format. If you do not know the version information, type wbadmin get versions . |
| -backupTarget | Specifies the storage location that contains the backups for which you want the details. Use for listing backups stored at that target location. Backup target locations can be a locally attached disk drive or a remote shared folder. If wbadmin get items is run on the same computer where the backup was created, this parameter is not needed. However, this parameter is required to get information about a backup created from another computer. |
| -machine | Specifies the name of the computer that you want the backup details for. Useful when multiple computers have been backed up to the same location. Should be used when - backupTarget is specified. |

Examples

To list items from the backup that was run on March 31, 2013 at 9:00 A.M., type:

```
wbadmin get items -version:03/31/2013-09:00
```

To list items from the backup of server01 that was run on April 30, 2013 at 9:00 A.M. and stored on \\servername\share, type:

```
wbadmin get items -version:04/30/2013-09:00 -backupTarget:\\servername\share -machine:server01
```

Additional references

- [Command-Line Syntax Key](#)
- [Wbadmin](#)
- [Get-WBBackupSet](#) cmdlet

wbadmin start recovery

4/13/2018 • 5 minutes to read • [Edit Online](#)

Runs a recovery operation based on the parameters that you specify.

To perform a recovery with this subcommand, you must be a member of the **Backup Operators** group or the **Administrators** group, or you must have been delegated the appropriate permissions. In addition, you must run **wbadmin** from an elevated command prompt. (To open an elevated command prompt, click **Start**, right-click **Command Prompt**, and then click **Run as administrator**.)

For examples of how to use this subcommand, see [Examples](#).

Syntax

```
wbadmin start recovery
-version:<VersionIdentifier>
-items:{<VolumesToRecover> | <AppsToRecover> | <FilesOrFoldersToRecover>}
-itemtype:{Volume | App | File}
[-backupTarget:{<VolumeHostingBackup> | <NetworkShareHostingBackup>}]
[-machine:<BackupMachineName>]
[-recoveryTarget:{<TargetVolumeForRecovery> | <TargetPathForRecovery>}]
[-recursive]
[-overwrite:{Overwrite | CreateCopy | Skip}]
[-notRestoreAcl]
[-skipBadClusterCheck]
[-noRollForward]
[-quiet]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|---|
| -version | Specifies the version identifier of the backup to recover in MM/DD/YYYY-HH:MM format. If you do not know the version identifier, type wbadmin get versions . |
| -items | Specifies a comma-delimited list of volumes, applications, files, or folders to recover. <ul style="list-style-type: none">- If -itemtype is Volume, you can specify only a single volume—by providing the volume drive letter, volume mount point, or GUID-based volume name.- If -itemtype is App, you can specify only a single application. To be recovered, the application must have registered with Windows Server Backup. You can also use the value ADIFM to recover an installation of Active Directory. See Remarks in for more information.- If -itemtype is File, you can specify files or folders, but they should be part of the same volume and they should be under the same parent folder. |
| -itemtype | Specifies type of items to recover. Must be Volume , App , or File . |

| PARAMETER | DESCRIPTION |
|----------------------|---|
| -backupTarget | Specifies the storage location that contains the backup that you want to recover. This parameter is useful when the location is different from where backups of this computer are usually stored. |
| -machine | Specifies the name of the computer that you want to recover the backup for. This parameter is useful when multiple computers have been backed up to the same location. It should be used when the -backupTarget parameter is specified. |
| -recoveryTarget | Specifies the location to restore to. This parameter is useful if this location is different than the location that was previously backed up. It can also be used for restorations of volumes, files, or applications. If you are restoring a volume, you can specify the volume drive letter of the alternate volume. If you are restoring a file or application, you can specify an alternate recovery location. |
| -recursive | Valid only when recovering files. Recovers the files in the folders and all files subordinate to the specified folders. By default, only files which reside directly in the specified folders are recovered. |
| -overwrite | Valid only when recovering files. Specifies the action to take when a file that is being recovered already exists in the same location. <ul style="list-style-type: none"> - Skip causes Windows Server Backup to skip the existing file and continue with recovery of the next file. - CreateCopy causes Windows Server Backup to create a copy of the existing file so that the existing file is not modified. - Overwrite causes Windows Server Backup to overwrite the existing file with the file from the backup. |
| -notRestoreAcl | Valid only when recovering files. Specifies to not restore the security access control lists (ACLs) of the files being recovered from the backup. By default, the security ACLs are restored (the default value is true). If this parameter is used, the ACLs for the restored files will be inherited from the location to which the files are being restored. |
| -skipBadClusterCheck | Valid only when recovering volumes. Skips checking the disks that you are recovering to for bad cluster information. If you are recovering to an alternate server or hardware, we recommend that you do not use this parameter. You can manually run the command chkdsk /b on these disks at any time to check them for bad clusters, and then update the file system information accordingly. Important: Until you run chkdsk as described, the bad clusters reported on your recovered system may not be accurate. |
| -noRollForward | Valid only when recovering applications. Allows for previous point-in-time recovery of an application if the latest version from the backups is selected. For other versions of the application that are not the latest, previous point-in-time recovery is done as the default. |

| PARAMETER | DESCRIPTION |
|-----------|--|
| -quiet | Runs the subcommand with no prompts to the user. |

Remarks

- To view a list of items that are available for recovery from a specific backup version, use **wbadmin get items**. If a volume did not have a mount point or drive letter at the time of backup, then this subcommand would return a GUID-based volume name that should be used for recovering the volume.
- When the **-itemtype** is **App**, you can use a value of **ADIFM** for **-item** to perform an install from media operation to recover all the related data needed for Active Directory Domain Services. **ADIFM** creates a copy of the Active Directory database, registry, and SYSVOL state, and then saves this information in the location specified by **-recoveryTarget**. Use this parameter only when **-recoveryTarget** is specified.

[!NOTE]

Before using ****wbadmin**** to perform an install from media operation, you should consider using the ****ntdsutil**** command because ****ntdsutil**** only copies the minimum amount of data needed, and it uses a more secure data transport method.

Examples

To run a recovery of the backup from March 31, 2013, taken at 9:00 A.M., of volume d:, type:

```
wbadmin start recovery -version:03/31/2013-09:00 -itemType:Volume -items:d:
```

To run a recovery to drive d of the backup from March 31, 2013, taken at 9:00 A.M., of the registry, type:

```
wbadmin start recovery -version:03/31/2013-09:00 -itemType:App -items:Registry -recoverytarget:d:\
```

To run a recovery of the backup from March 31, 2013, taken at 9:00 A.M., of the d:\folder and folders subordinate to d:\folder, type:

```
wbadmin start recovery -version:03/31/2013-09:00 -itemType:File -items:d:\folder -recursive
```

To run a recovery of the backup from March 31, 2013, taken at 9:00 A.M., of the volume \\?\Volume{cc566d14-44a0-11d9-9d93-806e6f6e6963}, type:

```
wbadmin start recovery -version:03/31/2013-09:00 -itemType:Volume
-items:\\?\Volume{cc566d14-44a0-11d9-9d93-806e6f6e6963}\
```

To run a recovery of the backup from April 30, 2013, taken at 9:00 A.M., of the shared folder \\servername\share from server01, type:

```
wbadmin start recovery -version:04/30/2013-09:00 -backupTarget:\\servername\share -machine:server01
```

Additional references

- [Command-Line Syntax Key](#)
- [Wbadmin](#)
- [Start-WBFileRecovery cmdlet](#)

- [Start-WBHyperVRecovery](#) cmdlet
- [Start-WBSystemStateRecovery](#) cmdlet
- [Start-WBVolumeRecovery](#) cmdlet

wbadmin get status

4/13/2018 • 2 minutes to read • [Edit Online](#)

Reports the status of the backup or recovery operation that is currently running.

To use this subcommand, you must be a member of the **Backup Operators** group or the **Administrators** group, or you must have been delegated the appropriate permissions. In addition, you must run **wbadmin** from an elevated command prompt. (To open an elevated command prompt right-click **Command Prompt**, and then click **Run as administrator**.)

Syntax

```
wbadmin get status
```

Parameters

This subcommand has no parameters.

Remarks

- This subcommand will not stop until the current backup or recovery operation is finished—the subcommand will continue to run even if you close the command window.
- If you want to stop the current backup or recovery operation, use the **wbadmin stop job** subcommand.

Additional references

- [Command-Line Syntax Key](#)
- [Wbadmin](#)
- [Get-WBJob](#) cmdlet

wbadmin get disks

4/13/2018 • 2 minutes to read • [Edit Online](#)

Lists the internal and external disks that are currently online for the local computer.

To list the disks that are online with this subcommand, you must be a member of the **Backup Operators** group or the **Administrators** group, or you must have been delegated the appropriate permissions. In addition, you must run **wbadmin** from an elevated command prompt. (To open an elevated command prompt right-click **Command Prompt**, and then click **Run as administrator**.)

Syntax

```
wbadmin get disks
```

Parameters

This subcommand has no parameters.

Additional references

- [Command-Line Syntax Key](#)
- [Wbadmin](#)
- [Get-WBDisk](#) cmdlet

wbadmin start systemstaterecovery

4/13/2018 • 2 minutes to read • [Edit Online](#)

Performs a system state recovery to a location and from a backup that you specify.

NOTE

Windows Server Backup does not back up or recover registry user hives (HKEY_CURRENT_USER) as part of system state backup or system state recovery.

To perform a system state recovery with this subcommand, you must be a member of the **Backup Operators** group or the **Administrators** group, or you must have been delegated the appropriate permissions. In addition, you must run **wbadmin** from an elevated command prompt. (To open an elevated command prompt right-click **Command Prompt**, and then click **Run as administrator**.)

For examples of how to use this command, see [Examples](#).

Syntax

Syntax for Windows Server 2008:

```
wbadmin start systemstaterecovery
-version:<VersionIdentifier>
-showsummary
[-backupTarget:{<BackupDestinationVolume> | <NetworkSharePath>}]
[-machine:<BackupMachineName>]
[-recoveryTarget:<TargetPathForRecovery>]
[-authsysvol]
[-quiet]
```

Syntax for Windows Server 2008 R2 or later:

```
wbadmin start systemstaterecovery
-version:<VersionIdentifier>
-showsummary
[-backupTarget:{<BackupDestinationVolume> | <NetworkSharePath>}]
[-machine:<BackupMachineName>]
[-recoveryTarget:<TargetPathForRecovery>]
[-authsysvol]
[-autoReboot]
[-quiet]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| -version | Specifies the version identifier for the backup to recover in MM/DD/YYYY-HH:MM format. If you do not know the version identifier, type wbadmin get versions . |

| PARAMETER | DESCRIPTION |
|-----------------|--|
| -showsummary | Reports the summary of the last system state recovery (after the restart required to complete the operation). This parameter cannot be accompanied by any other parameters. |
| -backupTarget | Specifies the storage location that contains the backup or backups you want to recover. This parameter is useful when the storage location is different from where the backups of this computer are usually stored. |
| -machine | Specifies the name of the computer that you want to recover. This parameter is useful when multiple computers have been backed up to the same location. Should be used when the - backupTarget parameter is specified. |
| -recoveryTarget | Specifies the directory to restore to. This parameter is useful if the backup is restored to an alternate location. |
| -authsysvol | If used, performs an authoritative restore of SYSVOL (the System Volume shared directory). |
| -autoReboot | Specifies to restart the system at the end of the system state recovery operation. This parameter is valid only for a recovery to the original location. We do not recommend you use this parameter if you need to perform steps after the recovery operation. |
| -quiet | Runs the subcommand with no prompts to the user. |

Examples

- To perform a system state recovery of the backup from 03/31/2013 at 9:00 A.M., type:

```
wbadmin start systemstaterecovery -version:03/31/2013-09:00
```

- To perform a system state recovery of the backup from 04/30/2013 at 9:00 A.M. that is stored on the shared resource \\servername\share for server01, type:

```
wbadmin start systemstaterecovery -version:04/30/2013-09:00 -backupTarget:\\servername\share -  
machine:server01
```

Additional references

- [Command-Line Syntax Key](#)
- [Wbadmin](#)
- [Start-WBSystemStateRecovery cmdlet](#)

wbadmin start systemstatebackup

4/13/2018 • 2 minutes to read • [Edit Online](#)

Creates a system state backup of the local computer and stores it on the location specified.

NOTE

Windows Server Backup does not back up or recover registry user hives (HKEY_CURRENT_USER) as part of system state backup or system state recovery.

To perform a system state backup with this subcommand, you must be a member of the **Backup Operators** group or the **Administrators** group, or you must have been delegated the appropriate permissions. In addition, you must run **wbadmin** from an elevated command prompt. (To open an elevated command prompt right-click **Command Prompt**, and then click **Run as administrator**.)

For examples of how to use this subcommand, see [Examples](#).

Syntax

```
wbadmin start systemstatebackup  
-backupTarget:<VolumeName>  
[-quiet]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------|---|
| -backupTarget | Specifies the location where you want to store the backup. The storage location requires a drive letter or a GUID-based volume of the format: \\?\Volume{GUID}. A system state backup to a shared network folder is not supported on a computer running Windows Server 2008. If your server is running Windows Server 2008 R2 or later you can use the command - backupTarget: \\servername\sharedFolder\ to store system state backups. |
| -quiet | Runs the subcommand with no prompts to the user. |

Remarks

For information about saving a system state backup to a volume that, in turn, contains system state files, see article 944530 in the Microsoft Knowledge Base (<https://go.microsoft.com/fwlink/?LinkId=110439>).

Examples

To create a system state backup and store it on volume f, type:

```
wbadmin start systemstatebackup -backupTarget:f:
```

Additional references

- [Command-Line Syntax Key](#)
- [Wbadmin](#)
- [Start-WBBackup](#) cmdlet

wbadmin delete systemstatebackup

4/13/2018 • 2 minutes to read • [Edit Online](#)

Deletes the system state backups that you specify. If the specified volume contains backups other than system state backups of your local server, those backups will not be deleted.

NOTE

Windows Server Backup does not back up or recover registry user hives (HKEY_CURRENT_USER) as part of system state backup or system state recovery.

To delete a system state backup with this subcommand, you must be a member of the **Backup Operators** group or the **Administrators** group, or you must have been delegated the appropriate permissions. In addition, you must run **wbadmin** from an elevated command prompt. (To open an elevated command prompt right-click **Command Prompt**, and then click **Run as administrator**.)

For examples of how to use this command, see [Examples](#).

Syntax

```
wbadmin delete systemstatebackup
{-keepVersions:<NumberOfCopies> | -version:<VersionIdentifier> | -deleteOldest}
[-backupTarget:<VolumeName>]
[-machine:<BackupMachineName>]
[-quiet]
```

IMPORTANT

One and only one of these parameters must be specified: **-keepVersions**, **-version**, or **-deleteOldest**.

Parameters

| PARAMETER | DESCRIPTION |
|---------------|---|
| -keepVersions | Specifies the number of the latest system state backups to keep. The value must be a positive integer. The parameter value -keepVersions:0 deletes all the system state backups. |
| -version | Specifies the version identifier of the backup in MM/DD/YYYY-HH:MM format. If you do not know the version identifier, type wbadmin get versions . Versions that are exclusively system state backups can be deleted using this command. Use wbadmin get items to view the version type. |
| -deleteOldest | Deletes the oldest system state backup. |

| PARAMETER | DESCRIPTION |
|---------------|--|
| -backupTarget | Specifies the storage location for the backup that you want to delete. The storage location for backups of disks can be a drive letter, a mount point, or a GUID-based volume path. This value only needs to be specified for locating backups that are not of the local computer. Information about backups for the local computer will be available in the backup catalog on the local computer. |
| -machine | Specifies the computer whose system state backup you want to delete. Useful when multiple computers were backed up to the same location. Should be used when the -backupTarget parameter is specified. |
| -quiet | Runs the subcommand with no prompts to the user. |

Examples

To delete the system state backup created on March 31, 2013 at 10:00 AM, type:

```
wbadmin delete systemstatebackup -version:03/31/2013-10:00
```

To delete all system state backups, except the three most recent, type:

```
wbadmin delete systemstatebackup -keepVersions:3
```

To delete the oldest system state backup stored on disk f, type:

```
wbadmin delete systemstatebackup -backupTarget:f -deleteOldest
```

Additional references

- [Command-Line Syntax Key](#)
- [Wbadmin](#)

wbadmin start sysrecovery

4/13/2018 • 2 minutes to read • [Edit Online](#)

Performs a system recovery (bare metal recovery) using the parameters that you specify.

NOTE

This subcommand can be run only from the Windows Recovery Environment, and it is not listed by default in the usage text of **Wbadmin**. For more information, see [Windows Recovery Environment \(Windows RE\) Overview](#).

To perform a system recovery with this subcommand, you must be a member of the **Backup Operators** group or the **Administrators** group, or you must have been delegated the appropriate permissions.

For examples of how to use this subcommand, see [Examples](#).

Syntax

```
wbadmin start sysrecovery
-version:<VersionIdentifier>
-backupTarget:{<BackupDestinationVolume> | <NetworkShareHostingBackup>}
[-machine:<BackupMachineName>]
[-restoreAllVolumes]
[-recreateDisks]
[-excludeDisks]
[-skipBadClusterCheck]
[-quiet]
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------------|--|
| -version | Specifies the version identifier for the backup to recover in MM/DD/YYYY-HH:MM format. If you do not know the version identifier, type wbadmin get versions . |
| -backupTarget | Specifies the storage location that contains the backup or backups that you want to recover. This parameter is useful when the storage location is different from where backups of this computer are usually stored. |
| -machine | Specifies the name of the computer that you want to recover. This parameter is useful when multiple computers have been backed up to the same location. Should be used when the -backupTarget parameter is specified. |
| -restoreAllVolumes | Recovers all volumes from the selected backup. If this parameter is not specified, only critical volumes (volumes that contain the system state and operating system components) are recovered. This parameter is useful when you need to recover non-critical volumes during system recovery. |

| PARAMETER | DESCRIPTION |
|----------------------|---|
| -recreateDisks | Recovers a disk configuration to the state that existed when the backup was created. Warning: This parameter deletes all data on volumes that host operating system components. It might also delete data from data volumes. |
| -excludeDisks | Valid only when specified with the -recreateDisks parameter and must be input as a comma-delimited list of disk identifiers (as listed in the output of wbadmin get disks). Excluded disks are not partitioned or formatted. This parameter helps preserve data on disks that you do not want modified during the recovery operation. |
| -skipBadClusterCheck | Skips checking your recovery disks for bad cluster information. If you are restoring to an alternate server or hardware, we recommend that you do not use this parameter. You can manually run chkdsk /b on your recovery disks at any time to check them for bad clusters, and then update the file system information accordingly. Warning: Until you run chkdsk as described, the bad clusters reported on your recovered system may not be accurate. |
| -quiet | Runs the command with no prompts to the user. |

Examples

To start recovering the information from the backup that was run on March 31, 2013 at 9:00 A.M., located on drive d:, type:

```
wbadmin start sysrecovery -version:03/31/2013-09:00 -backupTarget:d:
```

To start recovering the information from the backup that was run on April 30, 2013 at 9:00 A.M., located in the shared folder \\servername\shared: for server01, type:

```
wbadmin start sysrecovery -version:04/30/2013-09:00 -backupTarget:\\servername\share -machine:server01
```

Additional references

- [Command-Line Syntax Key](#)
- [Wbadmin](#)
- [Get-WBBareMetalRecovery cmdlet](#)

wbadmin restore catalog

4/13/2018 • 2 minutes to read • [Edit Online](#)

Recovers a backup catalog for the local computer from a storage location that you specify.

To recover a backup catalog with this subcommand, you must be a member of the **Backup Operators** group or the **Administrators** group, or you must have been delegated the appropriate permissions. In addition, you must run **wbadmin** from an elevated command prompt. (To open an elevated command prompt right-click **Command Prompt**, and then click **Run as administrator**.)

For examples of how to use this subcommand, see [Examples](#).

Syntax

```
wbadmin restore catalog  
-backupTarget:{<BackupDestinationVolume> | <NetworkShareHostingBackup>}  
[-machine:<BackupMachineName>]  
[-quiet]
```

Parameters

| PARAMETER | DESCRIPTION |
|---------------|---|
| -backupTarget | Specifies the location of the backup catalog of the system as it was at the point after the backup was created. |
| -machine | Specifies the name of the computer that you want to recover the backup catalog for. Use when backups for multiple computers have been stored at the same location. Should be used when -backupTarget is specified. |
| -quiet | Runs the subcommand with no prompts to the user. |

Remarks

If the location (disk, DVD, or remote shared folder) where you store your backups is damaged or lost and cannot be used to restore the backup catalog, use **wbadmin delete catalog** to delete the corrupted catalog. In this case, you should create a new backup once your backup catalog is deleted.

Examples

To restore a catalog from a backup stored on disk d:, type:

```
wbadmin restore catalog -backupTarget:d
```

To restore a catalog from a backup stored in the shared folder \\servername\share of server01, type:

```
wbadmin restore catalog -backupTarget:\\servername\share -machine:server01
```

Additional references

- [Command-Line Syntax Key](#)
- [Wbadmin](#)
- [Restore-WBCatalog](#) cmdlet

wbadmin delete catalog

4/13/2018 • 2 minutes to read • [Edit Online](#)

Deletes the backup catalog that is stored on the local computer. Use this command when the backup catalog has been corrupted and you cannot restore it using **wbadmin restore catalog**.

To delete a backup catalog with this subcommand, you must be a member of the **Backup Operators** group or the **Administrators** group, or you must have been delegated the appropriate permissions. In addition, you must run **wbadmin** from an elevated command prompt. (To open an elevated command prompt right-click **Command Prompt**, and then click **Run as administrator**.)

Syntax

```
wbadmin delete catalog  
[-quiet]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| -quiet | Runs the subcommand with no prompts to the user. |

Remarks

If you delete the backup catalog for a computer, you will not be able to access the backups created of that computer using the Windows Server Backup snap-in. In this case, if you can access another backup location, use **wbadmin restore catalog** to restore the backup catalog from that location. You should create a new backup once your backup catalog is deleted.

Additional references

- [Command-Line Syntax Key](#)
- [Wbadmin](#)
- [Remove-WBCatalog](#)

wdsutil

10/17/2017 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

wdsutil is a command-line utility used for managing your Windows Deployment Services server. To run these commands, click **start**, right-click **Command prompt**, and click **Run as administrator**.

Commands

| COMMAND | DESCRIPTION |
|--|---|
| Using the add Command | adds objects or prestages computers. |
| Using the Approve-AutoaddDevices Command | Approves computers that are pending administrator approval. |
| Using the convert-RipreplImage Command | converts an existing remote Installation Preparation (RIPrep) image to a Windows Image (.wim) file. |
| Using the copy Command | Copies an image or a driver group. |
| Using the delete-AutoaddDevices Command | deletes computers that are in the Auto-add database (which stores information about the computers on the server). |
| Using the Disable Command | Disables all services for Windows Deployment Services. |
| Using the Disconnect-Client Command | Disconnects a client from a multicast transmission or namespace. |
| Using the Enable Command | Enables all services for Windows Deployment Services. |
| Using the Export-Image Command | Exports an image from the image store to a .wim file. |
| Using The get Command | Retrieves properties and attributes about the specified object. |
| Using the Initialize-Server Command | Configures a Windows Deployment Services server for initial use. |
| Using the New Command | creates new capture and discover images as well as multicast transmissions and namespaces. |
| The progress Command | Displays the progress status while a command is being executed. |
| Using The Reject-AutoaddDevices Command | Rejects computers that are pending administrator approval. |
| Using the remove Command | removes objects. |

| COMMAND | DESCRIPTION |
|---|---|
| Using the replace-Image Command | replaces a boot or installation image with a new version of that image. |
| The Set Command | Sets properties and attributes on the specified object. |
| The start Server Command | starts all services on the Windows Deployment Services server, including multicast transmissions, namespaces, and the Transport Server. |
| The Stop Server Command | Stops all services on the Windows Deployment Services server. |
| The uninitialize-Server Option | reverts changes made during server initialization. |
| The Update-ServerFiles Command | Updates server files on the remoteInstall share. |
| The verbose Command | Displays verbose output for the specified command. |

Enables you to create and manage subscriptions to events that are forwarded from remote computers. The remote computer must support the WS-Management protocol. For examples of how to use this command, see [Examples](#).

Syntax

```
wecutil [{es | enum-subscription}]
[{gs | get-subscription} <Subid> [/f:<Format>] [/uni:<Unicode>]]
[{gr | get-subscriptionruntimestatus} <Subid> [<Eventsource> ...]]
[{ss | set-subscription} [<Subid> [/e:[<Subenabled>]] [/esa:<Address>] [/ese:[<Srcenabled>]] [/aes] [/res]
[/un:<Username>] [/up:<Password>] [/d:<Desc>] [/uri:<Uri>] [/cm:<Configmode>] [/ex:<Expires>] [/q:<Query>]
[/dia:<Dialect>] [/tn:<Transportname>] [/tp:<Transportport>] [/dm:<Deliverymode>] [/dmi:<Deliverymax>] [/dmlt:
<Deliverytime>] [/hi:<Heartbeat>] [/cf:<Content>] [/l:<Locale>] [/ree:[<Readexist>]] [/lf:<Logfile>] [/pn:
<Publishername>] [/essp:<Enableport>] [/hn:<Hostname>] [/ct:<Type>]] [/c:<Configfile>] [/cun:<Username>] /cup:
<Password>]]]
[{cs | create-subscription} <Configfile> [/cun:<Username>] /cup:<Password>]]
[{ds | delete-subscription} <Subid>]
[{rs | retry-subscription} <Subid> [<Eventsource>...]]
[{qc | quick-config} [/q:[<Quiet>]]].
```

Parameters

| PARAMETER | DESCRIPTION |
|---|---|
| {es enum-subscription} | Displays the names of all remote event subscriptions that exist. |
| {gs get-subscription} <Subid> [/f:<Format>] [/uni:<Unicode>] | Displays remote subscription configuration information. <Subid> is a string that uniquely identifies a subscription. <Subid> is the same as the string that was specified in the <SubscriptionId> tag of the XML configuration file, which was used to create the subscription. |
| {gr get-subscriptionruntimestatus} <Subid> [<Eventsource>...] | Displays the runtime status of a subscription. <Subid> is a string that uniquely identifies a subscription. <Subid> is the same as the string that was specified in the <SubscriptionId> tag of the XML configuration file, which was used to create the subscription. <Eventsource> is a string that identifies a computer that serves as a source of events. <Eventsource> should be a fully qualified domain name, a NetBIOS name, or an IP address. |

| PARAMETER | DESCRIPTION |
|---|--|
| <pre>{ss set-subscription} <Subid> [/e:[<Subenabled>]] [/esa: <Address>] [/ese:[<Srcenabled>]] [/aes] [/res] [/un: <Username>] [/up:<Password>] [/d:<Desc>] [/uri:<Uri>] [/cm:<Configmode>] [/ex:<Expires>] [/q:<Query>] [/dia: <Dialect>] [/tn:<Transportname>] [/tp:<Transportport>] [/dm: <Deliverymode>] [/dmi:<Deliverymax>] [/dmlt: <Deliverytime>] [/hi:<Heartbeat>] [/cf:<Content>] [/l: <Locale>] [/ree:[<Readexist>]] [/lf:<Logfile>] [/pn: <Publishername>] [/essp:<Enableport>] [/hn:<Hostname>] [/ct:<Type>] or {ss set-subscription /c:<Configfile> [/cun:<Comusername> /cup:<Compassword>]</pre> | <p>Changes the subscription configuration. You can specify the subscription ID and the appropriate options to change subscription parameters, or you can specify an XML configuration file to change subscription parameters.</p> |
| <pre>{cs create-subscription} <Configfile> [/cun:<Username> /cup: <Password>]</pre> | <p>Creates a remote subscription. <Configfile> specifies the path to the XML file that contains the subscription configuration. The path can be absolute or relative to the current directory.</p> |
| <pre>{ds delete-subscription} <Subid></pre> | <p>Deletes a subscription and unsubscribes from all event sources that deliver events into the event log for the subscription. Any events already received and logged are not deleted. <Subid> is a string that uniquely identifies a subscription. <Subid> is the same as the string that was specified in the <SubscriptionId> tag of the XML configuration file, which was used to create the subscription.</p> |
| <pre>{rs retry-subscription} <Subid> [<Eventsource>...]</pre> | <p>Retries to establish a connection and send a remote subscription request to an inactive subscription. Attempts to reactivate all event sources or specified event sources. Disabled sources are not retried. <Subid> is a string that uniquely identifies a subscription. <Subid> is the same as the string that was specified in the <SubscriptionId> tag of the XML configuration file, which was used to create the subscription. <Eventsource> is a string that identifies a computer that serves as a source of events. <Eventsource> should be a fully qualified domain name, a NetBIOS name, or an IP address.</p> |
| <pre>{qc quick-config} [/q:[<Quiet>]]</pre> | <p>Configures the Windows Event Collector service to ensure a subscription can be created and sustained through reboots. This includes the following steps:</p> <ol style="list-style-type: none"> 1. Enable the ForwardedEvents channel if it is disabled. 2. Set the Windows Event Collector service to delay start. 3. Start the Windows Event Collector service if it is not running. |

Options

| OPTION | DESCRIPTION |
|-------------|--|
| /f:<Format> | <p>Specifies the format of the information that is displayed. <Format> can be XML or Terse. If is XML, the output is displayed in XML format. If <Format> is Terse, the output is displayed in name-value pairs. The default is Terse.</p> |

| OPTION | DESCRIPTION |
|---------------------|---|
| /c:<Configfile> | Specifies the path to the XML file that contains a subscription configuration. The path can be absolute or relative to the current directory. This option can only be used with the /cun and /cup options and is mutually exclusive with all other options. |
| /e:[<Subenabled>] | Enables or disables a subscription. <Subenabled> can be true or false. The default value of this option is true. |
| /esa:<Address> | Specifies the address of an event source. <Address> is a string that contains a fully qualified domain name, a NetBIOS name, or an IP address, which identifies a computer that serves as a source of events. This option should be used with the /ese , /aes , /res , or /un and /up options. |
| /ese:[<Srcenabled>] | Enables or disables an event source. <Srcenabled> can be true or false. This option is allowed only if the /esa option is specified. The default value of this option is true. |
| /aes | Adds the event source that is specified by the /esa option if it is not already a part of the subscription. If the address specified by the /esa option is already a part of the subscription, an error is reported. This option is only allowed if the /esa option is specified. |
| /res | Removes the event source that is specified by the /esa option if it is already a part of the subscription. If the address specified by the /esa option is not a part of the subscription, an error is reported. This option is only allowed if /esa option is specified. |
| /un:<Username> | Specifies the user credential to use with the event source specified by the /esa option. This option is only allowed if the /esa option is specified. |
| /up:<Password> | Specifies the password that corresponds to the user credential. This option is only allowed if the /un option is specified. |
| /d:<Desc> | Provides a description for the subscription. |
| /uri:<Uri> | Specifies the type of the events that are consumed by the subscription. <Uri> contains a URI string that is combined with the address of the event source computer to uniquely identify the source of the events. The URI string is used for all event source addresses in the subscription. |
| /cm:<Configmode> | Sets the configuration mode. <Configmode> can be one of the following strings: Normal, Custom, MinLatency or MinBandwidth. The Normal, MinLatency, and MinBandwidth modes set delivery mode, delivery max items, heartbeat interval, and delivery max latency time. The /dm , /dmi , /hi or /dmlt options may only be specified if the configuration mode is set to Custom. |

| OPTION | DESCRIPTION |
|----------------------|--|
| /ex:<Expires> | Sets the time when the subscription expires. <Expires> should be defined in standard XML or ISO8601 date-time format: yyyy-MM-ddThh:mm:ss[.sss][Z], where T is the time separator and Z indicates UTC time. |
| /q:<Query> | Specifies the query string for the subscription. The format of <Query> may be different for different URI values and applies to all sources in the subscription. |
| /dia:<Dialect> | Defines the dialect that the query string uses. |
| /tn:<Transportname> | Specifies the name of the transport that is used to connect to a remote event source. |
| /tp:<Transportport> | Sets the port number that is used by the transport when connecting to a remote event source. |
| /dm:<Deliverymode> | Specifies the delivery mode. <Deliverymode> can be either pull or push. This option is only valid if the /cm option is set to Custom. |
| /dmi:<Deliverymax> | Sets the maximum number of items for batched delivery. This option is only valid if /cm is set to Custom. |
| /dmlt:<Deliverytime> | Sets the maximum latency in delivering a batch of events. <Deliverytime> is the number of milliseconds. This option is only valid if /cm is set to Custom. |
| /hi:<Heartbeat> | Defines the heartbeat interval. <Heartbeat> is the number of milliseconds. This option is only valid if /cm is set to Custom. |
| /cf:<Content> | Specifies the format of the events that are returned. <Content> can be Events or RenderedText. When the value is RenderedText, the events are returned with the localized strings (such as event description) attached to the event. The default value is RenderedText. |
| /l:<Locale> | Specifies the locale for delivery of the localized strings in RenderedText format. <Locale> is a language and country/region identifier, for example, "EN-us". This option is only valid if the /cf option is set to RenderedText. |
| /ree:[<Readexist>] | Identifies the events that are delivered for the subscription. <Readexist> can true or false. When the is true, all existing events are read from the subscription event sources. When the is false, only future (arriving) events are delivered. The default value is true for a /ree option without a value. If no /ree option is specified, the default value is false. |
| /lf:<Logfile> | Specifies the local event log that is used to store events received from the event sources. |
| /pn:<Publishername> | Specifies the publisher name. It must be a publisher that owns or imports the log specified by the /lf option. |

| OPTION | DESCRIPTION |
|--------------------|---|
| /essp:<Enableport> | Specifies that the port number must be appended to the service principal name of the remote service. <Enableport> can be true or false. The port number is appended when is true. When the port number is appended, some configuration may be required to prevent the access to event sources from being denied. |
| /hn:<Hostname> | Specifies the DNS name of the local computer. This name is used by remote event source to push back events and must be used only for a push subscription. |
| /ct:<Type> | Sets the credential type for the remote source access. <Type> should be one of the following values: default, negotiate, digest, basic or localmachine. The default value is default. |
| /cun:<Comusername> | Sets the shared user credential to be used for event sources that do not have their own user credentials. If this option is specified with the /c option, UserName and UserPassword settings for individual event sources from the configuration file are ignored. If you want to use a different credential for a specific event source, you should override this value by specifying the /un and /up options for a specific event source on the command line of another ss command. |
| /cup:<Compassword> | Sets the user password for the shared user credential. When <Compassword> is set to * (asterisk), the password is read from the console. This option is only valid when the /cun option is specified. |
| /q:[<Quiet>] | Specifies whether the configuration procedure prompts for confirmation. <Quiet> can be true or false. If is true, the configuration procedure does not prompt for confirmation. The default value of this option is false. |

Remarks

IMPORTANT

If you receive the message, "The RPC server is unavailable?" when you try to run wecutil, you need to start the Windows Event Collector service (wecsvc). To start wecsvc, at an elevated command prompt type net start wecsvc.

- The following example shows the contents of a configuration file:

```
<Subscription xmlns="https://schemas.microsoft.com/2006/03/windows/events/subscription">
  <Uri>https://schemas.microsoft.com/wbem/wsmn/1/windows/EventLog</Uri> <!-- Use Normal (default), Custom,
  MinLatency, MinBandwidth --> <ConfigurationMode>Normal</ConfigurationMode> <Description>Forward Sample
  Subscription</Description> <SubscriptionId>SampleSubscription</SubscriptionId> <Query><![CDATA[ <QueryList>
  <Query Path="Application"> <Select>*</Select> </Query> </QueryList> ]]></Query> <EventSources> <EventSource
  Enabled="true"> <Address>mySource.myDomain.com</Address> <UserName>myUserName</UserName> <Password>*
  </Password> </EventSource> </EventSources> <CredentialsType>Default</CredentialsType> <Locale Language="EN-
  US"></Locale> </Subscription>
```

Examples

Output configuration information for a subscription named sub1:

```
wecutil gs sub1
```

Example output:

```
EventSource[0]:  
Address: localhost  
Enabled: true  
Description: Subscription 1  
Uri: wsman:microsoft/logrecord/sel  
DeliveryMode: pull  
DeliveryMaxSize: 16000  
DeliveryMaxItems: 15  
DeliveryMaxLatencyTime: 1000  
HeartbeatInterval: 10000  
Locale:  
ContentFormat: renderedtext  
LogFile: HardwareEvents
```

Display the runtime status of a subscription named sub1:

```
wecutil gr sub1
```

Update the subscription configuration named sub1 from a new XML file called WsSelRg2.xml:

```
wecutil ss sub1 /c:%Windir%\system32\WsSelRg2.xml
```

Update the subscription configuration named sub2 with multiple parameters:

```
wecutil ss sub2 /esa:myComputer /ese /un:uname /up:* /cm:Normal
```

Create a subscription to forward events from a Windows Vista Application event log of a remote computer at mySource.myDomain.com to the ForwardedEvents log (see Remarks for an example of a configuration file):

```
wecutil cs subscription.xml
```

Delete a subscription named sub1:

```
wecutil ds sub1
```

Additional references

[Command-Line Syntax Key](#)

wevtutil

7/26/2018 • 8 minutes to read • [Edit Online](#)

Enables you to retrieve information about event logs and publishers. You can also use this command to install and uninstall event manifests, to run queries, and to export, archive, and clear logs. For examples of how to use this command, see [Examples](#).

Syntax

```
wevtutil [{el | enum-logs}] [{gl | get-log} <Logname> [/f:<Format>]]  
[{sl | set-log} <Logname> [/e:<Enabled>] [/i:<Isolation>] [/lfn:<Logpath>] [/rt:<Retention>] [/ab:<Auto>] [/ms:<MaxSize>] [/l:<Level>] [/k:<Keywords>] [/ca:<Channel>] [/c:<Config>]]  
[{ep | enum-publishers}]  
[{gp | get-publisher} <Publishername> [/ge:<Metadata>] [/gm:<Message>] [/f:<Format>]] [{im | install-manifest} <Manifest>]  
[{um | uninstall-manifest} <Manifest>] [{qe | query-events} <Path> [/lf:<Logfile>] [/sq:<Structquery>] [/q:<Query>] [/bm:<Bookmark>] [/sbm:<Savebm>] [/rd:<Direction>] [/f:<Format>] [/l:<Locale>] [/c:<Count>] [/e:<Element>]]  
[{gli | get-logininfo} <Logname> [/lf:<Logfile>]]  
[{ep1 | export-log} <Path> <Exportfile> [/lf:<Logfile>] [/sq:<Structquery>] [/q:<Query>] [/ow:<Overwrite>]]  
[{al | archive-log} <Logpath> [/l:<Locale>]]  
[{cl | clear-log} <Logname> [/bu:<Backup>] [/r:<Remote>] [/u:<Username>] [/p:<Password>] [/a:<Auth>] [/uni:<Unicode>]]
```

Parameters

| PARAMETER | DESCRIPTION |
|---|---|
| {el enum-logs} | Displays the names of all logs. |
| {gl get-log} <Logname> [/f:<Format>] | Displays configuration information for the specified log, which includes whether the log is enabled or not, the current maximum size limit of the log, and the path to the file where the log is stored. |
| {sl set-log} <Logname> [/e:<Enabled>] [/i:<Isolation>] [/lfn:<Logpath>] [/rt:<Retention>] [/ab:<Auto>] [/ms:<MaxSize>] [/l:<Level>] [/k:<Keywords>] [/ca:<Channel>] [/c:<Config>] | Modifies the configuration of the specified log. |
| {ep enum-publishers} | Displays the event publishers on the local computer. |
| {gp get-publisher} <Publishername> [/ge:<Metadata>] [/gm:<Message>] [/f:<Format>] | Displays the configuration information for the specified event publisher. |
| {im install-manifest} <Manifest> | Installs event publishers and logs from a manifest. For more information about event manifests and using this parameter, see the Windows Event Log SDK at the Microsoft Developers Network (MSDN) Web site (https://msdn.microsoft.com). |

| PARAMETER | DESCRIPTION |
|---|--|
| {um uninstall-manifest} <Manifest> | Uninstalls all publishers and logs from a manifest. For more information about event manifests and using this parameter, see the Windows Event Log SDK at the Microsoft Developers Network (MSDN) Web site (https://msdn.microsoft.com). |
| {qe query-events} <Path> [/f:<Logfile>] [/sq:<Structquery>] [/q:<Query>] [/bm:<Bookmark>] [/sbm:<Savebm>] [/rd:<Direction>] [/f:<Format>] [/l:<Locale>] [/c:<Count>] [/e:<Element>] | Reads events from an event log, from a log file, or using a structured query. By default, you provide a log name for <Path>. However, if you use the /lf option, then <Path> must be a path to a log file. If you use the /sq parameter, <Path> must be a path to a file that contains a structured query. |
| {gli get-loginfo} <Logname> [/f:<Logfile>] | Displays status information about an event log or log file. If the /lf option is used, <Logname> is a path to a log file. You can run wevtutil el to obtain a list of log names. |
| {ep export-log} <Path> <Exportfile> [/f:<Logfile>] [/sq:<Structquery>] [/q:<Query>] [/ow:<Overwrite>] | Exports events from an event log, from a log file, or using a structured query to the specified file. By default, you provide a log name for <Path>. However, if you use the /lf option, then <Path> must be a path to a log file. If you use the /sq option, <Path> must be a path to a file that contains a structured query. <Exportfile> is a path to the file where the exported events will be stored. |
| {al archive-log} <Logpath> [/l:<Locale>] | Archives the specified log file in a self-contained format. A subdirectory with the name of the locale is created and all locale-specific information is saved in that subdirectory. After the directory and log file are created by running wevtutil al , events in the file can be read whether the publisher is installed or not. |
| {cl clear-log} <Logname> [/bu:<Backup>] | Clears events from the specified event log. The /bu option can be used to back up the cleared events. |

Options

| OPTION | DESCRIPTION |
|----------------|--|
| /f:<Format> | Specifies that the output should be either XML or text format. If <Format> is XML, the output is displayed in XML format. If <Format> is Text, the output is displayed without XML tags. The default is Text. |
| /e:<Enabled> | Enables or disables a log. <Enabled> can be true or false. |
| /i:<Isolation> | Sets the log isolation mode. <Isolation> can be system, application or custom. The isolation mode of a log determines whether a log shares a session with other logs in the same isolation class. If you specify system isolation, the target log will share at least write permissions with the System log. If you specify application isolation, the target log will share at least write permissions with the Application log. If you specify custom isolation, you must also provide a security descriptor by using the /ca option. |

| OPTION | DESCRIPTION |
|-------------------|--|
| /fn:<Logpath> | Defines the log file name. <Logpath> is a full path to the file where the Event Log service stores events for this log. |
| /rt:<Retention> | Sets the log retention mode. <Retention> can be true or false. The log retention mode determines the behavior of the Event Log service when a log reaches its maximum size. If an event log reaches its maximum size and the log retention mode is true, existing events are retained and incoming events are discarded. If the log retention mode is false, incoming events overwrite the oldest events in the log. |
| /ab:<Auto> | Specifies the log auto-backup policy. <Auto> can be true or false. If this value is true, the log will be backed up automatically when it reaches the maximum size. If this value is true, the retention (specified with the /rt option) must also be set to true. |
| /ms:<MaxSize> | Sets the maximum size of the log in bytes. The minimum log size is 1048576 bytes (1024KB) and log files are always multiples of 64KB, so the value you enter will be rounded off accordingly. |
| /l:<Level> | Defines the level filter of the log. <Level> can be any valid level value. This option is only applicable to logs with a dedicated session. You can remove a level filter by setting to 0. |
| /k:<Keywords> | Specifies the keywords filter of the log. <Keywords> can be any valid 64 bit keyword mask. This option is only applicable to logs with a dedicated session. |
| /ca:<Channel> | Sets the access permission for an event log. <Channel> is a security descriptor that uses the Security Descriptor Definition Language (SDDL). For more information about SDDL format, see the Microsoft Developers Network (MSDN) Web site (https://msdn.microsoft.com). |
| /c:<Config> | Specifies the path to a configuration file. This option will cause log properties to be read from the configuration file defined in <Config>. If you use this option, you must not specify a parameter. The log name will be read from the configuration file. |
| /ge:<Metadata> | Gets metadata information for events that can be raised by this publisher. <Metadata> can be true or false. |
| /gm:<Message> | Displays the actual message instead of the numeric message ID. <Message> can be true or false. |
| /lf:<Logfile> | Specifies that the events should be read from a log or from a log file. <Logfile> can be true or false. If true, the parameter to the command is the path to a log file. |
| /sq:<Structquery> | Specifies that events should be obtained with a structured query. <Structquery> can be true or false. If true, is the path to a file that contains a structured query. |

| OPTION | DESCRIPTION |
|-----------------|--|
| /q:<Query> | Defines the XPath query to filter the events that are read or exported. If this option is not specified, all events will be returned or exported. This option is not available when /sq is true. |
| /bm:<Bookmark> | Specifies the path to a file that contains a bookmark from a previous query. |
| /sbm:<Savebm> | Specifies the path to a file that is used to save a bookmark of this query. The file name extension should be .xml. |
| /rd:<Direction> | Specifies the direction in which events are read. <Direction> can be true or false. If true, the most recent events are returned first. |
| /l:<Locale> | Defines a locale string that is used to print event text in a specific locale. Only available when printing events in text format using the /f option. |
| /c:<Count> | Sets the maximum number of events to read. |
| /e:<Element> | Includes a root element when displaying events in XML. <Element> is the string that you want within the root element. For example, /e:root would result in XML that contains the root element pair <root>. |
| /ow:<Overwrite> | Specifies that the export file should be overwritten. <Overwrite> can be true or false. If true, and the export file specified in already exists, it will be overwritten without confirmation. |
| /bu:<Backup> | Specifies the path to a file where the cleared events will be stored. Include the .evtx extension in the name of the backup file. |
| /r:<Remote> | Runs the command on a remote computer. <Remote> is the name of the remote computer. The im and um parameters do not support remote operation. |
| /u:<Username> | Specifies a different user to log on to a remote computer. <Username> is a user name in the form domain\user or user. This option is only applicable when the /r option is specified. |
| /p:<Password> | Specifies the password for the user. If the /u option is used and this option is not specified or <Password> is "", <i>the user will be prompted to enter a password. This option is only applicable when the **/u* option is specified.</i> |
| /a:<Auth> | Defines the authentication type for connecting to a remote computer. <Auth> can be Default, Negotiate, Kerberos or NTLM. The default is Negotiate. |
| /uni:<Unicode> | Displays the output in Unicode. <Unicode> can be true or false. If is true then the output is in Unicode. |

Remarks

- Using a configuration file with the sl parameter

The configuration file is an XML file with the same format as the output of wevtutil gl <Logname> /f.xml. The following example shows the format of a configuration file that enables retention, enables autobackup, and sets the maximum log size on the Application log:

```
<?xml version="1.0" encoding="UTF-8"?>
<channel name="Application" isolation="Application"
xmlns="https://schemas.microsoft.com/win/2004/08/events">
  <logging>
    <retention>true</retention>
    <autoBackup>true</autoBackup>
    <maxSize>9000000</maxSize>
  </logging>
  <publishing>
    </publishing>
</channel>
```

Examples

List the names of all logs:

```
wevtutil el
```

Display configuration information about the System log on the local computer in XML format:

```
wevtutil gl System /f.xml
```

Use a configuration file to set event log attributes (see Remarks for an example of a configuration file):

```
wevtutil sl /c:config.xml
```

Display information about the Microsoft-Windows-Eventlog event publisher, including metadata about the events that the publisher can raise:

```
wevtutil gp Microsoft-Windows-Eventlog /ge:true
```

Install publishers and logs from the myManifest.xml manifest file:

```
wevtutil im myManifest.xml
```

Uninstall publishers and logs from the myManifest.xml manifest file:

```
wevtutil um myManifest.xml
```

Display the three most recent events from the Application log in textual format:

```
wevtutil qe Application /c:3 /rd:true /f:text
```

Display the status of the Application log:

```
wevtutil gli Application
```

Export events from System log to C:\backup\system0506.evtx:

```
wevtutil epl System C:\backup\system0506.evtx
```

Clear all of the events from the Application log after saving them to C:\admin\backups\a10306.evtx:

```
wevtutil cl Application /bu:C:\admin\backups\a10306.evtx
```

Additional References

[Command-Line Syntax Key](#)

where

7/11/2018 • 2 minutes to read • [Edit Online](#)

Displays the location of files that match the given search pattern.

For examples of how to use this command, see [Examples](#).

Syntax

```
where [/r <Dir>] [/q] [/f] [/t] [$<ENV>:|<Path>:]<Pattern>[ ...]
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------------------------|--|
| /r <Dir> | Indicates a recursive search, starting with the specified directory. |
| /q | Returns an exit code (0 for success, 1 for failure) without displaying the list of matched files. |
| /f | Displays the results of the where command in quotation marks. |
| /t | Displays the file size and the last modified date and time of each matched file. |
| [\$<ENV>: <Path>:]<Pattern>[...] | Specifies the search pattern for the files to match. At least one pattern is required, and the pattern can include wildcard characters (* and ?). By default, where searches the current directory and the paths that are specified in the PATH environment variable. You can specify a different path to search by using the format \$ENV:Pattern (where ENV is an existing environment variable containing one or more paths) or by using the format Path:Pattern (where Path is the directory path you want to search). These optional formats should not be used with the /r command-line option. |
| /? | Displays help at the command prompt. |

Remarks

- If you do not specify a file name extension, the extensions listed in the PATHEXT environment variable are appended to the pattern by default.
- **Where** can run recursive searches, display file information such as date or size, and accept environment variables in place of paths on local computers.

Examples

To find all files named Test in drive C of the current computer and its subdirectories, type:

```
where /r c:\ test
```

To list all files in the Public directory, type:

```
where $public:*.*
```

To find all files named Notepad in drive C of the remote computer, Computer1, and its subdirectories, type:

```
where /r \\computer1\c notepad.*
```

Additional references

[Command-Line Syntax Key](#)

whoami

4/13/2018 • 2 minutes to read • [Edit Online](#)

Displays user, group and privileges information for the user who is currently logged on to the local system. If used without parameters, **whoami** displays the current domain and user name.

For examples of how to use this command, see [Examples](#).

Syntax

```
whoami [/upn | /fqdn | /logonid]
whoami {[[/user] [/groups] [/priv]]} [/fo <Format>] [/nh]
whoami /all [/fo <Format>] [/nh]
```

Parameters

| PARAMETER | DESCRIPTION |
|--------------|--|
| /upn | Displays the user name in user principal name (UPN) format. |
| /fqdn | Displays the user name in fully qualified domain name (FQDN) format. |
| /logonid | Displays the logon ID of the current user. |
| /user | Displays the current domain and user name and the security identifier (SID). |
| /groups | Displays the user groups to which the current user belongs. |
| /priv | Displays the security privileges of the current user. |
| /fo <Format> | Specifies the output format. Valid values include: table Displays output in a table. This is the default value. list Displays output in a list. csv Displays output in comma-separated value (CSV) format. |
| /all | Displays all information in the current access token, including the current user name, security identifiers (SID), privileges, and groups that the current user belongs to. |
| /nh | Specifies that the column header should not be displayed in the output. This is valid only for table and CSV formats. |
| /? | Displays help at the command prompt. |

Examples

To display the domain and user name of the person who is currently logged on to this computer, type:

```
whoami
```

Output similar to the following appears:

```
DOMAIN1\administrator
```

To display all of the information in the current access token, type:

```
whoami /all
```

Additional references

[Command-Line Syntax Key](#)

winnt

4/13/2018 • 2 minutes to read • [Edit Online](#)

Winnt is deprecated, and is not guaranteed to be supported in future releases of Windows.

This tool is included in Windows Server 2003. For more information see [Winnt](#).

winnt32

7/11/2018 • 8 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Performs an installation of or upgrade to a product in Windows Server 2003. You can run **winnt32** at the command prompt on a computer running Windows 95, Windows 98, Windows Millennium edition, Windows NT, Windows 2000, Windows XP, or a product in the Windows Server 2003. If you run **winnt32** on a computer running Windows NT version 4.0, you must first apply Service Pack 5 or later.

Syntax

```
winnt32 [/checkupgradeonly] [/cmd: <CommandLine>] [/cmdcons] [/copydir:{i386|ia64}\<FolderName>] [/copysource: <FolderName>] [/debug[<Level>]:[ <FileName>]] [/dudisable] [/dupprepare: <pathName>] [/dushare: <pathName>] [/emsport:{com1|com2|usebiossettings|off}] [/emsbaudrate: <BaudRate>] [/m: <FolderName>] [/makelocalsource] [/noreboot] [/s: <Sourcepath>] [/syspart: <DriveLetter>] [/tempdrive: <DriveLetter>] [/udf: <ID>[,<UDB_File>]] [/unattend[<Num>]:[ <AnswerFile>]]
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------------|--|
| /checkupgradeonly | Checks your computer for upgrade compatibility with products in Windows Server 2003. if you use this option with /unattend , no user input is required. Otherwise, the results are displayed on the screen, and you can save them under the file name you specify. The default file name is upgrade.txt in the systemroot folder. |
| /cmd | Instructs setup to carry out a specific command before the final phase of setup. This occurs after your computer has restarted and after setup has collected the necessary configuration information, but before setup is complete. |
| <CommandLine> | Specifies the commandline to be carried out before the final phase of setup. |
| /cmdcons | On an x86-based computer, installs the recovery Console as a startup option. The recovery Console is a command-line interface from which you can perform tasks such as starting and stopping services and accessing the local drive (including drives formatted with NTFS). You can only use the /cmdcons option after setup is finished. |

| PARAMETER | DESCRIPTION |
|--------------|--|
| /copydir | <p>creates an additional folder within the folder in which the operating system files are installed. for example, for x86 and x64-based computers, you could create a folder called <i>Private_drivers</i> within the i386 source folder for your installation, and place driver files in the folder. type /copydir:i386\Private_drivers to have setup copy that folder to your newly installed computer, making the new folder location systemroot\Private_drivers.</p> <ul style="list-style-type: none"> - i386 specifies i386 - ia64 specifies ia64 <p>You can use /copydir to create as many additional folders as you want.</p> |
| <FolderName> | Specifies the folder that you created to hold modifications for your site. |
| /copysource | <p>creates a temporary additional folder within the folder in which the operating system files are installed. You can use /copysource to create as many additional folders as you want.</p> <p>Unlike the folders /copydir creates, /copysource folders are deleted after Setup completes.</p> |
| /debug | creates a debug log at the level specified, for example, /debug4:Debug.log . The default log file is C:\systemroot\winnt32.log , and |
| <level> | <p>Level Values and descriptions</p> <ul style="list-style-type: none"> - 0: Severe Errors - 1: Errors - 2: Default level. Warnings - 3: Information - 4: detailed information for debugging <p>Each level includes the levels below it.</p> |
| /dudisable | Prevents Dynamic Update from running. Without Dynamic Update, setup runs only with the original setup files. This option will disable Dynamic Update even if you use an answer file and specify Dynamic Update options in that file. |
| /dupprepare | Carries out preparations on an installation share so that it can be used with Dynamic Update files that you downloaded from the Windows Update Web site. This share can then be used for installing Windows XP for multiple clients. |
| <pathName> | Specifies full path name. |
| /dushare | Specifies a share on which you previously downloaded Dynamic Update files (updated files for use with Setup) from the Windows Update Web site, and on which you previously ran /dupprepare:< pathName> . When run on a client, specifies that the client installation will make use of the updated files on the share specified in . |

| PARAMETER | DESCRIPTION |
|------------------|---|
| /emsport | <p>Enables or disables Emergency Management Services during setup and after the server operating system has been installed. With Emergency Management Services, you can remotely manage a server in emergency situations that would typically require a local keyboard, mouse, and monitor, such as when the network is unavailable or the server is not functioning properly. Emergency Management Services has specific hardware requirements, and is available only for products in Windows Server 2003.</p> <ul style="list-style-type: none"> - com1 is applicable only for x86-based computers (not Itanium architecture-based computers). - com2 is applicable only for x86-based computers (not Itanium architecture-based computers). - Default. Uses the setting specified in the BIOS Serial Port Console Redirection (SPCR) table, or, in Itanium architecture-based systems, through the EFI console device path. If you specify usebiossettings and there is no SPCR table or appropriate EFI console device path, Emergency Management Services will not be enabled. - off disables Emergency Management Services. You can later enable it by modifying the boot settings. |
| /emsbaudrate | for x86-based computers, specifies the baud rate for Emergency Management Services. (The option is not applicable for Itanium architecture-based computers.) Must be used with /emsport:com1 or /emsport:com2 (otherwise, /emsbaudrate is ignored). |
| <BaudRate> | Specifies baudrate of 9600, 19200, 57600, or 115200. 9600 is the default. |
| /m | Specifies that setup copies replacement files from an alternate location. Instructs setup to look in the alternate location first, and if files are present, to use them instead of the files from the default location. |
| /makelocalsource | Instructs setup to copy all installation source files to your local hard disk. Use /makelocalsource when installing from a cd to provide installation files when the cd is not available later in the installation. |
| /noreboot | Instructs setup to not restart the computer after the file copy phase of setup is completed so that you can run another command. |
| /s | Specifies the source location of the files for your installation. To simultaneously copy files from multiple servers, type the /s:<Sourcepath> option multiple times (up to a maximum of eight). If you type the option multiple times, the first server specified must be available, or setup will fail. |
| <Sourcepath> | Specifies full source path name. |

| PARAMETER | DESCRIPTION |
|---------------|---|
| /syspart | <p>On an x86-based computer, specifies that you can copy setup startup files to a hard disk, mark the disk as active, and then install the disk into another computer. When you start that computer, it automatically starts with the next phase of setup.</p> <p>You must always use the /tempdrive parameter with the /syspart parameter.</p> <p>You can start winnt32 with the /syspart option on an x86-based computer running Windows NT 4.0, Windows 2000, Windows XP, or a product in Windows Server 2003. If the computer is running Windows NT version 4.0, it requires Service Pack 5 or later. The computer cannot be running Windows 95, Windows 98, or Windows Millennium edition.</p> |
| <DriveLetter> | Specifies the drive letter. |
| /tempdrive | <p>directs setup to place temporary files on the specified partition.</p> <p>for a new installation, the server operating system will also be installed on the specified partition.</p> <p>for an upgrade, the /tempdrive option affects the placement of temporary files only; the operating system will be upgraded in the partition from which you run winnt32.</p> |
| /udf | <p>Indicates an identifier (<ID>) that setup uses to specify how a Uniqueness Database (UDB) file modifies an answer file (see the /unattend option). The UDB overrides values in the answer file, and the identifier determines which values in the UDB file are used. For example, /udf:RAS_user,Our_company.udb overrides settings specified for the RAS_user identifier in the Our_company.udb file. If no <UDB_file> is specified, setup prompts the user to insert a disk that contains the \$Unique\$.udb file.</p> |
| <ID> | Indicates an identifier used to specify how a Uniqueness Database (UDB) file modifies an answer file. |
| <UDB_file> | Specifies a Uniqueness Database (UDB) file. |
| /unattend | On an x86-based computer, upgrades your previous version of Windows NT 4.0 Server (with Service Pack 5 or later) or Windows 2000 in unattended setup mode. All user settings are taken from the previous installation, so no user intervention is required during setup. |
| <num> | Specifies the number of seconds between the time that setup finishes copying the files and when it restarts your computer. You can use <Num> on any computer running Windows 98, Windows Millennium edition, Windows NT, Windows 2000, Windows XP, or a product in Windows Server 2003 . If the computer is running Windows NT version 4.0, it requires Service Pack 5 or later. |
| <AnswerFile> | Provides setup with your custom specifications |

| PARAMETER | DESCRIPTION |
|-----------------|--------------------------------------|
| <code>/?</code> | Displays help at the command prompt. |

Remarks

If you are deploying Windows XP on client computers, you can use the version of winnt32.exe that comes with Windows XP. Another way to deploy Windows XP is to use winnt32.msi, which works through Windows Installer, part of the IntelliMirror set of technologies. For more information about client deployments, see the Windows Server 2003 Deployment Kit, which is described in [Using the Windows Deployment and Resource Kits](#).

On an Itanium-based computer, **winnt32** can be run from the Extensible Firmware Interface (EFI) or from Windows Server 2003 Enterprise, Windows Server 2003 R2 Enterprise, Windows Server 2003 R2 Datacenter, or Windows Server 2003 Datacenter. Also, on an Itanium architecture-based computer, **/cmdcons** and **/syspart** are not available, and options relating to upgrades are not available. for more information about hardware compatibility, see [Hardware compatibility](#). for more detailed information about using Dynamic Update and installing multiple clients, see the Windows Server 2003 Deployment Kit, which is described in [Using the Windows Deployment and Resource Kits](#). for information about modifying boot settings, see the Windows Deployment and Resource Kits for Windows Server 2003. For more information, see [Using the Windows Deployment and Resource Kits](#). Using the **/unattend** command-line option to automate setup affirms that you have read and accepted the Microsoft License Agreement for Windows Server 2003. Before using this command-line option to install Windows Server 2003 on behalf of an organization other than your own, you must confirm that the end user (whether an individual, or a single entity) has received, read, and accepted the terms of the Microsoft License Agreement for that product. OEMs may not specify this key on machines being sold to end users.

Additional references

- [Command-Line Syntax Key](#)

winpop

4/13/2018 • 2 minutes to read • [Edit Online](#)

Winpop is deprecated, and is not guaranteed to be supported in future releases of Windows.

This tool is included in Windows Server 2003. For more information see [winpop](#).

winrs

7/11/2018 • 2 minutes to read • [Edit Online](#)

Applies To: Windows Server (Semi-Annual Channel), Windows Server 2016, Windows Server 2012 R2, Windows Server 2012

Windows remote Management allows you to manage and execute programs remotely.

Syntax

```
winrs [/<parameter>[:<value>]] <command>
```

Parameters

| PARAMETER | DESCRIPTION |
|-----------------------|--|
| [/remote]:<endpoint> | <p>Specifies the target endpoint using a NetBIOS name or the standard connection:</p> <p>- : [<transport>:/]<target>[:<port>]</p> <p>if not specified, /r:localhost is used.</p> |
| /unencrypted] | <p>Specifies that the messages to the remote shell will not be encrypted. This is useful for troubleshooting or when the network traffic is already encrypted using ipsec, or when physical security is enforced.</p> <p>By default, the messages are encrypted using Kerberos or NTLM keys.</p> <p>This command-line option is ignored when HTTPS transport is selected.</p> |
| /username]:<username> | <p>Specifies username on command line.</p> <p>if not specified, the tool will use Negotiate authentication or prompt for the name.</p> <p>if /username is specified, /password must also be specified.</p> |
| /password]:<password> | <p>Specifies password on command line.</p> <p>if /password is not specified but /username is, the tool will prompt for the password.</p> <p>if /password is specified, /username must also be specified.</p> |
| /timeout:<seconds> | <p>This option is deprecated.</p> |

| PARAMETER | DESCRIPTION |
|------------------------|---|
| /directory:<path> | Specifies starting directory for remote shell. if not specified, the remote shell will start in the user's home directory defined by the environment variable %USERPROFILE% . |
| /environment:<string>= | Specifies a single environment variable to be set when shell starts, which allows changing default environment for shell. Multiple occurrences of this switch must be used to specify multiple environment variables. |
| /noecho | Specifies that echo should be disabled. This may be necessary to ensure that user's answers to remote prompts are not displayed locally. By default echo is "on". |
| /noprofile | Specifies that the user's profile should not be loaded. By default, the server will attempt to load the user profile. if the remote user is not a local administrator on the target system, then this option will be required (the default will result in error). |
| /allowdelegate | Specifies that the user's credentials can be used to access a remote share, for example, found on a different machine than the target endpoint. |
| /compression | Turn on compression. Older installations on remote machines may not support compression so it is off by default. Default setting is off, since older installations on remote machines may not support compression. |
| /usessl | Use an SSL connection when using a remote endpoint. Specifying this instead of the transport https: will use the default WinRM default port. |
| /? | Displays help at the command prompt. |

Remarks

- All command-line options accept either short form or long form. For example both **/r** and **/remote** are valid.
- To terminate the **/remote** command, the user can type **Ctrl-C** or **Ctrl-break**, which will be sent to the remote shell. The second **Ctrl-C** will force termination of **winrs.exe**.
- To manage active remote shells or winrs configuration, use the WinRM tool. The URI alias to manage active shells is **shell/cmd**. The URI alias for winrs configuration is **winrm/config/winrs**.

Examples

```
winrs /r:https://contoso.com command
```

```
winrs /r:contoso.com /u:ssl command
```

```
winrs /r:myserver command
```

```
winrs /r:http://127.0.0.1 command
```

```
winrs /r:http://169.51.2.101:80 /unencrypted command
```

```
winrs /r:https://[::FFFF:129.144.52.38] command
```

```
winrs /r:http://[1080:0:0:0:8:800:200C:417A]:80 command
```

```
winrs /r:https://contoso.com /t:600 /u:administrator /p:%fgh7 ipconfig
```

```
winrs /r:myserver /env:path=%path%;c:\tools /env:TEMP=d:\temp config.cmd
```

```
winrs /r:myserver netdom join myserver /domain:testdomain /userd:johns /passwordd:%fgh789
```

```
winrs /r:myserver /ad /u:administrator /p:%fgh7 dir \\anotherserver\share
```

Additional references

- [Command-Line Syntax Key](#)

wmic

7/11/2018 • 2 minutes to read • [Edit Online](#)

Displays WMI information inside an interactive command shell.

For examples of how to use this command, see [Examples](#).

Syntax

```
command </parameter>
```

Sub-commands

The following sub-commands are available at all times:

| SUB-COMMAND | DESCRIPTION |
|---------------|---|
| class | Escapes from the default alias mode of WMIC to access classes in the WMI schema directly. |
| path | Escapes from the default alias mode of WMIC to access instances in the WMI schema directly. |
| context | Displays the current values of all global switches. |
| [quit exit] | Exits the WMIC command shell. |

Parameters

| PARAMETER | DESCRIPTION |
|-----------|--|
| | <Concise description, starts with a verb.> |
| | <Another concise description, starts with a verb.> |

Examples

To display the current values of all global switches, type:

```
wmic context
```

Output similar to the following displays:

```
NAMESPACE      : root\cimv2
ROLE           : root\cli
NODE(S)        : BOBENTERPRISE
IMPLEVEL       : IMPERSONATE
[AUTHORITY     : N/A]
AUTHLEVEL      : PKTPRIVACY
LOCALE         : ms_409
PRIVILEGES     : ENABLE
TRACE          : OFF
RECORD         : N/A
INTERACTIVE    : OFF
FAILFAST       : OFF
OUTPUT         : STDOUT
APPEND         : STDOUT
USER           : N/A
AGGREGATE      : ON
```

To change the language ID used by the command line to English (locale ID 409), type:

```
wmic /locale:ms_409
```

Additional references

[Command-Line Syntax Key](#)

wscript

8/21/2018 • 3 minutes to read • [Edit Online](#)

Windows Script Host provides an environment in which users can execute scripts in a variety of languages that use a variety of object models to perform tasks.

Syntax

```
wscript [<scriptname>] [/b] [/d] [/e:<engine>] [{/h:cscript|/h:wscript}] [/i] [/job:<identifier>]
[/{logo|nologo}] [/s] [/t:<number>] [/x] [/?] [<ScriptArguments>]
```

Parameters

| PARAMETER | DESCRIPTION |
|-------------------|--|
| scriptname | Specifies the path and file name of the script file. |
| /b | Specifies batch mode, which does not display alerts, scripting errors, or input prompts. This is the opposite of /i . |
| /d | Starts the debugger. |
| /e | <p>Specifies the engine that is used to run the script. This lets you run scripts that use a custom file name extension. Without the /e parameter, you can only run scripts that use registered file name extensions. For example, if you try to run this command:</p> <pre>cscript test.admin</pre> <p>You will receive this error message: Input Error: There is no script engine for file extension ".admin."</p> <p>One advantage of using nonstandard file name extensions is that it guards against accidentally double-clicking a script and running something you really did not want to run.</p> <p>This does not create a permanent association between the .admin file name extension and VBScript. Each time you run a script that uses a .admin file name extension, you will need to use the /e parameter.</p> |
| /h:cscript | Registers cscript.exe as the default script host for running scripts. |
| /h:wscript | Registers wscript.exe as the default script host for running scripts. This is the default when the /h option is omitted. |
| /i | <p>Specifies interactive mode, which displays alerts, scripting errors, and input prompts.</p> <p>This is the default and the opposite of /b.</p> |
| /job:<identifier> | Runs the job identified by <i>identifier</i> in a .wsf script file. |
| /logo | <p>Specifies that the Windows Script Host banner is displayed in the console before the script runs.</p> <p>This is the default and the opposite of /nologo.</p> |

| PARAMETER | DESCRIPTION |
|-----------------|---|
| /nologo | Specifies that the Windows Script Host banner is not displayed before the script runs. This is the opposite of /logo . |
| /s | Saves the current command prompt options for the current user. |
| /t:<number> | Specifies the maximum time the script can run (in seconds). You can specify up to 32,767 seconds. The default is no time limit. |
| /x | Starts the script in the debugger. |
| ScriptArguments | Specifies the arguments passed to the script. Each script argument must be preceded by a slash (/). |
| /? | Displays Help at the command prompt. |

Remarks

- Performing this task does not require you to have administrative credentials. Therefore, as a security best practice, consider performing this task as a user without administrative credentials.
- To open a command prompt, on the **Start** screen, type **cmd**, and then click **command prompt**.
- Each parameter is optional; however, you cannot specify script arguments without specifying a script. If you do not specify a script or any script arguments, **wscript.exe** displays the **Windows Script Host Settings** dialog box, which you can use to set global scripting properties for all scripts that **wscript.exe** runs on the local computer.
- The **/t** parameter prevents excessive running of scripts by setting a timer. When the time exceeds the specified value, **wscript** interrupts the script engine and ends the process.
- Windows script files usually have one of the following file name extensions: **.wsf**, **.vbs**, **.js**.
- If you double-click a script file with an extension that has no association, the **Open With** dialog box appears. Select **wscript** or **cscript**, and then select **Always use this program to open this file type**. This registers **wscript.exe** or **cscript.exe** as the default script host for files of this file type.
- You can set properties for individual scripts. See [Windows Script Host overview](#) for more information.
- Windows Script Host can use **.wsf** script files. Each **.wsf** file can use multiple scripting engines and perform multiple jobs.

Additional references

- [Command-Line Syntax Key](#)

xcopy

8/21/2018 • 8 minutes to read • [Edit Online](#)

Copies files and directories, including subdirectories

For examples of how to use this command, see [Examples](#)

Syntax

```
Xcopy <Source> [<Destination>] [/w] [/p] [/c] [/v] [/q] [/f] [/l] [/g] [/d [:MM-DD-YYYY]] [/u] [/i] [/s [/e]]  
[/t] [/k] [/r] [/h] [{/a | /m}] [/n] [/o] [/x] [/exclude:FileName1[+FileName2][+FileName3]] [{/y | /-y}]  
[/z] [/b] [/j]
```

Parameters

| PARAMETER | DESCRIPTION |
|------------------|---|
| <Source> | Required. Specifies the location and names of the files you want to copy. This parameter must include either a drive or a path. |
| <Destination> | Specifies the destination of the files you want to copy. This parameter can include a drive letter and colon, a directory name, a file name, or a combination of these. |
| /w | Displays the following message and waits for your response before starting to copy files: Press any key to begin copying file(s) |
| /p | Prompts you to confirm whether you want to create each destination file. |
| /c | Ignores errors. |
| /v | Verifies each file as it is written to the destination file to make sure that the destination files are identical to the source files. |
| /q | Suppresses the display of xcopy messages. |
| /f | Displays source and destination file names while copying. |
| /l | Displays a list of files that are to be copied. |
| /g | Creates decrypted <i>Destination</i> files when the destination does not support encryption. |
| /d [:MM-DD-YYYY] | Copies source files changed on or after the specified date only. If you do not include a <i>MM-DD-YYYY</i> value, xcopy copies all <i>Source</i> files that are newer than existing <i>Destination</i> files. This command-line option allows you to update files that have changed. |

| PARAMETER | DESCRIPTION |
|-----------|---|
| /u | Copies files from <i>Source</i> that exist on <i>Destination</i> only. |
| /i | If <i>Source</i> is a directory or contains wildcards and <i>Destination</i> does not exist, xcopy assumes <i>Destination</i> specifies a directory name and creates a new directory. Then, xcopy copies all specified files into the new directory. By default, xcopy prompts you to specify whether <i>Destination</i> is a file or a directory. |
| /s | Copies directories and subdirectories, unless they are empty. If you omit /s , xcopy works within a single directory. |
| /e | Copies all subdirectories, even if they are empty. Use /e with the /s and /t command-line options. |
| /t | Copies the subdirectory structure (that is, the tree) only, not files. To copy empty directories, you must include the /e command-line option. |
| /k | Copies files and retains the read-only attribute on <i>Destination</i> files if present on the <i>Source</i> files. By default, xcopy removes the read-only attribute. |
| /r | Copies read-only files. |
| /h | Copies files with hidden and system file attributes. By default, xcopy does not copy hidden or system files |
| /a | Copies only <i>Source</i> files that have their archive file attributes set. /a does not modify the archive file attribute of the source file. For information about how to set the archive file attribute by using attrib , see Additional references . |
| /m | Copies <i>Source</i> files that have their archive file attributes set. Unlike /a , /m turns off archive file attributes in the files that are specified in the source. For information about how to set the archive file attribute by using attrib , see Additional references . |
| /n | Creates copies by using the NTFS short file or directory names. /n is required when you copy files or directories from an NTFS volume to a FAT volume or when the FAT file system naming convention (that is, 8.3 characters) is required on the <i>Destination</i> file system. The <i>Destination</i> file system can be FAT or NTFS. |
| /o | Copies file ownership and discretionary access control list (DACL) information. |
| /x | Copies file audit settings and system access control list (SACL) information (implies /o). |

| PARAMETER | DESCRIPTION |
|---|---|
| /exclude:FileName1[+[FileName2]][+[FileName3]()] | Specifies a list of files. At least one file must be specified. Each file will contain search strings with each string on a separate line in the file. When any of the strings match any part of the absolute path of the file to be copied, that file will be excluded from being copied. For example, specifying the string obj will exclude all files underneath the directory obj or all files with the .obj extension. |
| /y | Suppresses prompting to confirm that you want to overwrite an existing destination file. |
| /-y | Prompts to confirm that you want to overwrite an existing destination file. |
| /z | Copies over a network in restartable mode. |
| /b | Copies the symbolic link instead of the files. This parameter was introduced in Windows Vista®. |
| /j | Copies files without buffering. Recommended for very large files. This parameter was added in Windows Server 2008 R2. |
| /? | Displays help at the command prompt. |

Remarks

- Using **/z**

If you lose your connection during the copy phase (for example, if the server going offline severs the connection), it resumes after you reestablish the connection. **/z** also displays the percentage of the copy operation completed for each file.

- Using **/y** in the COPYCMD environment variable.

You can use **/y** in the COPYCMD environment variable. You can override this command by using **/-y** on the command line. By default, you are prompted to overwrite.

- Copying encrypted files

Copying encrypted files to a volume that does not support EFS results in an error. Decrypt the files first or copy the files to a volume that does support EFS.

- Appending files

To append files, specify a single file for destination, but multiple files for source (that is, by using wildcards or file1+file2+file3 format).

- Default value for *Destination*

If you omit *Destination*, the **xcopy** command copies the files to the current directory.

- Specifying whether *Destination* is a file or directory

If *Destination* does not contain an existing directory and does not end with a backslash (), the following message appears:

Does <Destination> specify a file name or directory name on the target(F = file, D = directory)?

Press F if you want the file or files to be copied to a file. Press D if you want the file or files to be copied to a directory.

You can suppress this message by using the **/i** command-line option, which causes **xcopy** to assume that the destination is a directory if the source is more than one file or a directory.

- Using the **xcopy** command to set archive attribute for *Destination* files

The **xcopy** command creates files with the archive attribute set, whether or not this attribute was set in the source file. For more information about file attributes and **attrib**, see [Additional references](#).

- Comparing **xcopy** and **diskcopy**

If you have a disk that contains files in subdirectories and you want to copy it to a disk that has a different format, use the **xcopy** command instead of **diskcopy**. Because the **diskcopy** command copies disks track by track, your source and destination disks must have the same format. The **xcopy** command does not have this requirement. Use **xcopy** unless you need a complete disk image copy.

- Exit codes for **xcopy**

To process exit codes returned by **xcopy**, use the **ErrorLevel** parameter on the **if** command line in a batch program. For an example of a batch program that processes exit codes using **if**, see [Additional references](#). The following table lists each exit code and a description.

| EXIT CODE | DESCRIPTION |
|-----------|--|
| 0 | Files were copied without error. |
| 1 | No files were found to copy. |
| 2 | The user pressed CTRL+C to terminate xcopy . |
| 4 | Initialization error occurred. There is not enough memory or disk space, or you entered an invalid drive name or invalid syntax on the command line. |
| 5 | Disk write error occurred. |

Examples

To copy all the files and subdirectories (including any empty subdirectories) from drive A to drive B, type:

```
xcopy a: b: /s /e
```

To include any system or hidden files in the previous example, add the **/h** command-line option as follows:

```
xcopy a: b: /s /e /h
```

To update files in the \Reports directory with the files in the \Rawdata directory that have changed since December 29, 1993, type:

```
xcopy \rawdata \reports /d:12-29-1993
```

To update all the files that exist in \Reports in the previous example, regardless of date, type:

```
xcopy \rawdata \reports /u
```

To obtain a list of the files to be copied by the previous command (that is, without actually copying the files), type:

```
xcopy \rawdata \reports /d:12-29-1993 /l > xcopy.out
```

The file xcopy.out lists every file that is to be copied.

To copy the \Customer directory and all subdirectories to the directory \\Public\Address on network drive H:, retain the read-only attribute, and be prompted when a new file is created on H:, type:

```
xcopy \customer h:\public\address /s /e /k /p
```

To issue the previous command, ensure that **xcopy** creates the \Address directory if it does not exist, and suppress the message that appears when you create a new directory, add the **/i** command-line option as follows:

```
xcopy \customer h:\public\address /s /e /k /p /i
```

You can create a batch program to perform **xcopy** operations and use the batch **if** command to process the exit code if an error occurs. For example, the following batch program uses replaceable parameters for the **xcopy** source and destination parameters:

```
@echo off
rem COPYIT.BAT transfers all files in all subdirectories of
rem the source drive or directory (%1) to the destination
rem drive or directory (%2)
xcopy %1 %2 /s /e
if errorlevel 4 goto lowmemory
if errorlevel 2 goto abort
if errorlevel 0 goto exit
:lowmemory
echo Insufficient memory to copy files or
echo invalid drive or command-line syntax.
goto exit
:abort
echo You pressed CTRL+C to end the copy operation.
goto exit
:exit
```

To use this batch program to copy all files in the C:\Prgmcode directory and its subdirectories to drive B, type:

```
copyit c:\prgmcode b:
```

The command interpreter substitutes **C:\Prgmcode** for %1 and **B:** for %2, then uses **xcopy** with the **/e** and **/s** command-line options. If **xcopy** encounters an error, the batch program reads the exit code and goes to the label indicated in the appropriate **IF ERRORLEVEL** statement, then displays the appropriate message and exits from the batch program.

Additional references

- [Copy](#)
- [Move](#)
- [Dir](#)
- [Attrib](#)
- [Diskcopy](#)
- [If](#)
- [Command-Line Syntax Key](#)