



# **system node commands**

## ONTAP 9.3 commands

NetApp  
September 27, 2022

# Table of Contents

system node commands	1
system node halt	1
system node migrate-root	2
system node modify	3
system node reboot	4
system node rename	5
system node restore-backup	6
system node revert-to	6
system node run-console	7
system node run	8
system node show-discovered	10
system node show-memory-errors	11
system node show	13
system node autosupport invoke-core-upload	16
system node autosupport invoke-performance-archive	17
system node autosupport invoke-splog	18
system node autosupport invoke	19
system node autosupport modify	20
system node autosupport show	24
system node autosupport check show-details	28
system node autosupport check show	30
system node autosupport destinations show	32
system node autosupport history cancel	33
system node autosupport history retransmit	34
system node autosupport history show-upload-details	35
system node autosupport history show	37
system node autosupport manifest show	40
system node autosupport trigger modify	44
system node autosupport trigger show	46
system node coredump delete-all	48
system node coredump delete	49
system node coredump save-all	50
system node coredump save	50
system node coredump show	51
system node coredump status	54
system node coredump trigger	57
system node coredump upload	60
system node coredump config modify	61
system node coredump config show	62
system node coredump reports delete	63
system node coredump reports show	64
system node coredump reports upload	65
system node coredump segment delete-all	66

system node coredump segment delete	67
system node coredump segment show	67
system node environment sensors show	69
system node external-cache modify	74
system node external-cache show	75
system node firmware download	76
system node hardware nvram-encryption modify	77
system node hardware nvram-encryption show	78
system node hardware tape drive show	79
system node hardware tape library show	80
system node hardware unified-connect modify	82
system node hardware unified-connect show	83
system node image abort-operation	85
system node image get	86
system node image modify	86
system node image show-update-progress	87
system node image show	88
system node image update	89
system node image package delete	91
system node image package show	92
system node image package external-device delete	93
system node image package external-device show	93
system node internal-switch show	94
system node internal-switch dump eeprom	96
system node internal-switch dump port-mapping	97
system node internal-switch dump stat	98
system node power on	99
system node power show	100
system node root-mount create	101
system node root-mount delete	102
system node root-mount show	102
system node upgrade-revert show	104
system node upgrade-revert upgrade	106
system node virtual-machine hypervisor modify-credentials	107
system node virtual-machine hypervisor show-credentials	108
system node virtual-machine hypervisor show	110
system node virtual-machine instance show-system-disks	113
system node virtual-machine instance show	116

# system node commands

## system node halt

Shut down a node

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### Description

The `system node halt` command stops all activity on a node. You may supply a reason for the shutdown, which will be stored in the audit log. You may also keep partner nodes from performing storage takeover during the shutdown.

### Parameters

#### **-node {<nodename>|local} - Node**

Use this mandatory parameter to specify the node that you want to shut down. The value `local` specifies the current node.

#### **[-reason <text>] - Reason for Shutdown**

Use this parameter to enter a brief note to indicate the reason for the restart, which will be stored in the audit log. Providing this information assists support personnel with troubleshooting efforts.

#### **[-f, -inhibit-takeover <true>] - Disallow Storage Takeover by Partner**

This parameter optionally forces the shutdown and prevents storage failover. In a two-node MetroCluster configuration, this parameter prevents automatic unplanned switchover.



If `-inhibit-takeover` is set to `true`, the default behavior as seen with command `storage failover show` -fields onreboot`` is ignored.

If you enter this command without using this parameter, its effective value is `false` and storage takeover is allowed. If you enter this parameter without a value, it is automatically set to `true` and storage takeover is disabled during reboot.

#### **[-d, -dump <true>] - Create a Core Dump**

If this parameter is set to `true`, it forces a dump of the kernel core when halting the node.

#### **[-skip-lif-migration-before-shutdown <true>] - Skip Migrating LIFs Away from Node Prior to Shutdown**

If this parameter is specified, LIF migration prior to the shutdown will be skipped. However if LIFs on this node are configured for failover, those LIFs may still failover after the shutdown has occurred. The default is to migrate LIFs prior to the shutdown. In the default case, the command attempts to synchronously migrate data and cluster management LIFs away from the node prior to shutdown. If the migration fails or times out, the shutdown will be aborted.

#### **[-ignore-quorum-warnings <true>] - Skip Quorum Check Before Shutdown**

If this parameter is specified, quorum checks will be skipped prior to the shutdown. The operation will continue even if there is a possible data outage due to a quorum issue.

## Examples

The following example shuts down the node named cluster1 for hardware maintenance:

```
cluster1::> system halt -node cluster1 -reason 'hardware maintenance'
```

## Related Links

- [storage failover show](#)

# system node migrate-root

Start the root aggregate migration on a node

**Availability:** This command is available to *cluster* administrators at the *advanced* privilege level.

## Description

The `system node migrate-root` command migrates the root aggregate of a node to a different set of disks. You need to specify the node name and the list of disks on which the new root aggregate will be created. The command starts a job that backs up the node configuration, creates a new aggregate, set it as new root aggregate, restores the node configuration and restores the names of original aggregate and volume. The job might take as long as a few hours depending on time it takes for zeroing the disks, rebooting the node and restoring the node configuration.

## Parameters

**-node {<nodename>|local} - Node**

Specifies the node that owns the root aggregate that you wish to migrate. The value *local* specifies the current node.

**{ -disklist <disk path name>,... - List of Disks for New Root Aggregate**

Specifies the list of disks on which the new root aggregate will be created. All disks must be spares and owned by the same node. Minimum number of disks required is dependent on the RAID type.

**-raid-type {raid\_tec|raid\_dp|raid4} - RAID Type for the New Root Aggregate**

Specifies the RAID type of the root aggregate. The default value is *raid-dp*.

**| -resume <true> - Resume a Failed Migrate Operation }**

Resumes a failed migrate-root operation if the `new_root` aggregate is created and the old root aggregate is in the retracted state.

## Examples

The command in the following example starts the root aggregate migration on node node1:

```
cluster1::> system node migrate-root -node node1 -disklist
1.11.8,1.11.9,1.11.10,1.11.11,1.11.12 -raid-type raid-dp
```

## system node modify

### Modify node attributes

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### Description

The `system node modify` command sets the attributes of a node.

The owner, location, and asset tag attributes are informational only, and do not affect the operation of the node or the cluster. The cluster eligibility attribute marks a node as eligible to participate in a cluster. The epsilon attribute marks a node as the tie-breaker vote if the cluster has an even number of nodes.

Any field of type `<text>` may be set to any text value. However, if the value contains spaces or other special characters, you must enter it using double-quotes as shown in the example below.

Use the [system node show](#) command to display the field values that this command modifies.

### Parameters

#### **-node {<nodename>|local} - Node**

This mandatory parameter specifies which node will have its attributes modified. The value "local" specifies the current node.

#### **[-owner <text>] - Owner**

This optional text string identifies the node's owner. Fill it in as needed for your organization.

#### **[-location <text>] - Location**

Use this text string to identify the physical location of the node. This text string is optional; fill it in as needed for your organization.

#### **[-assettag <text>] - Asset Tag**

If your organization uses asset tags to track equipment, you can use this text string to store that tag's value.

#### **[-eligibility {true|false}] - Eligibility**

This parameter specifies whether the node is eligible to participate in a cluster. If you modify another node's eligibility to false, it will no longer be visible from other nodes in the cluster. If you modify the local node's eligibility to false, the node will no longer be active in the cluster and you will not be able to see any cluster nodes from it.

#### **[-epsilon {true|false}] - Epsilon**

If specified as true for a node, this value designates the specified node as epsilon for this cluster. In a cluster, only one node can be designated as epsilon at any given time. A node can be designated as Epsilon to add weight to its voting in a cluster with an even number of nodes.

### **[`-skip-quorum-check-before-ineligible <true>`] - Skip Quorum Check Before Setting Node Ineligible**

If this parameter is specified, quorum checks will be skipped prior to setting a node ineligible. When setting a node to ineligible, the operation will continue even if there is a possible data outage due to a quorum issue.

### **[`-is-diff-svcs {true|false}`] - Differentiated Services**

If set to `true` this means that the specified node and its HA partner is part of differentiated services storage infrastructure. The default value for this setting is `false`.

## Examples

The following example modifies the attributes of a node named `node0`. The node's owner is set to "IT" and its location to "Data Center 2."

```
cluster1::> system node modify -node node0 -owner "IT" -location "Data Center 2"
```

## Related Links

- [system node show](#)

# system node reboot

## Reboot a node

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description

The `system node reboot` command restarts a node. You can supply a reason for the reboot, which is stored in the audit log. You can also keep partner nodes from performing storage takeover during the reboot and instruct the rebooted node to create a core dump.

## Parameters

### **`-node {<nodename>|local}` - Node**

Specifies the node that is to be restarted. The value "local" specifies the current node.

### **[`-f, -inhibit-takeover <true>`] - Disallow Storage Takeover by Partner**

If set to `true`, this parameter specifies that the node's failover partner is not allowed to take over for the node when the node is rebooted. In a two-node MetroCluster configuration, this parameter prevents automatic unplanned switchover. If you enter this command without using this parameter, its effective value is `false` and storage takeover is allowed. If you enter this parameter without a value, it is automatically set to `true` and storage takeover is disabled during reboot.

### **[`-reason <text>`] - Reason for Reboot**

Use this parameter to enter a brief note to indicate the reason for the restart, which will be stored in the audit log. Providing this information assists support personnel with troubleshooting efforts.

### **[-d, -dump <true>] - Create a Core Dump**

If you would like the node to create a core dump before restarting, specify the `true` value with this parameter. If you enter this command without using this parameter, its effective value is `false` and the node doesn't create a core dump. If you enter this parameter without a value, it is automatically set to `true` and the node creates a core dump.

### **[-skip-lif-migration-before-reboot <true>] - Skip Migrating LIFs Away from Node Prior to Reboot**

If this parameter is specified, LIF migration prior to the reboot will be skipped. However if LIFs on this node are configured for failover, those LIFs may still failover after the reboot has occurred. The default is to migrate LIFs prior to the reboot. In the default case, the command attempts to synchronously migrate data and cluster management LIFs away from the node prior to reboot. If the migration fails or times out, the reboot will be aborted.

### **[-ignore-quorum-warnings <true>] - Skip Quorum Check Before Reboot**

If this parameter is specified, quorum checks will be skipped prior to the reboot. The operation will continue even if there is a possible data outage due to a quorum issue.

## **Examples**

The command in the following example restarts the node named `cluster1` for a software upgrade:

```
cluster1::> system node reboot -node cluster1 -reason "software upgrade"
```

## **system node rename**

Rename a node

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### **Description**

The `system node rename` command changes a node's name. Both the node to be modified and the new name of that node must be specified with the following parameters. This command is best executed from the node that is being renamed, using the `-node local` parameter.

Use the [system node show](#) command to display the names of all the nodes in the current cluster.

### **Parameters**

#### **-node {<nodename>|local} - Node**

This parameter specifies which node you are renaming. The value `local` specifies the current node.

#### **-newname <text> - New Name**

Use this parameter to specify the new name of the node.

- The name must contain only the following characters: A-Z, a-z, 0-9, "-" or "\_".
- The first character must be one of the following characters: A-Z or a-z.



- The last character must be one of the following characters: A-Z, a-z or 0-9.
- The maximum supported length is 47 characters.
- The system reserves the following names: "all", "cluster", "local" and "localhost".

## Examples

The following example changes the name of the node named node3 to node4.

```
cluster1::> system node rename -node node3 -newname node4
```

## Related Links

- [system node show](#)

# system node restore-backup

Restore the original backup configuration to the HA target node

**Availability:** This command is available to *cluster* administrators at the *advanced* privilege level.

## Description

The `system node restore-backup` command restores the backup configuration file that is stored on the partner node to the specified target node in an HA pair. The backup configuration file is restored after Data ONTAP has been installed on the target node.

The backup configuration file is stored on the HA partner node while the target node is down. After the target node has been installed, the partner node sends this backup configuration file to the target node through the management network by using the `system node restore-backup` command to restore the original configuration. This procedure is commonly used when replacing the target node's boot device.

The target IP address should be the address of the target node used for netboot installation.

## Parameters

**-node {<nodename>|local} - Node**

Specifies the partner node that sends the backup configuration file to the target node. The value "local" specifies the current node.

**-target-address <Remote InetAddress> - HA Partner IP Address**

Specifies the IP address for the target node.

# system node revert-to

Revert a node to a previous release of Data ONTAP

**Availability:** This command is available to *cluster* administrators at the *advanced* privilege level.

## Description

The `system node revert-to` command reverts a node's cluster configuration to the given version. After the `system node revert-to` command has finished, the `revert_to` command must be run from the nodeshell. The `revert_to` command reverts the filesystem on individual nodes to the target release. Before running `revert-to` in the cluster shell, the target release must be installed on the node.

## Parameters

### `-node {<nodename>|local}` - Node

Specifies the node that is to be reverted. The value `local` specifies the current node.

### `-version <revert version>` - Data ONTAP Version

Specifies the version of Data ONTAP to which the node is to be reverted.

### `[-check-only <true>]` - Capability Check

If set to `true`, this parameter specifies that the cluster configuration revert should perform checks to verify all of the preconditions necessary for `revert-to` to complete successfully. Setting the parameter to `true` does not run through the actual revert process. By default this option is set to `false`.

## Examples

The command in the following example reverts cluster configuration of a node named `node1` to Data ONTAP version 9.2

```
cluster1::*> system node revert-to -node node1 -version 9.2
```

## system node run-console

Access the console of a node

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description

This command allows you to access the console of any remote node on the same cluster. The remote access is helpful in situations where the node cannot be booted up or has network issues. This command establishes an SSH session with the Service Processor of a remote node and accesses that node's console over the serial channel. This command works even if Data ONTAP is not booted up on the remote node. You can get back to the original node by pressing Ctrl+D. This command works only on SSH sessions and not on physical console sessions.

## Parameters

### `-node {<nodename>|local}` - Node

This parameter specifies the node whose physical console you want to access.

## Examples

The following example accesses the console of node2 in the same cluster.

```
cluster1::~> system node run-console -node node2
Pressing Ctrl-D will end this session and any further sessions you might
open on top of this session.
Type Ctrl-D.
SP-login: admin
Password:
*****
* This is an SP console session. Output from the      *
* serial console is also mirrored on this session.  *
*****
node2::~>
node2::~> Connection to 192.168.1.202 closed.

cluster1::~>
```

## system node run

Run interactive or non-interactive commands in the nodeshell

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### Description

Use the `system node run` command to run certain commands from the nodeshell CLI on a specific node in the cluster. You can run a single nodeshell command from the clustershell that returns immediately, or you can start an interactive nodeshell session from which you can run multiple nodeshell commands.

Nodeshell commands are useful for root volume management and system troubleshooting. Commands that are available through the nodeshell are scoped to a single node in the cluster. That is, they affect only the node specified by the value of the `-node` parameter and do not operate on other nodes in the cluster. To see a list of available nodeshell commands, type `'?'` at the interactive nodeshell prompt. For more information on the meanings and usage of the available commands, use the `man` command in the nodeshell.

Only one interactive nodeshell session at a time can be run on a single node. Up to 24 concurrent, non-interactive sessions can be run at a time on a node.

When running the nodeshell interactively, exit the nodeshell and return to the clustershell by using the `exit` command. If the nodeshell does not respond to commands, terminate the nodeshell process and return to the clustershell by pressing Ctrl-D.

The `system node run` command is not available from the GUI interface.



An alternate way to invoke the `system node run` command is by typing the `run` as a single word.

## Parameters

### **-node {<nodename>|local} - Node**

Use this parameter to specify the name of the node on which you want to run the nodeshell command. If you specify only this parameter, the command starts an interactive nodeshell session that lasts indefinitely. You can exit the nodeshell to the clustershell by pressing Ctrl-D or by typing the `exit` command.

### **{ [-command <text>, ...] - Command to Run**

This optionally specifies the name of a single nodeshell command to run on the specified node. To see a list of available nodeshell commands, type '?' at an interactive nodeshell prompt.

### **| [-reset <>true>] - Reset Existing Connection }**

If this parameter is specified with the `true` value, it terminates any existing interactive nodeshell session on the specified node. The default value is `false`.

## Examples

The following example runs the nodeshell command `sysconfig -V` on a node named `node1`:

```
cluster1::> system node run -node node1 -command sysconfig -V
volume node1_aggr0 (1 RAID group):
    group 0: 3 disks
```

The following example starts a nodeshell session on a node named `node2` and then runs the nodeshell `sysconfig -V` command. The system remains in the nodeshell after running the `sysconfig -V` command.

```
cluster1::> run -node node2
Type 'exit' or 'Ctrl-D' to return to the CLI
node2> sysconfig -V
volume node2_aggr0 (1 RAID group):
    group 0: 3 disks
node2>
```

The following example starts a nodeshell session on a node named `node1` and then runs two nodeshell commands, `aggr status` first and `vol status` second. Use quotation marks and semicolons when executing multiple nodeshell commands with a single `run` command.

```

cluster1::> run -node node1 -command "aggr status; vol status"
    Aggr State           Status           Options
    aggr0 online         raid_dp, aggr   root
                        parity uninit'd!
                        32-bit
    aggr1 online         raid_dp, aggr   parity uninit'd!
                        32-bit

    Volume State         Status           Options
    vol0 online          raid_dp, flex    root, nvfail=on
                        parity uninit'd!
    root_vs0 online      raid_dp, flex    create_ucose=on,
                        cluster                convert_ucose=on,
                        parity uninit'd!      maxdirsize=102400

```

## system node show-discovered

Display all nodes discovered on the local network

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### Description

The `system node show-discovered` command displays information about all the detectable nodes on the local cluster network. This includes both nodes in a cluster and nodes that do not belong to a cluster. You can filter the output to show only nodes that do not belong to a cluster or nodes that are in a cluster.

To see a list of values that are in use for a particular field, use the `-fields` parameter of this command with the list of field names you wish to view.

### Parameters

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>, ...` parameter, the command only displays the fields that you specify.

**| [-instance ] }**

If the `-instance` parameter is specified, the command displays detailed information about each node.

**[-node <text>] - Node Name**

This parameter specifies the name of a node for which information is to be displayed. If this parameter is not specified, the command displays information about all discovered nodes.

**[-is-in-cluster {true|false}] - Is in a Cluster**

If this parameter is set to `false`, the command lists only nodes that do not belong to a cluster.

**[-cluster-uuid <UUID>] - Cluster UUID**

Displays information about nodes belonging to the cluster that has the UUID you specify.

**[-cluster-name <text>] - Cluster Name**

Displays information about nodes belonging to the cluster that has the name you specify.

**[-serial-number <text>] - Node Serial Number**

Displays information about the node that has the serial number you specify.

**[-addresses <IP Address>,...] - Cluster IP Addresses**

Displays information about the node that has the cluster IP addresses you specify.

**[-netmask <IP Address>] - Cluster Address Mask**

Displays information about the nodes that have the netmask address you specify.

**[-nvramid <nvramid>] - Node NVRAM ID**

Displays information about the node that has the NVRAM ID you specify.

**[-partner-nvramid <nvramid>] - Partner NVRAM ID**

Displays information about the node that has an HA partner with the NVRAM ID you specify.

**[-model <text>] - Model**

Displays the nodes that have the specified model number.

**[-version <text>] - Software Version**

Displays the nodes that have the specified version of Data ONTAP.

## Examples

The following example displays information about all discovered nodes in the cluster network:

```
cluster1::*> system node show-discovered
Node           Cluster      Addresses           NVRAM ID           Partner NVRAM
-----
4069114-60-0   -            169.254.232.178    4069114600         -
4069114-60-2   -            169.254.79.38     4069114602         -
4069114-60-3   -            169.254.195.76    4069114603         -
cluster1-01    cluster1     169.254.140.39    4069114628         4069114629
cluster1-02    cluster1     169.254.138.137   4069114629         4069114628
```

## system node show-memory-errors

Display Memory Errors on DIMMs

**Availability:** This command is available to *cluster* administrators at the *advanced* privilege level.

## Description

`system node show-memory-errors` prints the history of memory (storage controller's RAM) errors since boot. This command can be useful in diagnosing memory problems or determining which DIMM, if any, might need replacement. Some correctable ECC errors are to be expected under normal operation, but many occurring on a particular DIMM might indicate a problem. All the fields are read only and can be used to filter the output. The maximum number of physical address and timestamps reported is 160.

## Parameters

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>, ...` parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-verbose ]**

The `-verbose` parameter enables verbose mode, resulting in the display of more detailed output.

**| [-instance ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[-node {<nodename>|local}] - Node**

When provided, the `-node` parameter specifies the nodes for which the memory error statistics are to be displayed. When the `-node` is not provided, the command is applied to all the nodes in the cluster.

**[-id <integer>] - DIMM ID**

This parameter refers to the DIMM ID. It can be used to look at the correctable ECC error count on a specific DIMM.

**[-name <text>] - DIMM Name**

This parameter specifies the DIMM name for which the memory error statistics are to be displayed.

**[-cecc <integer>] - Correctable ECC Error Count**

This parameter can be used to get all the DIMMs with the specified correctable ECC error count.

**[-merr {true|false}] - Multiple Errors on Same Address**

Use this parameter with the values `true` to specify whether the error was seen multiple times on the same physical address. It can also be used to look at all the DIMMs with multiple errors on same address.

**[-timestamp <text>,...] - Error Time**

This specifies the time at which the error was seen on the DIMM.

**[-addr <text>,...] - Error Address**

This specifies the physical address on which the error was seen.

## Examples

```

cluster1::*> system node show-memory-errors
  Correctable ECC Memory Errors:
Node: localhost
DIMM      CECC  Multiple Err
  Name      Count  Same Address
  -----  -
DIMM-1      0  false
DIMM-2      0  false
DIMM-3      0  false
DIMM-4      0  false
DIMM-5      4  true
DIMM-6      1  false
DIMM-7      1  false
DIMM-8      0  false
  8 entries were displayed.
cluster1::*> system node show-memory-errors -verbose
  Correctable ECC Memory Errors:
Node: localhost
DIMM      CECC  Multiple Err          Physical
  Name      Count  Same Address  Timestamp          Address
  -----  -
DIMM-1      0  false          -          -
DIMM-2      0  false          -          -
DIMM-3      0  false          -          -
DIMM-4      0  false          -          -
DIMM-5      4  true          12/02/2013 08:17:43    0xD640
              12/02/2013 08:17:57    0x3F7FF800
              12/02/2013 08:18:03    0x11743D000
              12/02/2013 08:18:37    0x11743D000
DIMM-6      1  false          12/02/2013 08:17:53    0x87EC0
DIMM-7      1  false          12/02/2013 08:17:51    0x13DED8900
DIMM-8      0  false          -          -
  8 entries were displayed.

```

## system node show

Display the list of nodes in the cluster

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### Description

The `system node show` command displays information about the nodes in a cluster. You can limit output to specific types of information and specific nodes in the cluster, or filter output by specific field values.

To see a list of values that are in use for a particular field, use the `-fields` parameter of this command with



the list of field names you wish to view. Use the [system node modify](#) command to change some of the field values that this command displays.

## Parameters

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>, ...` parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-inventory ]**

Use this parameter to display inventory information such as serial numbers, asset tags, system identifiers, and model numbers.

**| [-messages ]**

Use this parameter to display system messages for each node.

**| [-instance ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[-node {<nodename>|local}] - Node**

Selects information for node names that match this parameter value.

**[-owner <text>] - Owner**

Selects nodes that have owner values that match this parameter value.

**[-location <text>] - Location**

Selects nodes at specific physical locations that match this parameter value.

**[-model <text>] - Model**

Selects nodes that have model numbers that match this parameter value.

**[-serialnumber <text>] - Serial Number**

Selects nodes that have serial numbers that match this parameter value.

**[-assettag <text>] - Asset Tag**

Selects nodes that have asset tags that match this parameter value.

**[-uptime {<seconds>| [<d> days] <hh>:<mm>[:<ss>]}] - Uptime**

Selects nodes that have uptimes that match this parameter value. This parameter is most useful when used with a range indicator such as less than or greater than, as in:

```
show -uptime >"30 days 00:00"
```

**[-nvramid <nvramid>] - NVRAM System ID**

Selects nodes that have NVRAM system IDs that match this parameter value.

### **[-systemid <text>] - System ID**

Selects nodes that have system IDs that match this parameter value.

### **[-vendor <text>] - Vendor**

Selects nodes that have vendor names that match this parameter value.

### **[-health {true|false}] - Health**

Selects nodes that have health values that match this parameter value. Specify `true` to display healthy nodes, and `false` to display unhealthy nodes.

### **[-eligibility {true|false}] - Eligibility**

Selects nodes that have voting eligibility values that match this parameter value.

### **[-epsilon {true|false}] - Epsilon**

Selects nodes that have epsilon holding designations that match this parameter value. This is useful to find out which node, if any, in the current cluster has been designated as epsilon. Specify `true` to display the node holding epsilon, and `false` to display nodes not holding epsilon.

### **[-uuid <UUID>] - UUID**

Selects nodes that have the specified universal unique identifiers that match this parameter value.

### **[-is-diff-svcs {true|false}] - Differentiated Services**

If true, the corresponding node is considered to be part of differentiated services storage infrastructure.

### **[-is-all-flash-optimized {true|false}] - All-Flash Optimized**

Selects nodes that have "All-Flash Optimized" personality values that match this parameter value. Specify `true` to display nodes which support only SSD drives, and `false` to display nodes which support all kinds of drives.

## **Examples**

The following example displays information about all nodes in the cluster:

```
cluster1::> system node show
Node   Health Eligibility Uptime           Model   Owner   Location
-----
node0  true   true           89 days 23:47 MODELXX  IT      Data Center 2
node1  true   true           15 days 22:37 MODELXX           Data Center 2
node2  true   true           15 days 23:00 MODELXX           Data Center 2
node3  true   true           15 days 22:37 MODELXX           Data Center 2
4 entries were displayed.
```

This example displays the locations and model numbers of all nodes that are in physical locations that have names beginning with "Lab":

```
cluster1::> system node show -location lab* -fields location, model
node          location model
-----
node5         Lab 1    MODELXX
node7         Lab 3    MODELXX
node9         Lab 5    MODELXX
```

## Related Links

- [system node modify](#)

# system node autosupport invoke-core-upload

Generate and send an AutoSupport message with an existing core file.

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description

The `system node autosupport invoke-core-upload` command sends two AutoSupport messages. The first AutoSupport message contains the content relevant to this core upload. This AutoSupport message has subject line with prefix "CORE INFO:". The second AutoSupport message contains the core file specified by the "-core-filename" option. This AutoSupport message has subject line with prefix "CORE UPLOAD:". The command requires that the specified file be present while the AutoSupport message is being transmitted.

## Parameters

**-node {<nodename>|local} - Node**

Use this parameter to specify the node from which the AutoSupport message is sent. Defaults to localhost.

**[-message <text>] - Message Included in the AutoSupport Subject**

Use this parameter to specify the text in the subject line of the AutoSupport message.

**[-uri <text>] - Alternate Destination for This AutoSupport**

Use this parameter to send the AutoSupport message to an alternate destination. Only "http" and "https" protocols are supported. If this parameter is omitted, the message is sent to all the recipients defined by the [system node autosupport modify](#) command.

**[-force <>true>] - Generate and Send Even if Disabled**

Use this parameter to generate and send the AutoSupport message even if AutoSupport is disabled on the node.

**[-case-number <text>] - Case Number for This Core Upload**

Use this parameter to specify the optional case number to be associated with this AutoSupport message.

**-core-filename <text> - The Existing Core Filename to Upload**

Use this parameter to specify the core file to be included in the AutoSupport message. Use the [system node coredump show](#) command to list the core files by name.

## Examples

Use this command to list the core files from a node:

```
cluster1::> system node coredump show -node local
Node:Type Core Name                               Saved Panic Time
-----
node:kernel
      core.4073000068.2013-09-11.15_05_01.nz      true  9/11/2013
15:05:01
```

Use this command to invoke an AutoSupport message with the corefile core.4073000068.2013-09-11.15\_05\_01.nz:

```
cluster1::> system node autosupport invoke-core-upload -core-filename
core.4073000068.2013-09-11.15_05_01.nz -node local
```

## Related Links

- [system node autosupport modify](#)
- [system node coredump show](#)

## system node autosupport invoke-performance-archive

Generate and send an AutoSupport message with performance archives.

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### Description

The `system node autosupport invoke-performance-archive` command sends an AutoSupport message with the performance archives from a node. The command requires that the performance archives in the specified date range be present while the AutoSupport message is being transmitted.

### Parameters

**-node {<nodename>|local} - Node**

Use this parameter to specify the node from which the AutoSupport message is sent. The default setting is localhost.

**[-message <text>] - Message Included in the AutoSupport Subject**

Use this parameter to specify the text in the subject line of the AutoSupport message.

**[-uri <text>] - Alternate Destination for This AutoSupport**

Use this parameter to send the AutoSupport message to an alternate destination. Only "file," "http," and "https" protocols are supported. If this parameter is omitted, the message is sent to the all of the recipients

defined by the system node `autosupport modify` command.

**`[-force <true>]` - Generate and Send Even if Disabled**

Use this parameter to generate and send the AutoSupport message even if AutoSupport is disabled on the node.

**`[-case-number <text>]` - Case Number for This Performance Archive Upload**

Use this parameter to specify the optional case number to be associated with this AutoSupport message.

**`-start-date <MM/DD/YYYY HH:MM:SS>` - Start Date for Performance Archive Dataset**

Use this parameter to specify the start date for the files in the performance archive dataset to be included in the AutoSupport message.

**`{ -end-date <MM/DD/YYYY HH:MM:SS>` - End Date for Performance Archive Dataset**

Use this parameter to specify the end date for the files in the performance archive dataset to be included in the AutoSupport message. The end date should be within six hours of the start date.

**`[-duration <[<integer>h] [<integer>m] [<integer>s]>` - Duration of Performance Archive Dataset }**

Use this parameter with `start-date` to specify the duration of the performance archive dataset to be included in the AutoSupport message. The maximum duration limit is six hours from the start date.

## Examples

Use this command to invoke an AutoSupport message to include the performance archives in the given date range:

```
cluster1:> system node autosupport invoke-performance-archive -node local
-start-date 9/12/2013 19:37:24 -end-date 9/12/2013 22:37:24
cluster1:> system node autosupport invoke-performance-archive -node local
-start-date 11/21/2013 13:42:09 -duration 6h
```

## system node autosupport invoke-splog

Generate and send an AutoSupport message with collected service-processor log files

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### Description

The `system node autosupport invoke-splog` command sends an AutoSupport message with the Service Processor log files from a specified node in the cluster.

### Parameters

**`-remote-node {<nodename>|local}` - Node**

Use this parameter to specify the node from which Service Processor log files are to be collected.

### **[`-log-sequence <integer>`] - Log File Sequence Number**

Use this parameter to specify the sequence number of the Service Processor log files to be collected. If this parameter is omitted, the latest Service Processor log files are collected.

### **[`-uri <text>`] - Alternate Destination for This AutoSupport**

Use this parameter to send the AutoSupport message to an alternate destination. Only "file," "http," and "https" protocols are supported. If this parameter is omitted, the message is sent to all of the recipients defined by the [system node autosupport modify](#) command.

### **[`-force <>true>`] - Generate and Send Even if Disabled**

Use this parameter to generate and send the AutoSupport message even if AutoSupport is disabled on the node.

## Examples

Use this command to invoke an AutoSupport message to include the Service Processor log files collected from node cluster1-02.

```
cluster1::> system node autosupport invoke-splug -remote-node cluster1-02
[Job 777] Job succeeded: Log files from the service processor have been
transferred to "/mroot/etc/log/sp/ondemand" on node cluster1-01, and
AutoSupport message has been triggered.

cluster1::>
```

## Related Links

- [system node autosupport modify](#)

# system node autosupport invoke

Generate and send an AutoSupport message

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description

The `system node autosupport invoke` command sends an AutoSupport message from a node.

## Parameters

### **`-node {<nodename>|local}` - Node**

Use this parameter to specify the node from which the AutoSupport message is sent.

### **[`-message <text>`] - Message Included in the AutoSupport Subject**

Use this parameter to specify text sent in the subject line of the AutoSupport message. This parameter is not available when the `-type` parameter is set to performance.

### **-type {test|performance|all} - Type of AutoSupport Collection to Issue**

Use this parameter to specify the type of AutoSupport collection to issue. There is no default; you must specify a `-type` .

- `test` - The message contains basic information about the node. When the AutoSupport message is received by technical support, an e-mail confirmation is sent to the system owner of record. This enables you to confirm that the message is being received by technical support.
- `all` - The message contains all collected information about the node.
- `performance` - The message contains only performance information about the node. This parameter has effect only if performance AutoSupport messages are enabled, which is controlled by the `-perf` parameter of the [system node autosupport modify](#) command.

### **[-uri <text>] - Alternate Destination for This AutoSupport**

Use this parameter to send the AutoSupport message to the destination you specify instead of the configured destination. Only "file", "mailto", "http", and "https" protocols are supported. If this parameter is omitted, the message is sent to the all of the recipients defined by the [system node autosupport modify](#) command.

### **[-force <>true>] - Generate and Send Even if Disabled**

Use this parameter to generate and send the message even if AutoSupport is disabled on the node.

## Examples

The following example sends a test AutoSupport message from a node named node0 with the text "Testing ASUP":

```
cluster1::> system node autosupport invoke -node node0 -type test -message "Testing ASUP"
```

## Related Links

- [system node autosupport modify](#)

# system node autosupport modify

Modify AutoSupport configuration

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description

The `system node autosupport modify` command modifies the AutoSupport configuration of a node.

## Parameters

### **-node {<nodename>|local} - Node**

Use this parameter to specify the node being configured.

### **[-state {enable|disable}] - State**

Use this parameter to specify whether AutoSupport is enabled or disabled on the node. The default setting is `enable`. When AutoSupport is disabled, messages are not sent to anyone, including the vendor's technical support, your internal support organization, or partners.

### **[-mail-hosts <text>,...] - SMTP Mail Hosts**

Use this parameter to specify up to five SMTP mail hosts through which the node sends AutoSupport messages. This parameter is required if you specify e-mail addresses in the `-to`, `-noteto`, or `-partner-address` parameters or if you specify `smtp` in the `-transport` parameter. Separate multiple mail hosts with commas with no spaces in between. The AutoSupport delivery engine attempts to use these hosts for delivery in the order that you specify.

You can optionally specify a port value for each mail server. A port value can be specified on none, all, or some of the mail hosts. The port specification for a mail host consists of a colon (":") and a decimal value between 1 and 65335, and follows the mailhost name (for example, `mymailhost.example.com:5678`). The default port value is `25`.

Also, you can optionally prepend a user name and password combination for authentication to each mail server. The format of the username and password pair is `user1:pw1@mymailhost.example.com`. The username and password can be specified on none, all, or some of the mail hosts.

The default value for this parameter is `mailhost`.

### **[-from <mail address>] - From Address**

Use this parameter to specify the e-mail address from which the node sends AutoSupport messages. The default is `Postmaster@xxx` where `xxx` is the name of the system.

### **[-to <mail address>,...] - List of To Addresses**

Use this parameter to specify up to five e-mail addresses to receive AutoSupport messages that are most relevant for your internal organization. Separate multiple addresses with commas with no spaces in between. For this parameter to have effect, the `-mail-hosts` parameter must be configured correctly. Individual trigger events can change the default usage of this parameter using the `-to` parameter of the [system node autosupport trigger modify](#) command. By default, no list is defined.

### **[-noteto <mail address>,...] - (DEPRECATED) List of Noteto Addresses**



This parameter has been deprecated and might be removed in a future version of Data ONTAP.

Use this parameter to specify up to five addresses to receive a short-note version of AutoSupport messages that are most relevant for your internal organization. Short-note e-mails contain only the subject line of the AutoSupport message, which is easier to view on a mobile device. For this parameter to have effect, the `-mail-hosts` parameter must be configured correctly. Individual trigger events can change the default usage of this parameter using the `-noteto` parameter of the [system node autosupport trigger modify](#) command. By default, no list is defined.

### **[-partner-address <mail address>,...] - List of Partner Addresses**

Use this parameter to specify up to five e-mail addresses to receive all AutoSupport messages including periodic messages. This parameter is typically used for support partners. For this parameter to have effect, the `-mail-hosts` parameter must be configured correctly. By default, no list is defined.



### **[`-support {enable|disable}`] - Send AutoSupport Messages to Vendor Support**

Use this parameter to specify whether to send all AutoSupport messages to your vendor's technical support. (Destination information is pre-defined and does not require configuration.) When `-state` is enabled and `-support` is disabled, messages are sent to the addresses specified in the `-to`, `-noteto`, or `-partner-address` parameters but are not sent to your vendor's technical support. The default is `enable`.

### **[`-transport {smtp|http|https}`] - Protocol to Contact Support**

Use this parameter to specify the protocol used to deliver AutoSupport messages to your vendor's technical support. This parameter applies only when the `-support` parameter is set to `enable`. If you specify `http` or `https` and your network uses a proxy, you must also set the `-proxy-url` parameter. If you specify `smtp`, you must also configure the `-mail-hosts` parameter.

### **[`-proxy-url <text>`] - Support Proxy URL**

Use this parameter to specify an HTTP or HTTPS proxy if the `-transport` parameter is set to `HTTP` or `HTTPS` and your organization uses a proxy. Enter the URL without an `http://` or `https://` prefix. If authentication is required, use the format "`[username]@[host][:[port]]`". You will be prompted for the password. The default is an empty string. To specify a proxy that contains a question mark, press `ESC` followed by the `"?"`. This field can be cleared by setting the value to an empty string using two double quotes (`""`).

### **[`-hostname-subj {true|false}`] - Hostname Subject**

Use this parameter to specify whether the hostname of the node is included in the subject line of the AutoSupport message. The default is `false`. This parameter applies only if the `-remove-private-data` parameter is `true`.

### **[`-nht {true|false}`] - NHT Enable**

Use this parameter to specify whether NHT disk drive health data is sent to technical support and addresses specified in the `-partner-address` parameter when disk drives fail. The default is `true`.

### **[`-perf {true|false}`] - Performance Data Enable**

Use this parameter to specify whether performance data is sent to technical support and addresses specified in the `-partner-address` parameter. The default is `true`.

### **[`-retry-interval <[<integer>h] [<integer>m] [<integer>s]>`] - Retry Interval**

Use this parameter to specify the amount of time to delay before trying to send an AutoSupport message again after a sending failure. Values may end with `"s"`, `"m"`, or `"h"` to indicate seconds, minutes, or hours, respectively. The minimum interval is 30 seconds and the maximum is 1 day. The default is 4 minutes.

### **[`-retry-count <integer>`] - Retry Count**

Use this parameter to specify the number of times to try resending mail before dropping it. The minimum number is 5 and the maximum is 30. The default is 15 times.

### **[`-reminder {true|false}`] - Reminder Enable**

Use this parameter to enable or disable a reminder message that is sent when AutoSupport is not configured to send messages to technical support. This reminder is logged as an EMS event called `"autosupport.general.reminder"` every 24 hours. The default is `true`.

### **[`-max-http-size {<integer>[KB|MB|GB|TB|PB]}`] - Maximum HTTP Size**

Use this parameter to specify the maximum file size (in bytes by default, but can also be specified in KB, MB, TB or PB) for HTTP and HTTPS transfers. This parameter applies only to messages sent to technical

support and only if the `-transport` parameter is set to HTTP or HTTPS. Setting the value to 0 disables the delivery size budget. The default is 50 MB and the minimum supported size is 2 MB.

If the size of the AutoSupport message exceeds this value, AutoSupport will deliver as much of the message as possible. You can use the "system node autosupport manifest show" command to identify the sections of the message that AutoSupport sent. AutoSupport collects and sends the content in order of priority. The priority is predefined for each AutoSupport message. To identify the collection order for an AutoSupport trigger, use the "system node autosupport trigger show" command with the `-instance` parameter.

#### **[`-max-smtp-size` {<integer>[KB|MB|GB|TB|PB]}] - Maximum SMTP Size**

Use this parameter to specify the maximum file size (in bytes by default, but can also be specified in KB, MB, TB or PB) for SMTP (e-mail) transfers. This parameter applies to messages sent to the addresses specified in the `-to`, `-noteto`, and `-partner-address` parameters. If the `-transport` parameter is set to `smtp`, this parameter also applies to messages sent to the vendor's technical support. Setting the value to 0 disables the delivery size budget. The default is 5 MB and the minimum supported size is 2 MB.

If the size of the AutoSupport message exceeds this value, AutoSupport will deliver as much of the message as possible. You can use the "system node autosupport manifest show" command to identify the sections of the message that AutoSupport sent. AutoSupport collects and sends the content in order of priority. The priority is predefined for each AutoSupport message. To identify the collection order for an AutoSupport trigger, use the "system node autosupport trigger show" command with the `-instance` parameter.

#### **[`-remove-private-data` {true|false}] - Remove Sensitive Data**

Use this parameter with the value `true` to remove, encode, or mask sensitive data from AutoSupport attachments and headers. Use this feature to eliminate private data from all AutoSupport messages.

Eliminated data might include: IP addresses, MAC addresses, URIs, DNS names, e-mail addresses, port numbers, node names, Vserver names, cluster names, aggregate names, volume names, junction paths, policy names, user IDs, group IDs, LUNs, and qtree names.

The default is `false`.



Changing this value from `false` to `true` deletes the AutoSupport history and all files associated with it.

#### **[`-validate-digital-certificate` {true|false}] - Validate Digital Certificate Received**

Use this parameter with the value `true` to force the node to validate digital certificates that it receives. The default is `true`. When this value is `true` the certificate might be validated by OCSP. The OCSP validation for these certificates is controlled by [security config ocsd enable -app autosupport](#) and [security config ocsd disable -app autosupport](#).

#### **[`-ondemand-state` {enable|disable}] - AutoSupport OnDemand State**

Use this parameter to specify whether the AutoSupport OnDemand feature is enabled or disabled on the node. The default is `enable`. When AutoSupport OnDemand is enabled, support personnel can remotely trigger new AutoSupport messages, re-send existing AutoSupport messages and decline the delivery of unwanted AutoSupport messages. When this option is disabled, this node will not respond to any AutoSupport OnDemand requests from support personnel.

#### **[`-ondemand-remote-diagnostics-state` {enable|disable}] - AutoSupport OnDemand Remote Diagnostics State**

Use this parameter to specify whether the AutoSupport OnDemand Remote Diagnostics feature is enabled or disabled on the node. The default is `enable`. When AutoSupport OnDemand Remote Diagnostics is enabled, support personnel can remotely trigger new AutoSupport messages on this node to gather information during troubleshooting scenarios. When this option is disabled, support personnel will still be able to re-send existing AutoSupport messages that may not have been transmitted correctly.

## Examples

The following example enables AutoSupport on a node named `node3` with the following settings:

- SMTP mail host named `smtp.example.com`.
- E-mail "from" address of [alerts@node3.example.com](mailto:alerts@node3.example.com)
- E-mail "to" address of [support@example.com](mailto:support@example.com)
- AutoSupport messages sent to support personnel
- HTTPS set as transport protocol
- Short-note address of [pda@example.com](mailto:pda@example.com)
- If sending fails, the system will wait 23 minutes before retrying.

```
cluster1::> system node autosupport modify -node node3 -state enable -mail
-mail-hosts smtp.example.com -from alerts@node3.example.com -to
support@example.com -support enable -transport https -noteto
pda@example.com -retry-interval 23m
```

The following examples show how to modify AutoSupport URLs when using IPv6 address literals:

```
cluster1::> system node autosupport modify -node node1 -mail-hosts
[2620:10a:4002:6004::bbbb]:25
```

```
cluster1::> system node autosupport modify -node node1 -proxy-url
username:password@[2620:10a:4002:6004::bbbb]:8080
```

## Related Links

- [system node autosupport trigger modify](#)
- [security config oosp enable](#)
- [security config oosp disable](#)

## system node autosupport show

Display AutoSupport configuration

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description

The `system node autosupport show` command displays the AutoSupport configuration of one or more nodes.

## Parameters

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>, ...` parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-config ]**

Use this parameter to display the retry interval, retry count, throttle, and reminder settings of all nodes in the cluster.

**| [-nht-performance ]**

Use this parameter to display NHT and performance information about all nodes in the cluster.

**| [-recent ]**

Use this parameter to display the subject and time of the last AutoSupport message generated by each node in the cluster.

**| [-support-http ]**

Use this parameter to display whether HTTP support is enabled for each node in the cluster, and identify the transport protocol and the support proxy URL used by each node.

**| [-support-smtp ]**

Use this parameter to display whether SMTP (e-mail) support is enabled for each node in the cluster, and identify the transport protocol and the "to" mail address used by each node.

**| [-instance ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[-node {<nodename>|local}] - Node**

Use this parameter to display detailed information about the node you specify.

**[-state {enable|disable}] - State**

Use this parameter to display information only about nodes that have the AutoSupport state you specify.

**[-mail-hosts <text>,...] - SMTP Mail Hosts**

Use this parameter to display information only about nodes that use the mail hosts you specify.

**[-from <mail address>] - From Address**

Use this parameter to display information only about nodes that have the "from" e-mail address you specify.

**[-to <mail address>,...] - List of To Addresses**

Use this parameter to display information only about nodes that have the "to" e-mail addresses you specify.

## **[-noteto <mail address>,...] - (DEPRECATED) List of Noteto Addresses**



This parameter has been deprecated and might be removed in a future version of Data ONTAP.

Use this parameter to display information only about nodes that send short-note e-mail messages to the e-mail addresses you specify. Short-note e-mails contain only the subject line of the AutoSupport message, which is easier to view on a mobile device.

## **[-partner-address <mail address>,...] - List of Partner Addresses**

Use this parameter to display information only about nodes that have the "partner-address" e-mail addresses you specify. These addresses are not subject to the delivery limitations configured for the "-to" addresses of AutoSupport triggers.

## **[-support {enable|disable}] - Send AutoSupport Messages to Vendor Support**

Use this parameter with the value "enable" to display information only about nodes that send AutoSupport messages to your vendor's technical support. Use this parameter with the value "disable" to display information only about nodes that do not send AutoSupport messages to your vendor's technical support.

## **[-transport {smtp|http|https}] - Protocol to Contact Support**

Use this parameter to display information only about nodes that use the protocol you specify to send AutoSupport messages.

## **[-url <text>] - Support URL for HTTP/HTTPS**

Use this parameter to display information only about nodes that use the URL you specify to send messages through HTTP and HTTPS POST operations.

## **[-put-url <text>] - Support URL for HTTP/S PUT**

Use this parameter to display information only about nodes that use the URL you specify to send messages through HTTP PUT operations.

## **[-proxy-url <text>] - Support Proxy URL**

Use this parameter to display information only about nodes that use the proxy URL you specify.

## **[-support-address <mail address>,...] - Support Address**

Use this parameter to display information only about nodes that use the external support address you specify.

## **[-hostname-subj {true|false}] - Hostname Subject**

Use this parameter to display information only about nodes that include their hostname in the "Subject:" line of AutoSupport messages. If the parameter "remove-private-data" is `false`, this parameter has no effect.

## **[-nht {true|false}] - NHT Enable**

Use this parameter with the value "true" to display information only about nodes that send NHT disk drive data. Use this parameter with the value "false" to display information only about nodes that do not send NHT data.

## **[-perf {true|false}] - Performance Data Enable**

Use this parameter with the value "true" to display information only about nodes that send periodic performance AutoSupport messages. Use this parameter with the value "false" to display information only about nodes that do not send periodic performance messages.

**[-retry-interval <[<integer>h] [<integer>m] [<integer>s]>] - Retry Interval**

Use this parameter to display information only about nodes that use the retry interval you specify.

**[-retry-count <integer>] - Retry Count**

Use this parameter to display information only about nodes that use the retry count you specify.

**[-reminder {true|false}] - Reminder Enable**

Use this parameter with the value "true" to display information only about nodes that send messages reminding administrators to enable AutoSupport if AutoSupport is not enabled. Use this parameter with the value "false" to display information only about nodes that do not send reminder messages.

**[-last-subject <text>] - Last Subject Sent**

Use this parameter to display information only about nodes whose last AutoSupport message had the "Subject:" line you specify.

**[-last-time <MM/DD/YYYY HH:MM:SS>] - Last Time Sent**

Use this parameter to display information only about nodes whose last AutoSupport message was sent at the date and time you specify. Specify the date and time in the format "MM/DD/YYYY HH:MM:SS".

**[-max-http-size {<integer>[KB|MB|GB|TB|PB]}] - Maximum HTTP Size**

Use this parameter to display information only about nodes that limit the maximum size of HTTP transfers to the file size you specify.

**[-max-smtp-size {<integer>[KB|MB|GB|TB|PB]}] - Maximum SMTP Size**

Use this parameter to display information only about nodes that limit the maximum size of SMTP (e-mail) transfers to the file size you specify.

**[-remove-private-data {true|false}] - Remove Sensitive Data**

Use this parameter with the value "true" to display information only about nodes that remove sensitive data from AutoSupport messages. Use this parameter with the value "false" to display information only about nodes that do not remove sensitive data.

**[-validate-digital-certificate {true|false}] - Validate Digital Certificate Received**

Use this parameter with the value "true" to display information only about nodes that validate digital certificates they receive. Use this parameter with the value "false" to display information only about nodes that do not validate digital certificates.

**[-ondemand-state {enable|disable}] - AutoSupport OnDemand State**

Use this parameter to display information only about nodes that have the AutoSupport OnDemand state you specify.

**[-ondemand-remote-diagnostics-state {enable|disable}] - AutoSupport OnDemand Remote Diagnostics State**

Use this parameter to display information only about nodes that have the AutoSupport OnDemand Remote Diagnostics state you specify.

**[-ondemand-server-url <text>] - AutoSupport OnDemand Server URL**

Use this parameter to display information only about nodes that have the AutoSupport OnDemand Server URL you specify.

## Examples

The following example displays the AutoSupport configuration for a node named node3:

```
cluster1::> system node autosupport show -node node3
                Node: node3
                State: enable
                SMTP Mail Hosts: smtp.example.com
                From Address: alerts@node3.example.com
                List of To Addresses: support@example.com
                List of Noteto Addresses: -
                List of Partner Addresses: partner@node4.example.com
Send AutoSupport Messages to Vendor Support: enable
                Protocol to Contact Support: https
                Support Proxy URL: support.proxy.example.com
                Hostname Subject: true
                NHT Enable: true
                Performance Data Enable: true
                Retry Interval: 4m
                Retry Count: 15
                Reminder Enable: true
                The Transmission Window: 1h
                Last Subject Sent: WEEKLY
                Last Time Sent: 3/11/2011 06:00:03
                Maximum HTTP Size: 50MB
                Maximum SMTP Size: 5MB
                Remove Sensitive Data: false
                Validate Digital Certificate Received: true
                Continue Local Collection while Disabled: true
```

## system node autosupport check show-details

Display detailed status of AutoSupport subsystem

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### Description

The `system node autosupport check show-details` command displays the detailed status of the AutoSupport subsystem. This includes verifying connectivity to your vendor's AutoSupport destinations by sending test messages and providing a list of possible errors in your AutoSupport configuration settings.

### Parameters

```
{ [-fields <fieldname>,...]
```

If you specify the `-fields <fieldname>`, ... parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**[ `-instance` ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[ `-node <nodename>` ] - Node**

Selects the check results that match this parameter value. This parameter specifies the node whose status is being displayed.

**[ `-check-type <Type of AutoSupport Check>` ] - AutoSupport Check Type**

Selects the check results that match this parameter value. This parameter specifies the type of AutoSupport check being performed.

**[ `-status {ok|warning|failed|not-run}` ] - Status of the Check**

Selects the check results that match this parameter value. This parameter specifies the result of this AutoSupport check.

**[ `-error-detail <text>` ] - Detailed Description of Error**

Selects the check results that match this parameter value. This parameter specifies the detailed error message for this AutoSupport check.

**[ `-corrective-action <text>` ] - Corrective Action**

Selects the check results that match this parameter value. This parameter specifies a description of how to correct any errors seen as part of this AutoSupport Check

## Examples

The following example displays the detailed status of the AutoSupport subsystem for a node named node2:

```
cluster1::> system node autosupport check show-details -node node2
```



```
Node: node2

Category: http-https
  Component: http-put-destination
    Status: ok
    Detail: Successfully connected to "support.netapp.com/put".
  Component: http-post-destination
    Status: ok
    Detail: Successfully connected to "support.netapp.com/post".

Category: smtp
  Component: mail-server
    Status: ok
    Detail: Successfully connected to "mailhost.netapp.com".
  Component: mail-server
    Status: ok
    Detail: Successfully connected to "sendmail.domain.com".
  Component: mail-server
    Status: ok
    Detail: Successfully connected to "qmail.domain.com".

Category: on-demand
  Component: ondemand-server
    Status: ok
    Detail: Successfully connected to "support.netapp.com/aods".

Category: configuration
  Component: configuration
    Status: ok
    Detail: No configuration issues found.
```

## system node autosupport check show

Display overall status of AutoSupport subsystem

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### Description

The `system node autosupport status check show` command displays the overall status of the AutoSupport subsystem.

### Parameters

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>, ...` parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-instance ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[-node <nodename>] - Node**

Selects the nodes that match this parameter value. This parameter specifies the node whose status is being displayed.

**[-http-status {ok|warning|failed|not-run}] - Overall Status of AutoSupport HTTP/HTTPS Destinations**

Selects the nodes that match this parameter value. This parameter specifies whether connectivity to the AutoSupport HTTP destination was established.

**[-aod-status {ok|warning|failed|not-run}] - Overall Status of AutoSupport OnDemand Server**

Selects the nodes that match this parameter value. This parameter specifies the detailed description of the connectivity status to the AutoSupport OnDemand Server.

**[-smtp-status {ok|warning|failed|not-run}] - Overall Status of AutoSupport SMTP Destinations**

Selects the nodes that match this parameter value. This parameter specifies whether connectivity to the AutoSupport mailhost was established.

**[-config-status {ok|warning|failed|not-run}] - Overall Status of AutoSupport Configuration**

Selects the nodes that match this parameter value. This parameter specifies whether the AutoSupport configuration check succeeded or not.

**[-warning-text <text>] - Conditional Warning Message**

Selects the nodes that match this parameter value. This parameter specifies how to get more details regarding the status of the AutoSupport subsystem, in case of any errors.

## Examples

The following example displays the overall status of the AutoSupport subsystem on a node named node2:

```
cluster1::> system node autosupport check show -node node2
```

```
On Demand
Server      SMTP      Configuration
-----
ok          ok        ok
```

# system node autosupport destinations show

Display a summary of the current AutoSupport destinations

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description

The `system node autosupport destinations show` command displays a list of all message destinations used by AutoSupport. The command provides you with a quick summary of all addresses and URLs that receive AutoSupport messages from all nodes in the cluster.

## Parameters

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>, ...` parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-instance ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[-node {<nodename>|local}] - Node**

Use this parameter to display only destinations that receive AutoSupport messages from the node you specify.

**[-destinations <text>,...] - Destinations**

Use this parameter to display only destination lists for nodes that send AutoSupport messages to the destinations you specify.

## Examples

This example displays all of the destinations in use by the current cluster. Each node uses the same destination for HTTP POST, HTTP PUT, and e-mail notifications.

```

cluster1::> system node autosupport destinations show
Node
  Destinations
-----
----
node1
  https://asuppost.example.com/cgi-bin/asup.cgi
  https://asupput.example.com/cgi-bin/asup.cgi
  support@example.com

node2
  https://asuppost.example.com/cgi-bin/asup.cgi
  https://asupput.example.com/cgi-bin/asup.cgi
  support@example.com

```

## system node autosupport history cancel

Cancel an AutoSupport Transmission.

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### Description

The `system node autosupport history cancel` command cancels an active AutoSupport transmission. This command is used to pause or abandon a long running delivery of an AutoSupport message. The cancelled AutoSupport message remains available for retransmission using the [system node autosupport history retransmit](#) command.

### Parameters

**-node {<nodename>|local} - Node**

Use this parameter to specify the node on which to cancel the AutoSupport message. The default setting is localhost.

**-seq-num <Sequence Number> - AutoSupport Sequence Number**

Use this parameter to specify the sequence number of the AutoSupport message you want to cancel.

**[-destination {smtp|http|noteto|retransmit}] - Destination for This AutoSupport**

Use this parameter to specify the destination type for the AutoSupport message you want to cancel.

### Examples

Use this command to cancel the AutoSupport message delivery with seq-num 10 to all destinations.

```

cluster1::> system node autosupport history cancel -node local -seq-num 10

```

Use this command to cancel the AutoSupport message delivery with seq-num 10 via HTTP only.

```
cluster1::> system node autosupport history cancel -node local -seq-num 10
-destination http
```

## Related Links

- [system node autosupport history retransmit](#)

# system node autosupport history retransmit

Selectively retransmit a previously collected AutoSupport.

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description

The `system node autosupport history retransmit` command retransmits a locally stored AutoSupport message.

Support personnel might ask you to run this command to retransmit an AutoSupport message. You might also retransmit an AutoSupport message if you run the [system node autosupport history show](#) command and notice that a message was not delivered.

If you retransmit an AutoSupport message, and if support already received that message, the support system will not create a duplicate case. If, on the other hand, support did not receive that message, then the AutoSupport system will analyze the message and create a case, if necessary.

Use the [system node autosupport history show](#) command to display the 50 most recent AutoSupport messages, which are available for retransmission.

## Parameters

**-node {<nodename>|local} - Node**

Use this parameter to specify the node from which the AutoSupport message is sent.

**-seq-num <Sequence Number> - AutoSupport Sequence Number**

Use this parameter to specify the sequence number of the AutoSupport message to retransmit.

**-uri <text> - Destination to Send this AutoSupport**

Use this parameter to specify the HTTP, HTTPS, FILE, or MAILTO uniform resource indicator (URI) to which the AutoSupport message is sent.

**[ -size-limit {<integer>[KB|MB|GB|TB|PB] } ] - Transmit Size Limit for this AutoSupport.**

Use this parameter to specify a size limit for the retransmitted AutoSupport message. If the message information exceeds this limit, it will be trimmed to fit the limit you specify. Omit the size limit or set it to 0 to disable it, which is useful to retransmit an AutoSupport message that was truncated by a mail or Web server due to the default size limits.

## Examples

The following example retransmits the AutoSupport message with sequence number 45 on the node "node1" to a support address by e-mail.

```
cluster1:~> system node autosupport history retransmit -node node1 -seq
-num 45 -uri mailto:support@example.com
```

## Related Links

- [system node autosupport history show](#)

# system node autosupport history show-upload-details

Display upload details of recent AutoSupport messages

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description

The `system node autosupport history show-upload-details` command displays upload details of the 50 most recent AutoSupport messages sent by nodes in the cluster. By default, it displays the following information:

- AutoSupport Sequence Number
- Destination
- Compressed Size
- Percentage Complete
- Rate of upload
- Time Remaining

## Parameters

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>, ...` parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-instance ] }**

Use this parameter to display the following detailed information about all entries:

- AutoSupport Sequence Number
- Destination
- Compressed Size
- Percentage Complete
- Rate of Upload

- Time Remaining

**[-node {<nodename>|local}] - Node**

Use this parameter to display details of AutoSupport messages sent from the node you specify.

**[-seq-num <Sequence Number>] - AutoSupport Sequence Number**

Use this parameter to display details of AutoSupport messages with the sequence number you specify. Sequence numbers are unique to a node. Use this parameter with the `-node` parameter to display information about an individual message.

**[-destination {smtp|http|noteto|retransmit}] - Destination for this AutoSupport**

Use this parameter to display details of AutoSupport messages that were sent to the destination type you specify.

**[-size {<integer>[KB|MB|GB|TB|PB]}] - Autosupport Compressed Size**

Use this parameter to display details of AutoSupport messages of the compressed size you specify.

**[-percent-complete <integer>] - Percent Complete**

Use this parameter to display the percentage completed for any active (incomplete) AutoSupport message.

**[-upload-rate {<integer>[Bps|KBps|MBps|GBps]|unlimited}] - Rate of Upload**

Use this parameter to display the rate in bytes per second that upload is using currently, otherwise zero when not active.

**[-time-remaining <[<integer>h] [<integer>m] [<integer>s]>] - Time remaining for Upload**

Use this parameter to display the estimated time for the transmission of the AutoSupport message to complete.

## Examples

The following example shows the first three results output by the `history show-upload-details` command. Note that "q" was pressed at the prompt.

```

cluster1::> system node autosupport history show-upload-details -node
node1

```

Node	Seq Num	Destination	Size	Percent Complete	Rate	Time Remaining
node1	13	smtp	755.9KB	100	142.88KBps	0s
		http	755.8KB	80	125.97KBps	10s
		noteto	-	-	-	-
	12	smtp	-	-	-	-
		http	316.4KB	100	158.22KBps	0s
		noteto	201B	100	201Bps	0s
	11	smtp	-	-	-	-
		http	626.1MB	100	649.56KBps	0s
		noteto	-	-	-	-

Press <space> to page down, <return> for next line, or 'q' to quit... q  
9 entries were displayed.

## system node autosupport history show

Display recent AutoSupport messages

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### Description

The `system node autosupport history show` command displays information about the 50 most recent AutoSupport messages sent by nodes in the cluster. By default, it displays the following information:

- AutoSupport sequence number
- Destination type, such as smtp
- Status of delivery, such as sent-successful
- Attempt count
- Time of last update

### Parameters

{ [-fields <fieldname>,...]

If you specify the `-fields <fieldname>, ...` parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.



**| [-delivery ]**

Use this parameter to display destination information about each AutoSupport message.

**| [-detail ]**

Use this parameter to display trigger and subject information about each AutoSupport message.

**| [-instance ] }**

Use this parameter to display the following detailed information about all entries:

- Trigger event
- Subject of the message
- Delivery URI
- Last error
- Compressed Size
- Decompressed Size
- Total Collection Time (in ms)

**[-node {<nodename>|local}] - Node**

Use this parameter to display only AutoSupport messages sent from the node you specify.

**[-seq-num <Sequence Number>] - AutoSupport Sequence Number**

Use this parameter to display only AutoSupport messages with the sequence number you specify. Sequence numbers are unique to a node. Use this parameter with the `-node` parameter to display information about an individual message.

**[-destination {smtp|http|noteto|retransmit}] - Destination for This AutoSupport**

Use this parameter to display only AutoSupport messages that were sent to the destination type you specify.

**[-trigger <Message Name>] - Trigger Event**

Use this parameter to display only AutoSupport messages that match the trigger event you specify.

**[-last-update <MM/DD/YYYY HH:MM:SS>] - Time of Last Update**

Use this parameter to display only AutoSupport messages that were updated most recently at the time you specify. Specify time in "MM/DD/YYYY HH:MM:SS" format.

**[-status <AutoSupport general status>] - Status of Delivery**

Use this parameter to display only AutoSupport messages with the status you specify. Possible statuses are:

- initializing - The AutoSupport message request is being processed.
- collection-failed - The AutoSupport collection failed. View the 'Last Error' field of this message for more information.
- collection-in-progress - The AutoSupport collection is in progress.
- queued - The AutoSupport message is queued for delivery.
- transmitting - The AutoSupport message transmission is in progress.

- sent-successful - The AutoSupport message was sent successfully.
- ignore - The AutoSupport message was processed successfully, but the trigger event is not configured for delivery to the current destination type.
- re-queued - The AutoSupport message transmission failed, has been re-queued, and will be retried.
- transmission-failed - The AutoSupport message transmission failed, and the retry limit was exceeded.
- ondemand-ignore - The AutoSupport message was processed successfully, but the AutoSupport On Demand server chose to ignore it.

**[-attempt-count <integer>] - Delivery Attempts**

Use this parameter to display only AutoSupport messages that the system has attempted to send the number of times you specify. This parameter is most useful when given a range, such as ">5".

**[-subject <text>] - AutoSupport Subject**

Use this parameter to display only AutoSupport messages of the type you specify.

**[-uri <text>] - Delivery URI**

Use this parameter to display only AutoSupport messages sent to the destination URI you specify.

**[-error <text>] - Last Error**

Use this parameter to display only AutoSupport messages that failed with the "Last Error" description you specify.

**[-generated-on <MM/DD/YYYY HH:MM:SS>] - Time of Generation**

Use this parameter to display only AutoSupport messages that were generated (collected) at a particular time.

**[-size {<integer>[KB|MB|GB|TB|PB]}] - AutoSupport Compressed Size**

Use this parameter to display only AutoSupport messages of the compressed size you specify.

**[-percent-complete <integer>] - Percent Complete**

Use this parameter to display the percentage completed for any active (incomplete) AutoSupport message.

**[-upload-rate {<integer>[Bps|KBps|MBps|GBps]|unlimited}] - Rate of Upload**

Use this parameter to display the rate in bytes per second that upload is using currently, otherwise zero when not active.

**[-time-remaining <[<integer>h] [<integer>m] [<integer>s]>] - Time Remaining for Upload**

Use this parameter to display the estimated time for the transmission of the AutoSupport message to complete.

**[-decompressed-size {<integer>[KB|MB|GB|TB|PB]}] - AutoSupport Decompressed Size**

Use this parameter to display only AutoSupport messages of the decompressed size you specify.

**[-total-time <integer>] - Total Collection Time (ms)**

Use this parameter to display only AutoSupport messages of total collection time you specify. A value is only shown when the collection has completed.

## Examples

The following example shows the first three results output by the history command. Note that "q" was pressed at the prompt.

```
cluster1::> system node autosupport history show -node node1
      Seq          Attempt  Last
Node  Num  Destination Status          Count  Update
-----
node1  56
      smtp          ignore          1      11/18/2010
01:10:01
      http          re-queued      2      11/18/2010
02:50:07
      noteto        transmitting   1      11/18/2010
01:10:01
      55
      smtp          ignore          1      11/18/2010
00:53:59
      http          sent-successful 3      11/18/2010
01:50:03
      noteto        sent-successful 1      11/18/2010
00:53:59
      54
      smtp          ignore          1      11/17/2010
12:18:58
      http          sent-successful 4      11/17/2010
16:07:22
      noteto        sent-successful 1      11/17/2010
12:18:58
Press <space> to page down, <return> for next line, or 'q' to quit... q
9 entries were displayed.
```

## system node autosupport manifest show

Display AutoSupport content manifest

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### Description

The `system node autosupport manifest show` command reports what information is contained in AutoSupport messages. The name and size of each file collected for the message is reported, along with any errors.

## Parameters

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>`, ... parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-content ]**

Use this parameter to display detailed information about the content of the files contained in the report.

**| [-instance ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[-node {<nodename>|local}] - Node**

Use this parameter to display information about only AutoSupport messages sent from the node you specify.

**[-seq-num <Sequence Number>] - AutoSupport Sequence Number**

Use this parameter to display information about only AutoSupport message content with the sequence number you specify. Sequence numbers are unique to a node. Use this parameter with the `-node` parameter to display information about an individual message.

**[-prio-num <integer>] - Priority Order of Collection**

Use this parameter to display information about only AutoSupport message content with the collection priority you specify. Content is collected in order, by priority number.

**[-subsys <subsys1,subsys2,...>] - Subsystem**

Use this parameter to display information about only AutoSupport message content collected by the AutoSupport subsystem you specify.

**[-cmd-tgt <Execution domain of AutoSupport content>] - Execution Domain for Command**

Use this parameter to display information about only AutoSupport message content produced in the execution domain you specify.

**[-body-file <text>] - The AutoSupport Content Filename for this Data**

Use this parameter to display information about only AutoSupport message content stored in the body file with the file name you specify.

**[-cmd <text>] - Actual Data Being Collected**

Use this parameter to display information about only AutoSupport message content produced by the D-Blade command, BSD command, file, or XML table you specify.

**[-query <text>] - Table Query for XML Collection**

Use this parameter to display information about only AutoSupport message content produced by the table query you specify.

**[-size-collected {<integer>[KB|MB|GB|TB|PB]}] - Number of Bytes Collected**

Use this parameter to display information about only AutoSupport message content collected in files with the file size you specify.

**[`-time-collected <integer>`] - Collection Time for this Data Item (ms)**

Use this parameter to display information about only AutoSupport message content collected in the amount of time you specify, in milliseconds.

**[`-status <AutoSupport manifest collection status>`] - Status of this Data Item**

Use this parameter to display information about only AutoSupport message content with the collection status you specify. Possible statuses are:

- `requested` - The AutoSupport request has been added to the queue and is waiting processing by the collector.
- `working` - The AutoSupport collector is actively gathering the needed data.
- `file-not-found` - AutoSupport data collection failed because a necessary file is missing.
- `no-such-table` - The AutoSupport collector was unable to find the requested SMF table.
- `collection-truncated-size-limit` - AutoSupport data was truncated due to size limits, but partial data is available.
- `collection-truncated-file-size-limit` - AutoSupport data for a particular data item or file was truncated due to file size limits, but partial data is available.
- `collection-skipped-size-limit` - AutoSupport data was skipped due to size limits, and no data is available.
- `collection-truncated-time-limit` - AutoSupport data was truncated due to time limits, but partial data is available.
- `collection-skipped-time-limit` - AutoSupport data was skipped due to time limits, and no data is available.
- `delivery-skipped-size-limit` - AutoSupport data was skipped at delivery time due to size limits.
- `general-error` - AutoSupport data collection failed. Additional information (if any) is in the Error String field.
- `completed` - AutoSupport data collection is complete, and the AutoSupport message is ready for delivery.
- `content-not-collected-precheck` - AutoSupport content was not collected due to pre-check function violation.
- `content-not-collected-privacy` - AutoSupport content was not collected because the operation is disabled in privacy mode.
- `content-empty` - AutoSupport content was collected successfully, but the output was empty.
- `collection-aborted` - AutoSupport data collection was aborted.

**[`-error <text>`] - Textual Description of Error**

Use this parameter to display information about only AutoSupport message content with the error text you specify. If data collection has failed, the error text contains a description of the failure. If data collection completes successfully, this field is empty.

**[`-content-type <Type of AutoSupport content>`] - AutoSupport Content Type for this Data**

Use this parameter to display information about only AutoSupport message content of the type you specify. Types supported are:

- basic - Configuration data about this subsystem
- troubleshooting - Detailed diagnostic data about this subsystem

**[`-orig-size-collected` {<integer>[KB|MB|GB|TB|PB]}] - Initial Number of Bytes Collected**

Use this parameter to display information about only AutoSupport message content collected in files with the original file size you specify.

**[`-size-compressed` {<integer>[KB|MB|GB|TB|PB]}] - Compressed Size**

Use this parameter to display information about only AutoSupport message content collected in files with the compressed file size you specify.

## Examples

This example displays the content of AutoSupport message number 372 on the node "node1".

```
cluster1::> system node autosupport manifest show -node node1 -seq-num 372
```

Node	Sequence	AutoSupport Body Filename	Collected Size	Status	Error
node1	372	SYSCONFIG-A.txt	1.73KB	completed	
		OPTIONS.txt	29.44KB	completed	
		software_image.xml	7.56KB	completed	
		CLUSTER-INFO.xml	3.64KB	completed	
		autosupport.xml	12.29KB	completed	
		autosupport_budget.xml	7.01KB	completed	
		autosupport_history.xml	46.52KB	completed	
		X-HEADER-DATA.TXT	717.00B	completed	
		SYSTEM-SERIAL-NUMBER.TXT	35.00B	completed	
		cluster_licenses.xml	3.29KB	completed	
		cm_hourly_stats.gz	151.4KB	completed	
		boottimes.xml	56.86KB	completed	
		rdb_txn_latency_stats_hrly.xml	39.31KB	completed	
		rdb_voting_latency_stats_hrly.xml	3.43KB	completed	

14 entries were displayed.

This example shows how you can use parameters to limit output to specific fields of a specific AutoSupport message. This is helpful when troubleshooting.

```

cluster1::> system node autosupport manifest show -node node5 -seq-num 842
-fields body-file,status,size-collected,time-collected,cmd,cmd-tgt,subsys
node          seq-num prio-num subsys      cmd-tgt body-file          cmd
size-collected time-collected status
-----
node5          842      0          mandatory dblade  SYSCONFIG-A.txt "sysconfig
-a" 16.44KB          256          completed
node5          842      1          mandatory dblade  OPTIONS.txt      options
29.67KB          3542          completed
node5          842      2          mandatory smf_table software_image.xml
software_image 8.68KB      33          completed
node5          842      3          mandatory smf_table CLUSTER-INFO.xml
asup_cluster_info 4.75KB      7          completed
node5          842      4          mandatory smf_table autosupport.xml
autosupport 12.32KB          10          completed
node5          842      5          mandatory smf_table autosupport_budget.xml
autosupport_budget 7.03KB 29          completed
node5          842      6          mandatory smf_table autosupport_history.xml
autosupport_history 62.77KB 329        completed
node5          842      7          mandatory custom_fx X-HEADER-DATA.TXT "Custom
function" 720.00B 3          completed
node5          842      8          mandatory custom_fx SYSTEM-SERIAL-NUMBER.TXT
"Custom function" 31.00B 2          completed
node5          842      9          mandatory smf_table cluster_licenses.xml
cluster_licenses 5.62KB 9          completed
node5          842      10         log_files custom_fx log_files.xml "Custom
function" 13.07KB      4          completed
node5          842      11         log_files custom_fx EMS-LOG-FILE.gz "Custom
function" 25.33KB      24          completed
node5          842      12         log_files dblade_file EMS-LOG-FILE-PARTNER.gz
/etc/log/ems - -          content-not-collected-precheck
node5          842      13         log_files dblade_file MESSAGES.gz
/etc/log/messages 35.40KB      42          completed
node5          842      14         log_files dblade_file MESSAGES-PARTNER.gz
/etc/log/messages - -          content-not-collected-precheck
14 entries were displayed.

```

## system node autosupport trigger modify

Modify AutoSupport trigger configuration

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description

Use the `system node autosupport trigger modify` command to enable and disable AutoSupport messages for individual triggers, and to specify additional subsystem reports to include if an individual trigger sends an AutoSupport message.

## Parameters

### **-node {<nodename>|local} - Node**

Use this parameter to specify the node whose AutoSupport trigger configuration is modified.

### **-autosupport-message <Autosupport Message> - EMS Message**

Use this parameter to specify the AutoSupport trigger to modify. AutoSupport triggers are EMS messages whose names begin with "callhome.". However, for the purposes of this command, "callhome." is implied, does not need to be entered, and will not be displayed in command output.

### **[-to {enabled|disabled}] - Deliver to AutoSupport -to Addresses**

Use this parameter with the value "enabled" to enable sending AutoSupport messages to the configured "to" addresses.

### **[-noteto {enabled|disabled}] - (DEPRECATED) Deliver to AutoSupport -noteto Addresses**



This parameter has been deprecated and might be removed in a future version of Data ONTAP.

Use this parameter with the value "enabled" to enable sending short notes to the configured "noteto" addresses.

### **[-basic-additional <subsys1,subsys2,...>,...] - Additional Subsystems Reporting Basic Info**

Use this parameter to include *basic* content from the additional subsystems you specify. Content is collected from these subsystems in addition to the default list of subsystems.

### **[-troubleshooting-additional <subsys1,subsys2,...>,...] - Additional Subsystems Reporting Troubleshooting Info**

Use this parameter to include *troubleshooting* content from the additional subsystems you specify. Content is collected from these subsystems in addition to the default list of subsystems.

### **[-suppress {true|false}] - Suppress all occurrences of this trigger**

Use this parameter with the value "true" to suppress the collection when the AutoSupport message is triggered.

## Examples

The following example enables messages to the configured "to" addresses from the `battery.low` trigger on the node `node1`.

```
cluster1::> system node autosupport trigger modify -node node1
-autosupport-message battery.low -to enabled
```



# system node autosupport trigger show

Display AutoSupport trigger configuration

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description

The `system node autosupport trigger show` command displays what system events trigger AutoSupport messages. When a trigger event occurs, the node may send an AutoSupport message to a predefined destination, and a short note to another destination. The full AutoSupport message contains detail for troubleshooting. The short message is meant for short pager or SMS text messages.

Use the [system node autosupport destinations show](#) command to view available destinations.

## Parameters

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>, ...` parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-basic ]**

Use this parameter to display which subsystem information is included as *basic* content when the AutoSupport message is triggered.

**| [-troubleshooting ]**

Use this parameter to display which subsystem information is included as *troubleshooting* content when the AutoSupport message is triggered.

**| [-instance ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[-node {<nodename>|local}] - Node**

Use this parameter to display AutoSupport triggers only on the node you specify.

**[-autosupport-message <Autosupport Message>] - EMS Message**

Use this parameter to display only AutoSupport triggers with the name you specify. AutoSupport triggers are EMS messages whose names begin with "callhome.". However, for the purposes of this command, "callhome." is implied, does not need to be entered, and will not be displayed in command output.

**[-to {enabled|disabled}] - Deliver to AutoSupport -to Addresses**

Use this parameter with the value "enabled" to display only AutoSupport messages that send full messages to the "to" address when triggered. Use this parameter with the value "disabled" to display only AutoSupport messages that do not send full messages.

**[-noteto {enabled|disabled}] - (DEPRECATED) Deliver to AutoSupport -noteto Addresses**



This parameter has been deprecated and might be removed in a future version of Data ONTAP.

Use this parameter with the value "enabled" to display only AutoSupport messages that send short notes to

the "noteto" address when triggered. Use this parameter with the value "disabled" to display only AutoSupport messages that do not send short notes.

**[-basic-default <subsys1,subsys2,...>,...] - Default Subsystems Reporting Basic Info**

Use this parameter to display only AutoSupport triggers that include in their messages, by default, *basic* content from the subsystems you specify.

**[-troubleshooting-default <subsys1,subsys2,...>,...] - Default Subsystems Reporting Troubleshooting Info**

Use this parameter to display only AutoSupport triggers that include in their messages, by default, *troubleshooting* content from the subsystems you specify.

**[-additional-content <Type of AutoSupport content>,...] - Additional Content Flag**

Use this parameter to display only AutoSupport triggers that have been configured to include additional *basic* or *troubleshooting* content.

**[-basic-additional <subsys1,subsys2,...>,...] - Additional Subsystems Reporting Basic Info**

Use this parameter to display only AutoSupport triggers that have been configured to include additional *basic* content from the subsystems you specify.

**[-troubleshooting-additional <subsys1,subsys2,...>,...] - Additional Subsystems Reporting Troubleshooting Info**

Use this parameter to display only AutoSupport triggers that have been configured to include additional *troubleshooting* content from the subsystems you specify.

**[-suppress {true|false}] - Suppress all occurrences of this trigger**

Use this parameter with the value "true" to display only AutoSupport messages that have been suppressed.

## Examples

This example shows the first page of output from the command. Note that "q" was pressed at the prompt to quit.

```

cluster1::> system node autosupport trigger show
          AutoSupport
Node      Message                               To      Note To  Additional
-----  -
node1     aggr.offline                             enabled enabled   -
node1     aggr.restricted                          disabled enabled   -
node1     aggr.wafliron                             disabled enabled   -
node1     bad.ram                                   disabled disabled  -
node1     battery.failure                          enabled  enabled   -
node1     battery.low                              disabled disabled  -
node1     battery.notice                           enabled  enabled   -
node1     battery.overchg                           enabled  enabled   -
node1     battery.overtemp                          enabled  enabled   -
node1     battery.warning                          enabled  enabled   -
node1     bmc.bus                                   disabled disabled  -
node1     bmc.hb.stop                              disabled disabled  -
node1     bmc.post                                  disabled disabled  -
node1     bootfs.chkdisk                           enabled  enabled   -
node1     c.fan                                     enabled  enabled   -
node1     c.fan.fru.degraded                       disabled disabled  -
node1     c.fan.fru.fault                          disabled enabled   -
node1     c.fan.fru.rm                             disabled enabled   -
node1     c.fan.fru.shut                           enabled  enabled   -
node1     ch.ps.degraded                           disabled disabled  -
Press <space> to page down, <return> for next line, or 'q' to quit... q
20 entries were displayed.

```

## Related Links

- [system node autosupport destinations show](#)

## system node coredump delete-all

Delete all coredumps owned by a node

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### Description

The `system node coredump delete-all` command deletes either all unsaved core dumps or all saved core files on a node. You can specify whether saved core files or unsaved core dumps are deleted by using the optional `-saved` parameter. If the command is issued while a core dump is being saved, the command prompts you before stopping the save operation and deleting the core dump.

## Parameters

### **-node <nodename> - Node That Owns the Coredump**

This specifies the node from which core files or core dumps are to be deleted.

### **[-type {unsaved-kernel|saved-kernel|kernel|application|all}] - Type of Core to delete**

This parameter specifies the type of core file to be deleted. If the type is `unsaved`, all unsaved core dumps will be deleted. If the type is `saved`, all saved core files will be deleted. If the type is `kernel`, all kernel core files and kernel core dumps will be deleted. If the type is `application`, all application core files will be deleted. If the type is `all`, all core files will be deleted. The default setting is to delete only unsaved kernel core dumps and core files.

## Examples

The following example deletes all unsaved kernel core dumps on a node named `node0`:

```
cluster1::> system node coredump delete-all -node node0
```

## system node coredump delete

Delete a coredump

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description

The `system node coredump delete` command deletes a specified core dump. If the command is issued while the specified core dump is being saved, the command prompts you before stopping the save operation and deleting the core dump.

## Parameters

### **-node {<nodename>|local} - Node That Owns the Coredump**

This specifies the node from which core files are to be deleted.

### **[-type {kernel|ancillary-kernel-segment|application}] - Coredump Type**

This specifies the type of core file to be deleted. If the type is `kernel`, the specified kernel core file will be deleted. If the type is `application`, the specified application core file will be deleted.

### **-corename <text> - Coredump Name**

This specifies the core file that is to be deleted.

## Examples

The following example deletes a core dump named `core.101268397.2010-05-30.19_37_31.nz` from a node named `node0`:

```
cluster1::> system node coredump delete -node node0 -corename
core.101268397.2010-05-30.19_37_31.nz
```

## system node coredump save-all

Save all unsaved kernel coredumps owned by a node

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### Description

The `system node coredump save-all` saves all unsaved core dumps on a specified node. If the node has already attempted to save the core dump by the value set by the `-save-attempts` parameter, the command prompts you before continuing. The `save-attempts` parameter is set by invoking the command `system node coredump config modify`.

### Parameters

**-node <nodename> - Node That Owns the Coredump**

This specifies the node on which unsaved core dumps are to be saved.

### Examples

The following example saves all unsaved core dumps on a node named `node0`:

```
cluster1::> system node coredump save-all -node node0
```

## system node coredump save

Save an unsaved kernel coredump

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### Description

The `system node coredump save` command saves a specified core dump. If the node has already attempted to save the core dump by the value specified by the `-save-attempts` parameter, the command prompts you before continuing. The `-save-attempts` parameter is set by invoking the command [system node coredump config modify](#). A saved core dump can be uploaded to a remote site for support analysis; see the [system node coredump upload](#) command man page for more information.

### Parameters

**-node {<nodename>|local} - Node That Owns the Coredump**

This specifies the node on which the core dump is located.

### **-corename <text> - Coredump Name**

This specifies the core dump that is to be saved.

## **Examples**

The following example saves a core dump named `core.101268397.2010-05-30.19_37_31.nz` on a node named `node0`:

```
cluster1::> system node coredump save -node node0 -corename
core.101268397.2010-05-30.19_37_31.nz
```

## **Related Links**

- [system node coredump config modify](#)
- [system node coredump upload](#)

# **system node coredump show**

Display a list of coredumps

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## **Description**

The `system node coredump show` command displays basic information about core dumps, such as the core dump name, time of panic that triggered the core dump and whether the core file is saved. You can specify optional parameters to display information that matches only those parameters. For example, to display the list of kernel core files, run the command with `-type kernel`.

## **Parameters**

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>, ...` parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-system ]**

If you specify this parameter, the command displays the following information:

- Node name
- Core dump name
- Core dump ID
- Node that panicked and generated the core
- System ID of the node that panicked and generated the core
- Version of the core

**[`-instance` ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[`-node {<nodename>|local}`] - Node That Owns the Coredump**

If you specify both this parameter and the `-corename` parameter, the command displays detailed information about the specified core. If you specify this parameter by itself, the command displays information about the core files on the specified node.

**[`-type {kernel|ancillary-kernel-segment|application}`] - Coredump Type**

This parameter specifies the type of core files to be displayed. If the type is `kernel` and the system supports segmented core files, the command displays information about primary kernel core segment files. If the type is `kernel` and the system does not support segmented core files, the command displays information about full core files. If the type is `ancillary-kernel-segment`, the command displays information about ancillary kernel core segment files. If the type is `application`, the command displays information about application core files. If no type is specified, the command displays information about core files of type `kernel` or `application`.

**[`-corename <text>`] - Coredump Name**

If you specify both this parameter and the `-node` parameter, the command displays detailed information about the specified core. If you specify this parameter by itself, the command displays information about the core files that match the specified name.

**[`-panic-node <text>`] - Node That Generated Core**

If you specify this parameter with a node name, the command displays information only about the core files that were generated when the specified node panicked.

**[`-panic-systemid <integer>`] - System ID of Node That Generated Core**

If you specify this parameter, the command displays information only about the core files that were generated when the node with the specified system ID panicked.

**[`-version <text>`] - Data ONTAP Version of Core**

If you specify this parameter, the command displays information only about the core files that match the specified version.

**[`-panic-time <MM/DD/YYYY HH:MM:SS>`] - Time of Panic That Generated Core**

If you specify this parameter, the command displays information only about the core files that were generated by a panic at the specified time. Specify time in the format of `MM/DD/YYYY HH:MM:SS [+ - HH:MM]`. You can use `[+ - HH:MM]` to specify the time range within which all core files triggered by a panic are displayed. `[+ - HH:MM]` is relative to UTC.

**[`-panic-string <text>`] - Panic String**

If you specify this parameter, the command displays information only about the core files that match the specified panic string.

**[`-is-saved {true|false}`] - Saved Core**

If you specify this parameter, the command displays information only about the core dumps that are or are not saved yet to a core file.

**[`-is-partial {true|false}`] - Partial Core**

If you specify this parameter, the command displays information only about the core dumps that are or are

not partially saved.

**[-save-attempts <integer>] - Number of Attempts to Save Core**

If you specify this parameter, the command displays information only about the core dumps that have the specified number of successful or failed save attempts.

**[-space-needed {<integer>[KB|MB|GB|TB|PB]}] - Space Needed To Save Core**

If you specify this parameter, the command displays information only about the core dumps that need the specified amount of disk space to save into a core file.

**[-size <text>] - Size of Core (bytes)**

If you specify this parameter, the command displays information only about the saved core files that are of the specified size.

**[-md5-data-chksum <text>] - MD5 Checksum of the Compressed Data of Core**

If you specify this parameter, the command displays information only about the saved core files that have the specified MD5 checksum for compressed data of the core.

**[-ancillary-segment-directory <text>] - Directory Holding Ancillary Kernel Core Segments**

If you specify this parameter, the command displays information only about the saved core files that have the specified ancillary segment directory.

## Examples

The following examples display information about the core files:

```
cluster1::> system node coredump show
Node   Core Name                               Saved  Panic Time
-----
node0
  core.101182345.2010-02-01.14_19_08.nz    false  2/1/2010 09:19:08
  Partial Core: false
  Number of Attempts to Save Core: 2
  Space Needed To Save Core: 4.45GB
node1
  core.101268397.2010-05-30.19_37_31.nz    true   5/30/2010 15:37:31
node2
  core.101270930.2010-09-06.18_40_03.nz    true   9/6/2010 14:40:03
node3
  core.101271326.2010-09-06.19_06_18.nz    true   9/6/2010 15:06:18
  core.101271326.2010-09-06.19_09_49.nz    true   9/6/2010 15:09:49
4 entries were displayed.
```



```

cluster1::> system node coredump show -panic-time 9/6/2010 15:00:00+3:00
Node   Core Name                               Saved   Panic Time
-----
node3
  core.101271326.2010-09-06.19_06_18.nz   true   9/6/2010 15:06:18
  core.101271326.2010-09-06.19_09_49.nz   true   9/6/2010 15:09:49
2 entries were displayed.

```

## system node coredump status

Display kernel coredump status

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### Description

The `system node coredump status` command displays status information about core dumps. The command output depends on the parameters specified with the command. If a core dump is in the process of being saved into a core file, the command also displays its name, the total number of blocks that are to be saved, and the current number of blocks that are already saved.

You can specify additional parameters to display only information that matches those parameters. For example, to display coredump status information about the local node, run the command with the parameter `-node local`.

### Parameters

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>, ...` parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-disks ]**

If you specify this parameter, the command displays the following information:

- Node name
- Total number of disks
- Number of spare disks
- Number of disks used
- Number of disks with partial cores

**| [-spraycore ]**

If you specify this parameter, the command displays the following information:

- Node name
- Whether spray cores are supported

- Number of spray-core disks
- Number of spray-core blocks
- Number of disks needed for spray core
- Estimated number of blocks needed for spray core

**[`-instance` ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[`-node` {<nodename>|local}] - Node**

If you specify this parameter, the command displays the following information:

- Node name
- State of the core-dump process
- Space available on the internal file system
- Name of the core being saved, if applicable
- Total number of blocks in the core being saved, if applicable
- Number of blocks currently saved, if applicable
- Type of core dump
- Number of unsaved complete cores on the node
- Number of unsaved partial cores on the node
- Whether spray cores are supported on the node
- Whether any spare disks are available on the node
- Number of disks that have cores
- Number of unsaved cores
- Number of disks that have partial cores
- Number of partial cores
- Number of unused spray-core disks
- Number of spray-core blocks
- Number of disks available for core dumps
- Estimated number of blocks needed for spray core
- Number of disks needed for spray core

**[`-state` <text>] - State**

If you specify this parameter, the command displays information only about the nodes that are in the specified core dump state. Possible values include: `nocore`, `idle`, `init`, `saving`, and `waitdump`.

**[`-space-available` {<integer>[KB|MB|GB|TB|PB]}] - Space Available On Internal Filesystem**

If you specify this parameter, the command displays information only about the nodes that have the specified amount of available space, in bytes, on their internal file systems.

**[`-corename` <text>] - Name of Core Being Saved**

If you specify this parameter, the command displays information only about the node that is currently saving

the specified core file name.

**[-total-blocks <integer>] - Total Number of Blocks in Core Being Saved**

If you specify this parameter, the command displays information only about the nodes that have the specified number of blocks in the core dump being saved.

**[-blocks-saved <integer>] - Number of Blocks saved**

If you specify this parameter, the command displays information only about the nodes that have the specified number of blocks saved.

**[-type <text>] - Type of Core Dump**

If you specify this parameter, the command displays information only about the nodes that have the specified core dump type. Possible values include zipped, sprayed, and spare.

**[-spraycore-supported {true|false}] - Spray Core Supported on Node**

If you specify this parameter, the command displays information only about the nodes that do or do not support the spray method of dumping core.

**[-spares-available {true|false}] - Spare Disk(s) Available on Node**

If you specify this parameter, the command displays information only about the nodes that do or do not have spare disks available.

**[-disks-used <integer>] - Number of Disks with Cores**

If you specify this parameter, the command displays information only about the nodes that have the specified number of disks that contain core dumps.

**[-unsaved-cores <integer>] - Number of Unsaved Complete Cores**

If you specify this parameter, the command displays information only about the nodes that have the specified number of complete core dumps that are not yet saved into a core file.

**[-partial-disks <integer>] - Number of Disks with Partial Cores**

If you specify this parameter, the command displays information only about the nodes that have the specified number of disks with partial core dumps.

**[-partial-cores <integer>] - Number of Unsaved Partial Cores**

If you specify this parameter, the command displays information only about the nodes that have the specified number of partial core dumps that are not yet saved into a core file.

**[-spraycore-disks <integer>] - Number of Unused Spray Core Disks**

If you specify this parameter, the command displays information only about the nodes that have the specified number of unused spray-core disks.

**[-spraycore-blocks <integer>] - Number of Spray Core Blocks**

If you specify this parameter, the command displays information only about the nodes that have the specified number of spray-core blocks.

**[-numdisks <integer>] - Total Number of Disks Available for Core Dump**

If you specify this parameter, the command displays information only about the nodes that have the specified total number of disks available for core dump.

### **[`-blocks-needed <integer>`] - Estimated Number of Blocks Needed for Spray Core**

If you specify this parameter, the command displays information only about the nodes that have the specified number of estimated blocks needed for the spray method of dumping core.

### **[`-disks-needed <integer>`] - Number of Disks Needed for Spray Core**

If you specify this parameter, the command displays information only about the nodes that have the specified number of disks needed for the spray method of dumping core.

### **[`-space-needed {<integer>[KB|MB|GB|TB|PB]}`] - Space Needed to Save All Unsaved Cores**

If you specify this parameter, the command displays information only about the nodes that require the specified amount of disk space to save all unsaved core dumps.

### **[`-min-free {<integer>[KB|MB|GB|TB|PB]}`] - Minimum Free Bytes on Root Filesystem**

If you specify this parameter, the command displays information only about the nodes that need to have the specified number of bytes available on the root filesystem after a core dump is saved.

## Examples

The following example displays core dump information about the node named node0:

```
cluster1:> system node coredump status -node node0 -instance
Node: node0

                                State: idle
    Space Available On Internal Filesystem: 132.1GB
        Name of Core Being Saved: -
Total Number of Blocks in Core Being Saved: -
        Number of Blocks saved: -
                Type of core dump: spray
    Number of Unsaved Complete Cores: 0
    Number of Unsaved Partial Cores: 1
    Space Needed To Save All Unsaved Cores: 4.81GB
    Minimum Free Bytes On Internal Filesystem: 250MB
```

## system node coredump trigger

Make the node dump system core and reset

**Availability:** This command is available to *cluster* administrators at the *advanced* privilege level.

### Description

This command triggers a Non-maskable Interrupt (NMI) on the specified node via the Service Processor of that node, causing a dirty shutdown of the node. This operation forces a dump of the kernel core when halting the node. LIF migration or storage takeover occurs as normal in a dirty shutdown. This command is different from the `-dump` parameter of the `system node shutdown`, [system node halt](#), or [system node reboot](#) command in that this command uses a control flow through the Service Processor of the remote node, whereas the `-dump` parameter uses a communication channel between Data ONTAP running on the nodes. This command is helpful in cases where Data ONTAP on the remote node is hung or does not respond for some reason. If the

panic node reboots back up, then the generated core dump can be seen by using the [system node core dump show](#) command. This command works for a single node only and the full name of the node must be entered exactly.

## Parameters

**-node {<nodename>|local} - Node**

This parameter specifies the node for which you want to trigger a core dump.

## Examples

The following example triggers a NMI via the Service Processor and causes node2 to panic and generate a core dump. Once node2 reboots back up, the command [system node core dump show](#) can be used to display the generated core dump.

```

cluster1::> set advanced

Warning: These advanced commands are potentially dangerous; use them only
when
    directed to do so by NetApp personnel.
Do you want to continue? {y|n}: y

cluster1::*> system node coredump trigger -node node2

Warning: The Service Processor is about to perform an operation that will
cause
    a dirty shutdown of node "node2". This operation can
    cause data loss. Before using this command, ensure that the
cluster
    will have enough remaining nodes to stay in quorum. To reboot or
halt
    a node gracefully, use the "system node reboot" or "system node
halt"
    command instead. Do you want to continue? {yes|no}: yes

Warning: This operation will reboot the current node. You will lose this
login
    session. Do you want to continue? {y|n}: y

cluster1::*>
cluster1::> system coredump show
Node:Type Core Name                               Saved Panic Time
-----
node2:kernel
    core.1786429481.2013-10-04.11_18_37.nz         false 10/4/2013
11:18:37
    Partial Core: false
    Number of Attempts to Save Core: 0
    Space Needed To Save Core: 3.60GB
1 entries were displayed.

cluster1::>

```

## Related Links

- [system node halt](#)
- [system node reboot](#)
- [system node coredump show](#)

# system node coredump upload

(DEPRECATED)-Upload a coredump to a remote site

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description



This command is deprecated and might be removed in a future release of Data ONTAP. Use "[system node autosupport invoke-core-upload](#)" instead.

The `system node coredump upload` command uploads a saved core file to a specified URL. You should use this command only at the direction of technical support.

## Parameters

### **-node {<nodename>|local}** - Node That Owns the Coredump

This specifies the node on which the core file is located.

### **[-type {kernel|ancillary-kernel-segment|application}]** - Coredump Type

This specifies the type of core files to be uploaded. If the type is `kernel`, kernel core files will be uploaded. If the type is `application`, application core file will be uploaded.

### **-corename <text>** - Coredump Name

This specifies the name of the core file that is to be uploaded.

### **[-location <text>]** - URL for Coredump Upload Directory

This specifies the URL to which the core file is to be uploaded. If this parameter is not specified, the command uploads the core file to the location specified by the `-upload-location` parameter of the [system node coredump config modify](#) command. The following protocols are supported: `ftp` and `http`. (By default, the location is set to <ftp://ftp.netapp.com/to-ntap/>)

### **[-casenum <integer>]** - Case Number

This specifies the support case number that will be prefixed to the core file name at the destination. The case number is critical information for quick and automated processing of the received core file.

## Examples

The following example uploads a core file named `core.07142005145732.2010-10-05.19_03_41.nz` on a node named `node0` to the default location. The support case number is `2001234567`.

```
cluster1::> system node coredump upload -node node0 -corename
core.07142005145732.2010-10-05.19_03_41.nz -casenum 2001234567
```

## Related Links

- [system node autosupport invoke-core-upload](#)
- [system node coredump config modify](#)

# system node coredump config modify

Modify coredump configuration

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description

The `system node coredump config modify` command modifies the cluster's core dump configuration.

## Parameters

**-node {<nodename>|local} - Node**

This parameter specifies the node whose coredump configuration you want to modify.

**[-sparsecore-enabled {true|false}] - Enable Sparse Cores**

If you set this parameter to true, the command enables sparse cores. A sparse core omits all memory buffers that contain only user data.

**[-min-free {<integer>[KB|MB|GB|TB|PB]}] - Minimum Free Bytes On Root Filesystem**

If you specify this parameter, the command displays the number of bytes that need to be made available in the root file system after saving the core dump. If the minimum number of bytes cannot be guaranteed, core dumps are not generated. The default setting is 250 MB.

**[-coredump-attempts <integer>] - Maximum Number Of Attempts to Dump Core**

If you specify this parameter, the command displays the maximum number of times the system will attempt to generate a core dump when encountering repeated disk failures. The default setting is 2.

**[-save-attempts <integer>] - Maximum Number Attempts to Save Core**

If you specify this parameter, the command displays the maximum number of times the system will attempt to save a core dump. The default setting is 2.

**[-save-onstartup {true|false}] - Enable Auto Save of Coredumps on Startup**

If you set this parameter to true, the system will automatically start saving the core dump after reboot.

**[-upload-location <text>] - URL for Coredump Upload Directory**



This option is deprecated and might be removed in a future release of Data ONTAP. Use the `-uri` parameter of the "[system node autosupport invoke-core-upload](#)" command instead.

If you specify this parameter, the system uploads the core dumps to the specified URL. The following protocols are supported: ftp and http. (The default setting is <ftp://ftp.netapp.com/to-ntap/>.)

## Examples

The following example sets the maximum number of core dump attempts to 5 and the maximum number of save attempts to 5:



```
cluster1::> system node coredump config modify -coredump-attempts 5 -save
-attempts 5
```

## Related Links

- [system node autosupport invoke-core-upload](#)

# system node coredump config show

Display coredump configuration

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description

The `system node coredump config show` command displays basic information about a cluster's core dump configuration, such as whether sparse cores are enabled, minimum number of free bytes on the root volume file system that need to be available after saving the core files, maximum number of times the process attempts to generate a core dump when encountering repeated disk failures, maximum number of times the process attempts to save a core dump, the URL to which core dumps are uploaded, and whether core dumps are automatically saved when a node restarts.

## Parameters

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>, ...` parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-instance ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[-node {<nodename>|local}] - Node**

If you specify this parameter, the command displays the coredump configuration information of the specified node.

**[-sparsecore-enabled {true|false}] - Enable Sparse Cores**

If you specify this parameter, the command displays only the coredump information that matches the specified spare core setting. A sparse core omits all memory buffers that contain only user data.

**[-min-free {<integer>[KB|MB|GB|TB|PB]}] - Minimum Free Bytes On Root Filesystem**

If you specify this parameter, the command displays only the core dump information that matches the specified number of bytes that need to be made available in the root file system after saving the core dump.

**[-coredump-attempts <integer>] - Maximum Number Of Attempts to Dump Core**

If you specify this parameter, the command displays only the core dump information that matches the specified maximum number of times the system will attempt to generate a core dump when encountering repeated disk failures.

### **[`-save-attempts <integer>`] - Maximum Number Attempts to Save Core**

If you specify this parameter, the command displays only the coredump information that matches the maximum number of times the system will attempt to save a core dump.

### **[`-save-onstartup {true|false}`] - Enable Auto Save of Coredumps on Startup**

If you specify this parameter, the command displays only the coredump information that matches the specified configuration of whether the system will automatically start saving the core dump after reboot.

### **[`-upload-location <text>`] - URL for Coredump Upload Directory**



This option is deprecated and might be removed in a future release of Data ONTAP. Use the `-uri` parameter of the "[system node autosupport invoke-core-upload](#)" command instead.

If you specify this parameter, the command displays only the core dump information that matches the specified URL where core dumps are uploaded.

## Examples

The following example displays information about the cluster's core dump configuration:

```
cluster1::> system node coredump config show
      Sparse      Min      Max      Max On
      Core        Free      Dump      Save Startup
Node  Enabled      Bytes Attempts Attempts Enabled Coredump Location
-----
node0
      true        250MB      2          2 true      ftp://ftp.example.com/to-
example/
node1
      true        250MB      2          2 true      ftp://ftp.example.com/to-
example/
node2
      true        250MB      2          2 true      ftp://ftp.example.com/to-
example/
node3
      true        250MB      2          2 true      ftp://ftp.example.com/to-
example/
4 entries were displayed.
```

## Related Links

- [system node autosupport invoke-core-upload](#)

## system node coredump reports delete

Delete an application core report

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description

The `system node coredump reports delete` command deletes the specified application core report.

## Parameters

**-node {<nodename>|local} - Node That Owns the Coredump**

This specifies the node from which reports are to be deleted.

**-reportname <text> - Report Name**

This specifies the report that is to be deleted.

## Examples

The following example shows how a report named `notifyd.1894.80335005.2011-03-25.09_59_43.ucore.report` is deleted from a node named `node0`:

```
cluster1::> system node coredump reports delete -node node0 -reportname
notifyd.1894.80335005.2011-03-25.09_59_43.ucore.report
```

## system node coredump reports show

Display a list of application core reports

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description

The `system node coredump reports show` command displays basic information about application core reports, such as the report name and time of the panic that triggered the application core dump. You can specify optional parameters to display information that matches only those parameters. For example, to display the list of reports in the local node, run the command with `-node local`.

## Parameters

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>, ...` parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-instance ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[-node {<nodename>|local}] - Node That Owns the Coredump**

Selects information about all the reports on the specified node. If you specify both this parameter and the `-reportname` parameter, the command displays detailed information about the specified report.

### **[`-reportname <text>`] - Report Name**

Selects information about the reports that match the specified name. If you specify both this parameter and the `-node` parameter, the command displays detailed information about the specified report.

### **[`-panic-node <text>`] - Node That Generated Core**

Selects information about the reports that were generated by the specified node.

### **[`-panic-systemid <integer>`] - System ID of Node That Generated Core**

Selects information about the reports that were generated by the node with the specified system ID.

### **[`-version <text>`] - Data ONTAP Version of Core**

Selects information about the reports that match the specified version.

### **[`-panic-time <MM/DD/YYYY HH:MM:SS>`] - Time of Panic That Generated Core**

Selects information about the reports that were generated by a panic at the specified time. Specify time in the format of `MM/DD/YYYY HH:MM:SS [+ HH:MM]`. You can use `[+ HH:MM]` to specify the time range within which all core files triggered by a panic are displayed. `[+ HH:MM]` is relative to UTC.

### **[`-panic-string <text>`] - Panic String**

Selects information about the reports that match the specified panic string.

## Examples

The following example displays information about the reports:

```
cluster1::> system node coredump reports show
Node      Report Name                                     Panic Time
-----
node0    notifyd.1894.80335005.2011-03-25.09_59_43.ucore.report  3/25/2011
09:59:43
```

## system node coredump reports upload

(DEPRECATED)-Upload an application core report to a remote site

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### Description



This command is deprecated and might be removed in a future release of Data ONTAP. See core report information in the SmartSoft tool.

The `system node coredump reports upload` command uploads an application report to a specified URL. You should use this command only at the direction of technical support.

## Parameters

### **-node {<nodename>|local}** - Node That Owns the Coredump

This specifies the node on which the report is located.

### **-reportname <text>** - Report Name

This specifies the name of the report that is to be uploaded.

### **[-location <text>]** - URL for Coredump Upload Directory

This specifies the URL to which the report is to be uploaded. The following protocols are supported: ftp and http. (By default, the location is set to <ftp://ftp.netapp.com/to-ntap/>)

### **[-casenum <integer>]** - Case Number

This specifies the support case number that is be prefixed to the core file name at the destination. The case number is critical information for quick and automated processing of the received core file.

## Examples

The following example shows how a report named `notifyd.1894.80335005.2011-03-25.09_59_43.ucore.bz2` is uploaded on a node named `node0` to the default location. The support case number is `2001234567`.

```
cluster1::> system node coredump reports upload -node node0 -corename
notifyd.1894.80335005.2011-03-25.09_59_43.ucore.bz2 -casenum 2001234567
```

## system node coredump segment delete-all

Delete all core segments on a node

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### Description

This command deletes all the core segments on a node.

### Parameters

#### **-node {<nodename>|local}** - Node

This specifies the node on which to delete the core segments.

### Examples

This deletes all the core segments for `node1`.

```
cluster1::> system node coredump segment delete-all -node node1
```

# system node coredump segment delete

Delete a core segment

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description

This command deletes a core segment.

## Parameters

**-node {<nodename>|local} - Node**

This specifies the node on which to delete the core segments.

**-segment <text> - Core Segment**

This specifies the core segment to delete. The pathname is relative to the coredump directory. If a directory is specified, all core segment files within it are deleted. If the directory is empty, it is deleted.

**[-owner-node <text>] - Node That Owns the Core Segment File**

This specifies the node that owns the core segment. Use this parameter only in takeover mode to delete a partner's coredump segment.

## Examples

This deletes all core segments in the directory, *core.151708240.2012-01-11.05\_56\_52*.

```
cluster1::> system node coredump segment delete -node node1 -segment
core.151708240.2012-01-11.05_56_52
```

# system node coredump segment show

Display a list of core segments

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description

This command displays the following information about core segments:

- name of the core segment directory
- time of the panic that generated the core segment
- total number of core segment files
- core segment file name

## Parameters

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>`, ... parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-instance ] }**

Displays the following details:

- Core segment file name
- Node that owns the core segment file
- System ID of the node that generated the core
- MD5 checksum of the compressed data of the core segment file
- Name of the core segment
- Total number of core segments for the core file
- Timestamp of the panic that triggered the core segment

**[-node {<nodename>|local}] - Node**

Selects information about the core segments on the specified node.

**[-segment <text>] - Core Segment**

Selects information about the specified core segment. If segment is a directory, the command displays the information for the first core segment file. If segment is a file, the command displays the file information.

**[-owner-node <text>] - Node That Owns the Core Segment File**

Selects information about the core segments owned by the specified node. This parameter should only be used in takeover mode to display information about the partner's core segments.

**[-panic-system-id <integer>] - System ID of Node That Generated Core**

Selects information about the core segments that were generated when the node with the specified system ID panicked.

**[-md5-data-chksum <text>] - Md5 Checksum of the Compressed Data of the Core Segment**

Selects information about the core segments whose data segment's MD5 checksum matches the specified checksum.

**[-segment-name <text>] - Name of the Core Segment**

Selects information about the core segments with the specified name.

**[-total-segment-count <integer>] - Number of Segments Generated**

Selects information about the core segments with the specified name.

**[-panic-time <MM/DD/YYYY HH:MM:SS>] - Time of Panic That Generated Core**

Selects information about the core segments that were generated by a panic at the specified time.

**[-size <text>] - Size of Core Segment (bytes)**

Selects information about the core segments that are of the specified size.

## **[`-panic-string <text>`] - Panic String of Panic That Generated Core**

Selects information about the core segments that match the specified panic string.

### **Examples**

The example below displays the core segments on `node1`.

```
cluster1::> system node coredump segment show -node node1
Node: node1
Segment Directory: core.118049106.2012-01-05.17_11_11
                   Panic Time: 1/5/2012 12:11:11
Number of Segments: 2
Segment File Name:
                   core.118049106.2012-01-05.17_11_11.nvram.nz
                   core.118049106.2012-01-05.17_11_11.ontap.nz
2 entries were displayed.
```

The example below displays detailed information a specific core segment file on `node1`.

```
cluster1::> system node coredump segment show -node node1 -segment
core.118049106.2012-01-05.17_11_11.ontap.nz -instance
Node: node1
Core Segment: core.118049106.2012-
01-05.17_11_11.ontap.nz
Node That Owns the Core Segment File: node1
System ID of Node That Generated Core: 118049106
Md5 Checksum of the Compressed Data of the Core Segment:
1a936d805dcd4fd5f1180fa6464fdee4
Name of the Core Segment: ontap
Number of Segments Generated: 2
Time of Panic That Generated Core: 1/5/2012 12:11:11
```

## **system node environment sensors show**

Display the sensor table

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### **Description**

The `system node environment sensors show` command displays the following information:

- Node name
- Sensor name



- Sensor state
- Sensor value
- Sensor units
- Critically Low threshold for the sensor
- Warning Low threshold for sensor
- Warning High threshold for sensor
- Critically High threshold for the sensor
- FRU name (detailed view only)

## Parameters

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>`, ... parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-instance ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[-node {<nodename>|local}] - Node**

Selects information about the sensors on the specified node. If this parameter is specified with the `-name` parameter, the command displays information only about the specified sensor.

**[-name <text>] - Sensor Name**

Selects information about the sensors that have the specified name. If this parameter is specified with the `-node` parameter, the command displays information only about the specified sensor.

**[-fru <text>] - FRU**

Selects information about the sensors associated with the specified Field Replaceable Unit (FRU).

**[-type {fan|thermal|voltage|current|battery-life|discrete|fru|nvme|counter|minutes|percent|agent|unknown}] - Sensor Type**

Selects information about the sensors that have the specified sensor type. Possible values vary among platforms but may include *fan*, *temperature*, *thermal* and *voltage*.

**[-units <text>] - Value Units**

Selects information about the sensors that have readings displayed in the specified units of measure. Possible values vary among platforms but may include *RPM*, *C* and *mV*.

**[-state {normal|warn-low|warn-high|crit-low|crit-high|disabled|uninitialized|init-failed|not-available|invalid|retry|bad|not-present|failed|ignored|fault|unknown}] - Sensor State**

Selects information about the sensors that have the specified state. Possible values vary among platforms but may include *normal*, *warn\_lo*, *warn\_hi*, *crit\_lo*, *crit\_hi* and *failed*.

**[-discrete-state {normal|warn-low|warn-high|crit-low|crit-high|disabled|uninitialized|init-failed|not-available|invalid|retry|bad|not-present|failed|ignored|fault|unknown}] - Discrete Sensor State**

Selects information about the discrete-valued sensors that are in the specified state. A discrete-valued sensor has a set of possible discrete values rather than a range of possible values. For example, a presence sensor which has the discrete values PRESENT and NOT\_PRESENT is a discrete-valued sensor. Possible values vary among platforms but may include *normal* and *failed*.

**[-value <integer>] - Last Sensor Value**

Selects information about the sensors that have the specified sensor value.

**[-discrete-value <text>] - Discrete Sensor Value**

Selects information about the discrete-valued sensors that have the specified discrete value. Possible values vary among sensors but may include *PRESENT*, *NOT\_PRESENT*, *ON*, *OFF*, *OK* and *FAULT*.

**[-crit-low <integer>] - Critical Low Threshold**

Selects information about the sensors that have the specified critically low threshold.

**[-warn-low <integer>] - Warning Low Threshold**

Selects information about the sensors that have the specified warning-low threshold.

**[-warn-hi <integer>] - Warning Hi Threshold**

Selects information about the sensors that have the specified warning-high threshold.

**[-crit-hi <integer>] - Critical Hi Threshold**

Selects information about the sensors that have the specified critically high threshold.

**[-inactive {true|false}] - Show Inactive Sensors**

Specify *true* to include inactive sensors in the output. By default, only sensors with the value *false* are shown.

**[-hidden {true|false}] - Show Hidden Sensors**

Specify *true* to include hidden sensors in the output. By default, only sensors with the value *false* are shown.

## Examples

The following example displays information about all sensors on a cluster named cluster1:

```
cluster1::> system node environment sensors show
Node Sensor          State Value/Units Crit-Low Warn-Low Warn-Hi
Crit-Hi
-----
-----
mynode
  Partner IO Pre
                        NOT_PRESENT
  Partner Ctrl Pre
                        PRESENT
  PSU2 Over Curr
                        OK
```

```

PSU2 Over Volt
OK
PSU2 Over Temp
OK
PSU2 Fault
OK
PSU2 DC OK
OK
PSU2 Input OK
OK
PSU2 ON
ON
PSU2 Fan2 Fault
OK

```

```

Node Sensor          State Value/Units Crit-Low Warn-Low Warn-Hi
Crit-Hi
-----
-----

```

```

mynode

```

```

    PSU2 Fan2 Speed
normal  15400 RPM    3000    3500    -
25500
    PSU2 Fan1 Fault
OK
    PSU2 Fan1 Speed
normal  15700 RPM    3000    3500    -
25500
    PSU2 Curr
normal  28000 mA      -        -        -
-
    PSU2 Temp
normal   29 C        0        5       51
61
    PSU2 Present
PRESENT
    PSU1 Over Curr
OK
    PSU1 Over Volt
OK
    PSU1 Over Temp
OK

```

```

Node Sensor          State Value/Units Crit-Low Warn-Low Warn-Hi
Crit-Hi
-----
-----

```

```

-----
mynode
  PSU1 Fault
                                     OK
  PSU1 DC OK
                                     OK
  PSU1 Input OK
                                     OK
  PSU1 ON
                                     ON
  PSU1 Fan2 Fault
                                     OK
  PSU1 Fan2 Speed
                                     normal  15600 RPM    3000    3500    -
25500
  PSU1 Fan1 Fault
                                     OK
  PSU1 Fan1 Speed
                                     normal  16200 RPM    3000    3500    -
25500
  PSU1 Curr
                                     normal  27000 mA      -      -      -
-
  PSU1 Temp
                                     normal    28 C         0      5      51
61

```

```

Node Sensor          State Value/Units Crit-Low Warn-Low Warn-Hi
Crit-Hi
-----
-----
-----
-----
-----
-----

```

```

-----
mynode
  PSU1 Present
                                     PRESENT
  Battery 3.3V
                                     normal  3400 mV    3025    3100    3500
3575
  AUX 3.3V
                                     normal  3328 mV    3024    3104    3504
3568
  STBY 12V
                                     normal  12152 mV   10478    10602    13392
13516
  STBY 5V
                                     normal  4979 mV    4602    4696    5310
5404

```

```

    STBY 3.3V
          normal      3375 mV      3025      3100      3500
3575
    12V
          normal      12152 mV     10478     10726     13268
13516
    5V
          normal      5003 mV      4602      4696      5310
5404
    3.3V
          normal      3375 mV      3025      3100      3500
3575
[...]
```

## system node external-cache modify

Modify external cache settings.

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### Description

The `system node external-cache modify` command can be used to modify the following attributes of external-cache for a node:

- `is-enabled`
- `is-rewarm-enabled`
- `is-mbuf-inserts-enabled`
- `pcs-size`
- `is-hya-enabled`

### Parameters

**-node {<nodename>|local} - Node**

This specifies the node on which the modifications need to be made.

**[-is-enabled {true|false}] - Is Enabled**

Enables external-cache module (Flash Cache Family) on the storage system. Valid values for this option are true and false. If external-cache hardware is present, then this option will enable external-cache functionality in WAFL. If no hardware is present, this option will enable external-cache pcs (Predictive Cache Statistics). The default value for this option is false.

**[-is-rewarm-enabled {true|false}] - Is rewarm on**

Specifies whether an external-cache module should attempt to preserve data across reboots. Valid values for this option are true and false. This option applies only to cache hardware with persistent media. It does not apply to Predictive Cache Statistics (PCS). Enabling this option will marginally increase the duration of system boot and shutdown, but it will reduce or eliminate the time required for cache warming. The default

value for this option is determined by the cache hardware type. The option is disabled by default.

#### **[`-is-mbuf-inserts-enabled {true|false}`] - Is Mbuf inserts on**

Specifies whether the external-cache module allows insert of mbuf data as part of a network write. In rare cases, inserting mbuf data may cause excessive CPU usage. We provide this workaround to disable the behavior, if necessary. Do not change the value of this option unless directed to do so by technical support. The data from the mbuf network writes can still be stored in the external cache, but only after a subsequent disk read of that data.

#### **[`-pcs-size <integer>`] - PCS size**

Controls the size of the cache emulated by external-cache PCS. Valid values for this option are integers between 16 and 16383. This option is only used when PCS is enabled. The default value for this option is chosen automatically based on the amount of memory in the controller, and the upper limit is further restricted on controllers with smaller amounts of memory.

#### **[`-is-hya-enabled {true|false}`] - Is hya caching enabled**

Specifies whether the external-cache module allows caching of blocks targeted for hybrid aggregates. This option is set to true by default when the external-cache is enabled.

## Examples

```
cluster::> system node external-cache modify -node node1 -is-enabled true
```

The command enables the external-cache feature on `node1`.

## system node external-cache show

Display external cache settings.

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description

The `system node external-cache show` command displays external-cache information for each of the nodes available.

## Parameters

**{ [`-fields <fieldname>`,...]**

Valid values for this option are {`node|is-enabled|is-rewarm-enabled|is-mbuf-inserts-enabled|pcs-size|is-hya-enabled`}. Specifying the value will display all entries that correspond to it.

**| [`-instance` ] }**

This option does not need an input value. Specifying this option will display the information about all the entries.

### **[`-node` {<nodename>|local}] - Node**

Specify this parameter to display external-cache parameters that match the specified node.

### **[`-is-enabled` {true|false}] - Is Enabled**

Valid values for this option are true and false. Specifying the value will display all entries that correspond to it.

### **[`-is-rewarm-enabled` {true|false}] - Is rewarm on**

Valid values for this option are true and false. Specifying the value will display all entries that correspond to it.

### **[`-is-mbuf-inserts-enabled` {true|false}] - Is Mbuf inserts on**

Valid values for this option are true and false. Specifying the value will display all entries that correspond to it.

### **[`-pcs-size` <integer>] - PCS size**

Valid values for this option are integers between 16 and 16383. Specifying the value will display all entries that correspond to it.

### **[`-is-hya-enabled` {true|false}] - Is hya caching enabled**

Valid values for this option are true and false. Specifying the value will display all entries that correspond to it.

## Examples

```
cluster1::> system node external-cache show -node node1
Node: node1
    Is Enabled: false
    Is rewarm on: false
    Is Mbuf inserts on: true
    PCS size: 256
    Is hya caching enabled: true
```

Displays the external-cache information about `node1` in a list format.

## system node firmware download

Download motherboard firmware and system diagnostics

**Availability:** This command is available to *cluster* administrators at the *advanced* privilege level.

### Description

The `system node firmware download` command downloads new system firmware to the boot device. A reboot followed by the 'update\_flash' command at the firmware prompt is required for the firmware to take effect.

## Parameters

### **-node {<nodename>|local} - Node**

This specifies the node or nodes on which the firmware is to be updated.

### **-package <text> - Package URL**

This parameter specifies the URL that provides the location of the package to be fetched. Standard URL schemes, including HTTP, FTP and FILE, are accepted. The FILE URL scheme can be used to specify location of the package to be fetched from an external device connected to the storage controller. Currently, only USB mass storage devices are supported. The USB device is specified as `file://usb0/<filename>`. Typically, the file name is `image.tgz`. The package must be present in the root directory of the USB mass storage device.

## Examples

The following example downloads firmware to node-01 from a web server:

```
cluster1::*> system node firmware download -node node-01 -package
http://example.com/serviceimage.zip
```

## system node hardware nvram-encryption modify

### Configure NVRAM device encryption

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### Description

The `system node hardware nvram-encryption modify` command configures the encryption feature for the NVRAM or NVMEM data that is destaged to non-volatile flash storage.



This feature might be restricted in some countries due to local regulations concerning encrypted data.

## Parameters

### **-node {<nodename>|local} - Node**

Specifies the node containing the NVRAM or NVMEM subsystem.

### **[-is-enabled {true|false}] - Is Encryption Enabled**

Specifies whether the NVRAM or NVMEM encryption is disabled or enabled.

## Examples

The following commands enable or disable the NVRAM encryption:



```
cluster1::> system node hardware nvram-encryption modify -node node1 -is
-enabled false
      cluster1::> system node hardware nvram-encryption modify -node
node1 -is-enabled true
```

## system node hardware nvram-encryption show

Show NVRAM device encryption information

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### Description

The `system node hardware nvram-encryption show` command displays the configuration of the encryption feature for the NVRAM or NVMEM data that is destaged to non-volatile flash storage.

### Parameters

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>, ...` parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-instance ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[-node {<nodename>|local}] - Node**

If this parameter is specified, the command displays information about the NVRAM encryption configuration on the specified node.

**[-nvram-device-name <text>] - NVRAM Device Name**

If this parameter is specified, the command displays information about the NVRAM encryption configuration for the specified NVRAM device. Current platforms only support one device - NVRAM.0.

**[-is-supported {true|false}] - Is Encryption Support**

If this parameter is specified, the command displays information about the NVRAM encryption configuration for platforms that support it.

**[-is-enabled {true|false}] - Is Encryption Enabled**

If this parameter is specified, the command displays information about the NVRAM encryption configuration for the NVRAM or NVMEM devices where the device has the specified enabled value.

**[-key-id <text>] - Key ID of the Encryption Key**

If this parameter is specified, the command displays information about the NVRAM encryption configuration with the specified encryption Key ID used to encrypt the NVRAM or NVMEM data on flash storage.



**[-device-id <text>] - Device ID**

Selects information about the tape drive that has the specified device ID.

**[-description <text>] - Description**

Selects information about the tape drive or drives that have the specified description.

**[-wwn <text>] - World Wide Name**

Selects information about the tape drive that has the specified worldwide name.

**[-serial-number <text>] - Serial Number**

Selects information about the tape drive that has the specified serial number.

**[-ndmp-path <text>, ...] - NDMP Path**

Selects information about the tape drive or drives that have the specified NDMP path.

## Examples

The following example displays information about all tape drives in the cluster:

```

cluster1::> system node hardware tape drive show
Node   Device Id Drive Description      NDMP Path
-----
cluster1
      brocade-247-198:3.126L1      nrst0l nrst0m nrst0h nrst0a
          IBM LTO 4 ULTRIUM      rst0l rst0m rst0h rst0a
          urst0l urst0m urst0h urst0a
      brocade-247-198:3.126L2      nrst1l nrst1m nrst1h nrst1a
          IBM LTO 4 ULTRIUM      rst1l rst1m rst1h rst1a
          urst1l urst1m urst1h urst1a
      brocade-247-198:3.126L3      nrst2l nrst2m nrst2h nrst2a
          IBM LTO 4 ULTRIUM      rst2l rst2m rst2h rst2a
          urst2l urst2m urst2h urst2a
      brocade-247-198:3.126L4      nrst3l nrst3m nrst3h nrst3a
          IBM LTO 4 ULTRIUM      rst3l rst3m rst3h rst3a
          urst3l urst3m urst3h urst3a
      brocade-247-198:3.126L6      nrst5l nrst5m nrst5h nrst5a
          SONY      SDX-400C      rst5l rst5m rst5h rst5a
          urst5l urst5m urst5h urst5a

5 entries were displayed.

```

## system node hardware tape library show

Display information about tape libraries

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description

This command displays the following information about tape libraries:

- Node to which the tape library is attached
- Device ID of the tape library
- Description of the tape library
- NDMP path of the tape library

## Parameters

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>`, ... parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-instance ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[-node {<nodename>|local}] - Node**

Displays detailed information about tape libraries on the specified node.

**[-device-id <text>] - Device ID**

Selects information about the tape library that has the specified device ID.

**[-description <text>] - Description**

Selects information about the tape library or libraries that have the specified description.

**[-wwn <text>] - World Wide Name**

Selects information about the tape library that has the specified worldwide name.

**[-serial-number <text>] - Serial Number**

Selects information about the tape library that has the specified serial number.

**[-ndmp-path <text>] - NDMP Path**

Selects information about the tape library or libraries that have the specified NDMP path.

## Examples

The following example displays information about all tape libraries attached to the cluster:

```

cluster1::> system node hardware tape library show
Node   Device Id Drive Description      NDMP Path
-----
cluster1-00
      0b.125L1 HP          MSL G3          mc1
              Series
      0c.125L1 HP          MSL G3          mc0
              Series
2 entries were displayed.

```

## system node hardware unified-connect modify

Modify the Fibre Channel and converged networking adapter configuration

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### Description

The `system node hardware unified-connect modify` command changes the adapter configuration. Any changes to the adapter mode or type will require a reboot for the changes to take effect. The adapter must also be offline before you can make any changes.

The adapter argument is in the form `Xy` where `X` is an integer and `y` is a letter. For example: `4a`

For a target adapter, use the [network fcp adapter modify](#) command to bring the adapter offline.

For an initiator adapter, use the `system node run local storage disable adapter` command to take the adapter offline.

The `-mode` parameter refers to the mode of the adapter and can be either `fc` or `cna`.

The `-type` parameter refers to the FC-4 type of the adapter and can be `initiator`, `target`, or `fcvi`.

The `-force` parameter suppresses confirmation prompts.



The adapter type `fcvi` is supported only on platforms with FCVI adapters.

### Parameters

**-node {<nodename>|local} - Node**

Specifies the node of the adapter.

**-adapter <text> - Adapter**

Specifies the adapter.

**[-m, -mode {fc|cna}] - Configured Mode**

Specifies the mode.

### **[-t, -type {initiator|target|fcvi}] - Configured Type**

Specifies the FC-4 type.

### **[-f, -force <true>] - Force**

Suppresses warnings and confirmation prompts.

## Examples

```
cluster1::> system node hardware unified-connect modify -node node1
-adapter 0d -mode cna
```

Configures the mode of adapter 0d on node1 to CNA.

## Related Links

- [network fcp adapter modify](#)

# system node hardware unified-connect show

Displays information about Fibre Channel and converged networking adapters

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description

This command manages Fibre Channel and converged networking adapters used by the storage subsystem. Use the command to show the current mode and FC-4 type of adapters or the capabilities of adapters.

## Parameters

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>`, ... parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-capability ]**

If this parameter is specified, the command displays the capabilities of the adapters.

**| [-instance ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[-node {<nodename>|local}] - Node**

If this parameter is specified, the command displays information about Fibre Channel and converged networking adapters on the specified node.

**[-adapter <text>] - Adapter**

If this parameter is specified, the command displays information about the specified adapter.

**[-current-mode {fc|cna}] - Current Mode**

If this parameter is specified, the command displays adapters configured to the specified mode.

**[-current-type {initiator|target|fcvi}] - Current Type**

If this parameter is specified, the command displays adapters configured to the specified FC-4 type.

**[-pending-mode {fc|cna}] - Pending Mode**

If this parameter is specified, the command displays adapters configured to the specified mode on the next reboot.

**[-pending-type {initiator|target|fcvi}] - Pending Type**

If this parameter is specified, the command displays adapters configured to the specified FC-4 on the next reboot.

**[-status-admin <text>] - Administrative Status**

If this parameter is specified, the command displays adapters with the specified status.

**[-supported-modes {fc|cna}] - Supported Modes**

The list of modes that the adapter supports.

**[-supported-fc-types {initiator|target|fcvi}] - Supported FC Types**

The list of FC-4 types the adapter supports when configured into fc mode.

**[-supported-cna-types {initiator|target|fcvi}] - Supported CNA Types**

The list of FC-4 types the adapter supports when configured into cna mode.

## Examples

The following example displays information about all Fibre Channel and converged networking adapters in the cluster:

```
cluster1::> system node hardware unified-connect show
```

Node	Adapter	Current Mode	Current Type	Pending Mode	Pending Type	Admin Status
node1	0c	fc	initiator	-	-	online
node1	0d	fc	initiator	-	-	online
node1	3a	fc	target	-	-	online
node1	3b	fc	target	-	-	online
node1	4a	cna	target	-	-	online
node1	4b	cna	target	-	-	online
node1	6a	fc	target	-	-	online
node1	6b	fc	target	-	-	online
node2	0c	fc	initiator	-	-	online
node2	0d	fc	initiator	-	-	online
node2	3a	fc	target	-	-	online
node2	3b	fc	target	-	-	online
node2	4a	cna	target	-	-	online
node2	4b	cna	target	-	-	online
node2	6a	cna	target	-	-	online
node2	6b	cna	target	-	-	online

16 entries were displayed.

## system node image abort-operation

Abort software image 'update' or 'get' operation

**Availability:** This command is available to *cluster* administrators at the *advanced* privilege level.

### Description

The `system node image abort-operation` command aborts software installation ("update") or download ("get") operation on the specified node.

### Parameters

**-node {<nodename>|local} - Node**

This specifies the node on which to abort the operation.

### Examples

The following example aborts the software installation operation on a node named `node1`.

```
cluster1::> system node image abort-operation -node node1
```



# system node image get

Fetch a file from a URL

**Availability:** This command is available to *cluster* administrators at the *advanced* privilege level.

## Description

This command fetches a file from the specified URL and stores it in the `/mroot/etc/software` directory.

## Parameters

### `[-node {<nodename>|local}] - Node`

This parameter specifies the node that fetches and stores the package.

### `-package <text> - Package URL`

This parameter specifies the URL that provides the location of the package to be fetched. Standard URL schemes, including HTTP, FTP and FILE, are accepted. The FILE URL scheme can be used to specify the location of the package to be fetched from an external device connected to the storage controller. Currently, only USB mass storage devices are supported. The USB device is specified as `file://usb0/<filename>`. Typically, the file name is `image.tgz`. The package must be present in the root directory of the USB mass storage device.

### `[-replace-package <true>] - Replace the Local File`

Specifies whether an existing package is deleted and replaced with a new package. If you enter this command without using this parameter, its effective value is false and an existing package is not replaced with the new one. If you enter this parameter without a value, it is set to true and an existing package is replaced with the new one.

### `[-rename-package <text>] - Rename the File`

Use this parameter to enter a package name that is different than the file name in the URL.

### `[-background <true>] - Run in the background`

This parameter allows the operation to run in the background. The progress of the operation can be checked with the command `system image show-update-progress`. If this command is entered without using this parameter, its effective value is false and the operation runs in the foreground. If this parameter is used without a value, it is set to true.

## Examples

```
system image get http://example.com/image.tgz -rename-package image2.tgz
-replace-package
```

# system node image modify

Modify software image configuration

**Availability:** This command is available to *cluster* administrators at the *advanced* privilege level.

## Description

The `system node image modify` command sets the default software image on a specified node. The default software image is the image that is run when the node is started. A node holds two software images; when you set one as the default image, the other image is automatically unset as the default. Conversely, if you unset a software image as the default, the other image is automatically set as the default.

## Parameters

**-node {<nodename>|local} - Node**

This specifies the node on which the software image is located.

**-image {image1|image2|remote} - Image Name**

This specifies the software image that is to be set or unset as the default.

**[-isdefault {true|false}] - Is Default Image**

This optionally specifies whether the specified image is the default.

## Examples

The following example sets the software image named `image2` as the default image on a node named `node0`.

```
node::> system node image modify -node node0 -image image2 -isdefault true
Default Image Changed.
```

## system node image show-update-progress

Show progress information for a currently running update

**Availability:** This command is available to *cluster* administrators at the *advanced* privilege level.

## Description

The `system node image show-update-progress` command displays the progress of a software-image update initiated by using the [system node image update](#) command. The command displays progress until the update completes; you can also interrupt it by pressing Ctrl-C.

## Parameters

**-node {<nodename>|local} - Node**

This optionally specifies the name of a node whose image-update progress is to be displayed.

**[-follow <true>] - Follow the Progress in the Foreground**

Do not use background processing for this command. If you do not use this parameter, the value is `true`.

## Examples

The following example displays image-update progress:

```
node::> system node image show-update-progress  
  
ERROR: command failed: There is no update/install in progress
```

## Related Links

- [system node image update](#)

# system node image show

Display software image information

**Availability:** This command is available to *cluster* administrators at the *advanced* privilege level.

## Description

The `system node image show` command displays information about software images. By default, the command displays the following information:

- Node name
- Image name
- Whether the image is the default image
- Whether the image is the current image
- Software version
- Installation date

To display detailed information about a specific software image, run the command with the `-node` and `-image` parameters. The detailed view adds information about the kernel image path, and the root file system image path.

You can specify additional parameters to select specific information. For example, to display information only about software images that are currently running, run the command with the `-iscurrent true` parameter.

## Parameters

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>, ...` parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-instance ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

### **[-node {<nodename>|local}] - Node**

Selects information about the software images on the specified node. If this parameter and the -image parameter are both used, the command displays detailed information about the specified software image.

### **[-image {image1|image2|remote}] - Image Name**

Selects information about the software images that match the specified name. If this parameter and the -node parameter are specified, the command displays detailed information about the specified software image.

### **[-isdefault {true|false}] - Is Default Image**

Selects information about the software images with the specified default setting.

### **[-iscurrent {true|false}] - Is Current Image**

Selects information about the software images that have the specified currency value.

### **[-kernel-path <text>] - Kernel Image Path**

Selects information about the software images that have the specified kernel image path.

### **[-rootfs-path <text>] - Root Filesystem Image Path**

Selects information about the software images that have the specified root file system image path.

### **[-version <text>] - Software Version**

Selects information about the software images that have the specified root file system image path.

### **[-installdate <MM/DD/YYYY HH:MM:SS>] - Install Date**

Selects information about the software image that have the specified installation date. Specify the date in the format MM/DD/YYYY HH:MM:SS [+ HH:MM].

## **Examples**

The following example displays information about the software images on a node named node1:

```
cluster1::> system node image show -node node1
           Is      Is          Install
Node  Image  Default  Current  Version  Date
-----
node1
      image1 true     true     8.0      8/20/2009 17:42:42
      image2 false    false    8.0      6/26/2009 17:44:50
2 entries were displayed.
```

## **system node image update**

Perform software image upgrade/downgrade

**Availability:** This command is available to *cluster* administrators at the *advanced* privilege level.

## Description

The `system node image update` command downloads the software image from a specified location and updates the alternate software image (that is, the image that is not currently running on the node). By default, validation of the software image is not performed. Use the `"-validate-only"` parameter to validate the software image first, before performing the update on the cluster nodes.

At the advanced privilege level, you can specify whether to disable version-compatibility checking.

## Parameters

### `-node {<nodename>|local}` - Node

This specifies the node on which the software image is located.

### `-package <text>` - Package URL

This specifies the location from which the software image is to be downloaded. The location can be specified in any of the following ways:

- As an HTTP URL in the form `http://host_name:port/path_to_file`. For instance, `http://example.com/downloads/image.tgz`. The management utility prompts you for a user name and password before beginning the download.



If you use HTTP to transfer software images, be aware that the management utility does not check whether the Web server is password protected; if it is not, press Enter at the prompt for user name and password.

- As an FTP URL in the form `ftp://host_name:port/path_to_file`. For instance, `ftp://example.com/downloads/image.tgz`. If required, the management utility prompts you for a user name and password before beginning the download.
- As a filename of a package left behind by a previous installation, or a package fetched using `system node image get`. For example, `image.tgz`. Available packages can be displayed using `system node image package show`.
- As a path to a package in a mounted file system in the form `file://localhost/path_to_file`. For example, `file://localhost/mroot/etc/software/image.tgz`.
- The FILE URL scheme can be used to specify the location of the package to be fetched from an external device connected to the storage controller. Currently, only USB mass storage devices are supported. The USB device is specified as `file://usb0/<filename>`. Typically, the file name is `image.tgz`. The package must be present in the root directory of the USB mass storage device.

### `[-replace {image1|image2}]` - Image to Replace

This optionally specifies the image that is to be replaced when the node is booted from the network.

### `[-setdefault <true>]` - Set Newly Updated Image as Default

This optionally specifies whether to set the newly updated image as the default image (that is, the image that runs the next time the node is restarted). Note that for this parameter to work correctly, the cluster must be in quorum when the image is updated.

### `[-replace-package <true>]` - Replace the Local File

Specifies whether an existing package is deleted and replaced with a new package. If this command is entered without using this parameter, its effective value is false and an existing package is not replaced with

the new one. If this parameter is used without a value, it is set to true and an existing package is replaced with the new one.

#### **[`-rename-package <text>`] - Rename the File**

Use this parameter to enter a package name that is different than the file name in the URL.

#### **[`-background <true>`] - Run in the Background**

This parameter will allow the operation to run in the background. The progress of the operation can be checked with the command [system node image show-update-progress](#) . If this command is entered without using this parameter, its effective value is false and the operation will run in the foreground. If this parameter is used without a value, it is set to true.

#### **[`-validate-only <true>`] - Validate the Package before Installation**

Use this parameter to validate the package. Validation consists of verifying whether there is enough space on the system to install the package, verifying the checksum for each component within the package and so on. Validation usually takes from 30 to 60 minutes. If you specify this parameter, the package will be validated only, not installed.

## Examples

The following example updates the software image on a node named node0 from a software package located at <ftp://ftp.example.com/downloads/image.tgz>:

```
node::> system node image update -node node0 -package
ftp://ftp.example.com/downloads/image.tgz -setdefault true
```

## Related Links

- [system node image get](#)
- [system node image package show](#)
- [system node image show-update-progress](#)

## system node image package delete

Delete a software package

**Availability:** This command is available to *cluster* administrators at the *advanced* privilege level.

## Description

The delete command will delete the specified software package.

## Parameters

#### **`-node {<nodename>|local}` - Node**

The package will be deleted from the repository belonging to the node specified with this parameter. The local node is used as the default if this parameter is omitted.

### **-package <text> - Package File Name**

This parameter specifies the package to be deleted.

## **Examples**

```
cluster1::> system image package delete image.tgz
1 entry was deleted.
```

## **system node image package show**

Display software package information

**Availability:** This command is available to *cluster* administrators at the *advanced* privilege level.

### **Description**

The package show command displays details of the software packages residing on the storage controller.

### **Parameters**

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>, ...` parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-instance ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[-node {<nodename>|local}] - Node**

Selects which node's packages are displayed. The local node is the default if this parameter is omitted.

**[-package <text>] - Package File Name**

This parameter specifies which package's information will be displayed.

## **Examples**

```
cluster1::> system image package show
Package
Node Repository Package File Name
----
node-01
      mroot
              image.tgz
1 entries were displayed.
```

# system node image package external-device delete

Delete file on external device

**Availability:** This command is available to *cluster* administrators at the *advanced* privilege level.

## Description

The delete command deletes the specified file on the external device.

## Parameters

**-node {<nodename>|local} - Node**

The file is deleted from the external device of the node specified with this parameter. If this parameter is omitted, then the local node is used as the default node.

**-package <text> - File Name**

This parameter specifies the file to be deleted.

**-device <usb0> - Device**

This parameter specifies the name of the external device. Currently, only usb0 is supported.

## Examples

```
::> system node image package external delete -package image.tgz
```

# system node image package external-device show

Display file listing on external device

**Availability:** This command is available to *cluster* administrators at the *advanced* privilege level.

## Description

The external-device show command displays files residing on the external storage device.

## Parameters

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>, ...` parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-instance ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[-node {<nodename>|local}] - Node**

This parameter selects the node that has files that are to be displayed on the external storage device. If this parameter is omitted, then the local node is the default node.



### **[-package <text>] - File Name**

This parameter specifies the file for which the information is displayed.

### **[-device <usb0>] - Device**

This parameter specifies the name of the external device. Currently, only usb0 is supported.

## **Examples**

```
cluster1::> system image package external-device show
Node                               Device           Package File Name
-----
node-01                             usb0             image.tgz
node-01                             usb0             netboot.tgz
2 entries were displayed.
```

## **system node internal-switch show**

Display onboard switch attributes

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### **Description**

The `system node internal-switch show` command is used to display the internal switch state information and the link status.

### **Parameters**

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>, ...` parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-instance ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[-node {<nodename>|local}] - Node**

Use this parameter to specify the node the switch resides on.

**[-switch-id <integer>] - Switch**

Use this parameter to specify the switch id. For example, 1.

**[-port-id <integer>] - Port**

Use this parameter to specify the port id. For example, 0.

**[-port-name <text>] - Port Name**

Use this parameter to specify the port name. For example, e0M.

**[-auto-admin <Auto-negotiation setting>] - Auto-Negotiation Administrative**

Use this parameter to show the auto-negotiation administrative setting. 'enable' or 'disable'.

**[-auto-op <Auto-negotiation setting>] - Auto-Negotiation Operational**

Use this parameter to show the auto-negotiation operational setting. 'unknown', 'complete', 'incomplete', 'failed' or 'disabled'.

**[-duplex-admin <Duplex>] - Duplex Mode Administrative**

Use this parameter to show the duplex mode administrative setting. 'half' or 'full'.

**[-duplex-op <Duplex>] - Duplex Mode Operational**

Use this parameter to show the duplex mode operational setting. 'half' or 'full'.

**[-speed-admin <Link speed>] - Speed Administrative**

Use this parameter to show the speed administrative setting. '10', '100' or '1000'.

**[-speed-op <Link speed>] - Speed Operational**

Use this parameter to show the speed operational setting. '10', '100' or '1000'.

**[-link <Link Status>] - Link State**

Use this parameter to show the link state, 'up' or 'down'.

**[-up-admin <Link Status>] - Up Administrative**

Use this parameter to show the up administrative setting, 'up' or 'down'.

**[-fc-op <Flow control>] - Flow Control Operational**

Use this parameter to show the flow control operational setting, 'full', 'send', 'receive' or 'none'.

## Examples

The example shows the attributes of the internal switch 0 on the node Node1.

```

cluster1::> system node internal-switch show -node Node1 -switch-id 0

```

Port	Role	Link	Auto-Negot Admin/Oper	Duplex Admin/Oper	Speed (Mbps) Admin/Oper
Node: Node1 , Switch: 0					
0	sw-wrench	up	enable/complete	full/full	1000/1000
1	sw-locked-wrench	down	enable/incomplete	full/half	100/10
2	sw-e0M	up	enable/complete	full/full	1000/1000
3	sw-e0P	down	enable/incomplete	full/half	100/10
4	sw-midplane-1	down	enable/incomplete	full/half	100/10
5	sw-expander-1	up	enable/unknown	full/full	100/100
6	sw-sp-1	up	enable/unknown	full/full	100/100
7 entries were displayed.					

# system node internal-switch dump eeprom

Display onboard switch eeprom config

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description

The `system node internal-switch dump eeprom` command is used to show the internal switch eeprom firmware content.

## Parameters

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>`, ... parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-instance ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[-node <nodename>] - Node**

Use this parameter to specify the node the internal switch resides on.

**[-switch-id <integer>] - Switch**

Use this parameter to specify the switch id. For example, 1.

**[-version <text>] - Version**

Use this parameter to specify the version of firmware needs to be dumped.

**[-eeprom <text>] - Firmware**

Use this parameter to specify the firmware content needs to be dumped.

## Examples

The following example shows the internal switch eeprom firmware content of the internal switch 0 on Node1.

```

cluster1::> system node internal-switch dump eeprom -node Node1 -switch-id
0
EEPROM Configuration
-----
Node: Node1      , Switch: 0 , Version: Version2 SB_XX Switch0
7fa300007fa900007fa800007fa700007fa610007fa5d0007fa200007fa30000
7fa610107fa802337fa713137fa5b0007fa200007fa300007fa610207fa80233
7fa731317fa5b0007fa200007fa300007fa610407fa802117fa733337fa5b000
8027102080282c808047101080482c808087102080882c808107101081082c80
8207104082082c808407104084082c8088082c808809000a880a80fa8024007c
7fe990007fea00047fe9bf017fea00047fe9b0007fea00197fe9b0017fea0020
7fe9b0027fea4c887fe9b0037fea01197fe9b0047feafff07fe9b0057fea00ff
7fe9b0067fea10007fe9b0078024007ffffff56657273696f6e20320000005342
5f585800000053776974636820300000ffffff0040e0e8100065b4100065b4
100065b42b2adfe00040586000000000000000000000000000000000012b2adfe0100087700040e0e8
000003b6000400087fc5e9a000000000100087700000000000000000003b600040008
10008770100014682b2adfe000000000100010500000000017fc5e9a010001468
2ab094700040e0e8100065b4100065b42b2adfe0004059bc000000c0100018e8
7fc5e9d80040c018000000000000000c0000000260000000001000877000000000
000003b6000e001c10008770000000010000001e0000001e2ab094700040e0e8
7fc5ea180040491000000000000000060041bc2c7fc5

```

## system node internal-switch dump port-mapping

Display onboard switch port mapping

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### Description

The `system node internal-switch dump port-mapping` command is used to show the internal switch port connections.

### Parameters

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>, ...` parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-instance ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[-node <nodename>] - Node**

Use this parameter to specify the node the switch resides on.

### **[`-switch-id <integer>`] - Switch**

Use this parameter to specify the switch id. For example, 1.

### **[`-port-id <integer>`] - Port**

Use this parameter to specify the port id. For example, 0.

### **[`-role <text>`] - Port Name**

Use this parameter to specify the port name. For example, e0M.

## **Examples**

The following example shows the port connections of the internal switch 0 on Node1.

```
cluster1::> system node internal-switch dump port-mapping -node Node1
-switch-id 0
Port  Port Name
----  -
Node: Node1      , Switch: 0
0      sw-wrench
1      sw-locked-wrench
2      sw-e0M
3      sw-e0P
4      sw-midplane-1
5      sw-expander-1
6      sw-sp-1
7 entries were displayed.
```

## **system node internal-switch dump stat**

Display onboard switch port statistics counter

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### **Description**

The `system node internal-switch dump stat` command is used to display the counter information of the internal switch ports.

### **Parameters**

**{ [`-fields <fieldname>`,...]**

If you specify the `-fields <fieldname>`, ... parameter, the command output also includes the specified field or fields. You can use '`-fields ?`' to display the fields to specify.

**| [`-instance ]` }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[`-node <nodename>`] - Node**

Use this parameter to specify the node the switch resides on.

**[`-switch-id <integer>`] - Switch**

Use this parameter to specify the switch id. For example, 1.

**[`-port-id <integer>`] - Port**

Use this parameter to specify the port id. For example, 0.

**[`-stat-id <text>`] - Counter Name**

Use this parameter to specify the counter name.

**[`-valued <integer>`] - Counter Value**

Use this parameter to show the value of specified counter.

**[`-port-name <text>`] - Port Name**

Use this parameter to specify the port name. For example, e0M.

## Examples

The following example shows partial counter information of the internal switch 0 on Node1

```
cluster1::> system node internal-switch dump stat -node Node1 -switch-id 0
Port  Port Name          Counter              Value
----  -
Node: Node1          , Switch: 0
0     sw-wrench           1024ToMaxOctets     22480201
0     sw-wrench           128To255Octets      119552
0     sw-wrench           256To511Octets      345587
0     sw-wrench           512To1023Octets     1250437
0     sw-wrench           64Octets             803025
```

## system node power on

Power nodes on

**Availability:** This command is available to *cluster* administrators at the *advanced* privilege level.

### Description

This command switches on the power of the main controller of the specified node. This command works for a single node only and the full name of the node must be entered exactly.

### Parameters

**`-node {<nodename>|local}` - Node**

This parameter specifies the node whose power you want to switch on.

## Examples

The following example switches on the power of node2.

```
cluster1::> set advanced

Warning: These advanced commands are potentially dangerous; use them only
when
        directed to do so by NetApp personnel.
Do you want to continue? {y|n}: y

cluster1::*>
cluster1::*> system node power on -node node2

cluster1::*>
```

## system node power show

Display the current power status of the nodes

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### Description

This command displays the power status of the main controller in each node across the cluster.

### Parameters

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>`, ... parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-instance ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[-node {<nodename>|local}] - Node**

This optional parameter specifies the name of a node for which information is to be displayed. If this parameter is not specified, the command displays information about all nodes in the cluster.

**[-status {on|off}] - Current Power Status**

If the `-status` parameter is specified, the command only lists information about the node with the power status value you enter.

## Examples

The following example displays power status of all the nodes in cluster1.

```
cluster1::> system node power show
```

```
Node          Status
-----
```

```
node1         on
```

```
node2         on
```

```
2 entries were displayed.
```

```
cluster1::>
```

## system node root-mount create

Create a mount from one node to another node's root volume.

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### Description

The `system node root-mount create` command produces a root-mount from one node in the cluster to another node's root volume. The root-mount is marked for immediate creation by a background task. Use the [system node root-mount show](#) command to view the current status of root-mount or verify task completion.

### Parameters

**-node <nodename> - Owner of the Root-mount**

The node name where the root-mount will be created.

**-root-node <nodename> - Root-mount Destination Node**

The node name that the root-mount will access.

### Examples

The following example shows the creation of a root-mount from `cluster1::nodeA` to `cluster1::nodeB` and the verification of the successful completion.

```
cluster1::> system node root-mount show
```

```
This table is currently empty.
```

```
cluster1::> system node root-mount create -node nodeA -root-node nodeB
```

```
cluster1::> system node root-mount show
```

```
Node          Root Node      State      Last Error
```

```
-----
```

```
nodeA         nodeB          ready
```



## Related Links

- [system node root-mount show](#)

# system node root-mount delete

Delete a mount from one node to another node's root volume.

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description

The `system node root-mount delete` command removes a root-mount from one node in the cluster to another node's root volume. The root-mount is marked for immediate deletion by a background task. Use the [system node root-mount show](#) command to view the current status of root-mount or verify task completion.

## Parameters

**-node <nodename> - Owner of the Root-mount**

The node which has the mount.

**-root-node <nodename> - Root-mount Destination Node**

The node accessed by the mount.

## Examples

This example shows the deletion of a root-mount from `cluster1::nodeA` to `cluster1::nodeB` and the verification of the command's successful completion.

```
cluster1::> system node root-mount show
Node          Root Node      State      Last Error
-----
nodeA          NodeB          ready

cluster1::> system node root-mount delete -node nodeA -root-node nodeB

cluster1::> system node root-mount show
This table is currently empty.
```

## Related Links

- [system node root-mount show](#)

# system node root-mount show

Show the existing mounts from any node to another node's root volume.

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description

The `system node root-mount show` command displays the status of current root-mounts from any node to another node's root volume. These root-mounts are used by cluster services to access data on other nodes in the cluster. These root-mounts are not pre-created, but are created as they are needed. They can also be manually created or deleted.

## Parameters

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>, ...` parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-instance ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[-node <nodename>] - Owner of the Root-mount**

Selects information about root-mounts that exist on the specified node.

**[-root-node <nodename>] - Root-mount Destination Node**

Selects information about root-mounts that connect to the specified node.

**[-create-time <MM/DD/YYYY HH:MM:SS>] - Mount Creation Time**

Selects information about root-mounts that were created at the specified time.

**[-state <Mount State>] - State of the Root-Mount**

Selects information about root-mounts that have the specified state. The states are:

- *unknown* : The state of the root-mount is being determined.
- *initializing* : A root-mount was found and needs testing to determine the correct state.
- *mount-requested* : The root-mount has been requested, but is not ready.
- *mounting* : The root-mount is being created, but is not ready.
- *ready* : The root-mount is ready to be used.
- *not-responding* : The root-mount exists but is not responding.
- *does-not-exist* : No root-mount is possible to this node's root volume.
- *ha-busy* : The root-mount is busy pending completion of an HA event.
- *clean-up-requested* : The root-mount is being deleted.
- *cleaning-up* : The root-mount is being deleted.
- *create-error* : The root-mount could not be created.

**[-last-error <text>] - Last Error**

Selects information about root-mounts that have the specified last-error value.

## Examples

+ The following example shows the default state of the root-mounts on a cluster that is not using root-node services:

```
cluster1::> system node root-mount show
This table is currently empty.
```

+ The following example displays the root-mounts that exist for a cluster that has ``\_nodeA\_`` mounted to ``\_nodeB\_`` , and ``\_nodeB\_`` mounted to ``\_nodeA\_`` :

```
cluster1::> system node root-mount show
Node                Root Node          State          Last Error
-----
-----
nodeA                nodeB              ready
nodeB                nodeA              ready
2 entries were displayed.
```

## system node upgrade-revert show

Display upgrade/revert node status.

**Availability:** This command is available to *cluster* administrators at the *advanced* privilege level.

### Description

The `system node upgrade-revert show` command displays information about the status of upgrades or reversions. If an upgrade has failed, this command enables you to determine which phase of the upgrade contains the failed upgrade task and the reason for the failure.

### Parameters

{ [-fields <fieldname>,...]

If you specify the `-fields <fieldname>, ...` parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

| [-instance ] }

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[-node {<nodename>|local}] - Node**

Use this parameter to display status information only about upgrades or reversions that are slated to occur on the nodes you specify.

**[-upgrade-version <integer>] - Cluster Upgrade Version**

Selects status information about upgrades or reversions that are to the version number you specify.

**[-startup-phase {pre-root|pre-apps|post-apps}] - Startup Phase**

Selects status information about upgrades or reversions that are slated to occur during the startup phase you specify. Startup phases are:

- pre-root - Upgrade is applied before mroot is mounted
- pre-apps - Upgrade is applied before other cluster apps are started
- post-apps - Upgrade is applied after all RDB apps are online

**[-status <Upgrade/Revert Execution Status>] - Execution Status**

Selects status information about upgrades or reversions that have the execution status you specify. Execution statuses are:

- prepared - Ready to upgrade
- applied - Successful upgrade
- reverted - Successful reversion
- failed - Unsuccessful upgrade or reversion
- aborted - Unsuccessful upgrade or reversion
- skipped - Upgrade or reversion was skipped for that phase
- locked - Upgrading or reverting

**[-status-msg <text>] - Status Message**

Selects status information about upgrades or reversions that have the status message you specify. The status message displays the current status of the phase with which it appears.

**[-direction {upgrade|revert}] - Upgrade/Revert Direction**

Use this parameter with the value `upgrade` to select status information about upgrades. Use this parameter with the value `revert` to select status information about reversions.

**[-node-status {reverting|complete|not-needed|aborted|failed|waiting|in-progress|stopped}] - Node Status**

Selects status information about upgrades or reversions that have the status you specify on nodes where they are slated to occur. Node statuses are:

- aborted - Upgrade process aborted. Contact support personnel.
- failed - Upgrade process failed. Contact support personnel.
- stopped - Upgrade process stopped due to node or management application restart. Use the [system node upgrade-revert upgrade](#) command to complete the upgrade manually.
- complete - Upgrade process completed successfully.
- waiting - Upgrade process is waiting the replication database to come online or for applications to be

stable. If the RDB is not online, check network connectivity using [cluster show](#) and [cluster ping-cluster](#) to ensure that all nodes are healthy and in communication.

### **[`-node-status-msg <text>`] - Node Status Message**

Selects status information about upgrades or reversions that have the node status message you specify. The node status message displays the upgrade or reversion status of the node with which it appears. If the upgrade or reversion fails, this message provides information that helps to diagnose the cause of the failure.

## **Examples**

The following example shows typical output for a cluster with two nodes. Status messages for each phase display information about the tasks in that phase.

```
cluster1::*> system node upgrade-revert show

Node: node1                                     Status: complete

Status Message: The upgrade is complete.
Vers Phase      Status   Upgrade Phase Status Message
----
-----
200 pre-root    applied No upgrade is required for this phase.
200 pre-apps   applied Upgrade successful.
200 post-apps  applied Upgrade successful.

Node: node2                                     Status: complete

Status Message: The upgrade is complete.
Vers Phase      Status   Upgrade Phase Status Message
----
-----
200 pre-root    applied No upgrade is required for this phase.
200 pre-apps   applied Upgrade successful.
200 post-apps  applied Upgrade successful.
6 entries were displayed.
```

## **Related Links**

- [system node upgrade-revert upgrade](#)
- [cluster show](#)
- [cluster ping-cluster](#)

# **system node upgrade-revert upgrade**

Run the upgrade at a specific phase.

**Availability:** This command is available to *cluster* administrators at the *advanced* privilege level.

## Description

The `system node upgrade-revert upgrade` command manually executes an upgrade. Use this command to execute an upgrade after issues that caused an upgrade failure are resolved. If the upgrade is successful, no messages display.

Before the command executes upgrades, it checks the configuration of the nodes in the cluster. If no upgrades are needed, the command displays a message and does not execute any upgrades.

## Parameters

**-node {<nodename>|local} - Node**

Specifies the node that is to be upgraded. The value `local` specifies the current node.

## Examples

This example shows command output of a node named `node0` if node configuration is current.

```
cluster1::*> system node upgrade-revert upgrade -node node0
The node configuration is up-to-date. No upgrade is needed.
```

# system node virtual-machine hypervisor modify-credentials

Modify hypervisor IP address and its credentials

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description

The `system node virtual-machine hypervisor modify-credentials` command is used to set the *IP address* of the hypervisor on which the node is running or vSphere credentials (i.e. `-new-username` or `-new-password`).

## Parameters

**-node {<nodename>|local} - Node**

Name of the Data ONTAP node running in a virtual machine. It is a required field and the node must exist in the cluster.

**[-new-server <text>] - New Hypervisor IP Address**

New vSphere server controlling this virtual machine. It can be either an IP address or (if name resolution is enabled) a hostname.

**[-new-username <text>] - New Hypervisor Username**

New vSphere username for the `-new-server` specified above.

## **[~~-new-password~~ <text>] - New Hypervisor Password**

New vSphere password for the `-new-server` specified above.

## **Examples**

The following example sets the IP address and the credentials of the vSphere server on which the node is running.

```
cluster1::> system node virtual-machine hypervisor modify-credentials
  -node node1 -new-server 192.168.0.1 -new-username admin -new-password
  pass123
```

## **system node virtual-machine hypervisor show-credentials**

Display hypervisor IP address and its credentials

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## **Description**

The `system node virtual-machine hypervisor show-credentials` command displays the current vSphere authentication information (except the password). This consists of the vSphere server and username. The vSphere password is not displayed for security reasons. vSphere authentication information is required to use (`system node virtual-machine hypervisor show -node <node>` , `system node virtual-machine instance show-system-disks-node <node>` and `storage disk show -virtual-machine-disk -info-node <node>` ) to be able to gather information about the physical host machine. It also attempts to verify the current vSphere authentication information with the vSphere host. If the check succeeds and the credentials are correct, the command displays the following information. If you want to see details about a single node, use the `-node` parameter.

- Node
- Hypervisor name or IP Address
- vSphere Username
- Credentials Correct?: *true*

If the check fails or credentials are incorrect, the command displays an additional *Error* .

- Node
- Hypervisor name or IP Address
- vSphere Username
- Credentials Correct?: *false*
- Error:

## Parameters

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>`, ... parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-instance ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[-node {<nodename>|local}] - Node**

This optional parameter represents the name of the Data ONTAP node running in a virtual machine for which information is to be displayed. If this parameter is not specified, the command displays information about all nodes in the cluster.

**[-server <text>] - Hypervisor IP Address or Hostname**

Use this parameter to only display the Data ONTAP nodes in the cluster whose vSphere server matches this value.

**[-username <text>] - Hypervisor Username**

Use this parameter to only display the Data ONTAP nodes in the cluster whose vSphere username matches this value.

**[-are-credentials-correct {true|false}] - Credentials Correct?**

Get a list of Data ONTAP nodes running with either incorrect (false) or correct (true) vSphere credentials.

**[-error <text>] - Error**

Get a list of nodes with the specified error. This parameter is most useful when entered with wildcards.

## Examples

The following example shows the vSphere server and vSphere username. It also displays whether the server address or its credentials are correct and displays an error if they are not.

```
cluster1::> system node virtual-machine hypervisor show-credentials
                Node: node1
Hypervisor IP Address: 192.168.0.1
                vSphere Username: administrator
Credentials Correct?: true
Node: node2
Hypervisor IP Address:
                vSphere Username: admin
Credentials Correct?: false
Error: [13166] could not find IP addr for host.
Correct the vSphere credentials with the
"system node virtual-machine hypervisor modify-credentials -node" command.

2 entries were displayed.
```



## Related Links

- [system node virtual-machine hypervisor show](#)
- [system node virtual-machine instance show-system-disks](#)
- [storage disk show](#)

# system node virtual-machine hypervisor show

Display hypervisor information about Data ONTAP-v nodes

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description

The `system node virtual-machine hypervisor show` command displays information for each hypervisor that is hosting a Data ONTAP virtual machine. The output contains the hypervisor-specific information, such as host name and IP address, as well as network configuration details. The command only scans hypervisors on which Data ONTAP virtual machines are installed. To filter command output, specify any number of optional fields listed below.

## Parameters

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>`, ... parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-instance ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[-node {<nodename>|local}] - Node**

The name of the Data ONTAP node running in a virtual machine for which information is to be displayed. If this optional parameter is not specified, the command displays information about all nodes in the cluster.

**[-vm-uuid <UUID>] - UUID of the Virtual Machine**

The hypervisor-supplied unique ID for this virtual machine. This optional parameter selects information about the hypervisor on which the Data ONTAP virtual machine is running with the specified UUID. Since UUID is unique per host, an alternative and easier way is to use `-node` to filter out the same information.

**[-vmhost-bios-release-date <text>] - Release Date for the Hypervisor BIOS**

The release date for the currently running hypervisor BIOS. This optional parameter selects information about the hypervisors that have the specified BIOS release date.

**[-vmhost-bios-version <text>] - Current BIOS Version of the Hypervisor Physical Chassis**

The current BIOS version of the hypervisor physical chassis. This optional parameter selects information about the hypervisors that are running with the specified BIOS version.

**[-vmhost-boot-time <text>] - Time When Hypervisor was Last Booted**

The time when the hypervisor was last booted. This optional parameter selects information about the hypervisors which were last booted at the specified boot time.

**[`-vmhost-cpu-clock-rate <integer>`] - Speed of the Hypervisor CPU Cores (MHz)**

The speed of the hypervisor CPU cores. This optional parameter selects information about the hypervisors that are running with the specified CPU clock rate.

**[`-vmhost-cpu-core-count <integer>`] - Number of Physical CPU Cores on the Hypervisor**

The number of physical CPU cores on the hypervisor. Physical CPU cores are the processors contained by a CPU package. This optional parameter selects information about the hypervisors that are running with the specified CPU cores.

**[`-vmhost-cpu-socket-count <integer>`] - Number of Physical CPU Packages on the Hypervisor**

The number of physical CPU packages on the hypervisor. Physical CPU packages are chips that contain one or more processors. Processors contained by a package are also known as CPU cores. For example, one dual-core package is comprised of one chip that contains two CPU cores. This optional parameter selects information about the hypervisors that are running with the specified CPU sockets.

**[`-vmhost-cpu-thread-count <integer>`] - Number of Physical CPU Threads on the Hypervisor**

The number of physical CPU threads on the hypervisor. This optional parameter selects information about the hypervisors that are running with the specified CPU threads.

**[`-vmhost-gateway <text>`] - Default Gateway for the Hypervisor**

The default gateway for the hypervisor. This optional parameter selects information about the hypervisors with the specified gateway address.

**[`-vmhost-hardware-vendor <text>`] - Hardware Vendor of the Hypervisor**

The name of hypervisor hardware manufacturer. This optional parameter selects information about the hypervisors with the specified hardware vendor.

**[`-vmhost-hypervisor <text>`] - Complete Product Name, including the Version Information for the Hypervisor**

The complete product name, including the version information for the hypervisor. This optional parameter selects information about the hypervisors that are running with the specified hypervisor version.

**[`-vmhost-ip-address <text>`] - Primary IP Address Assigned to the Hypervisor**

The primary IP address assigned to the hypervisor. This optional parameter selects information about the hypervisors with the specified IP address.

**[`-vmhost-memory <integer>`] - Physical Memory Size of the Hypervisor (Bytes)**

The physical memory size of the hypervisor in bytes. This optional parameter selects information about the hypervisors that are running with the specified physical memory.

**[`-vmhost-model <text>`] - Hypervisor Manufacturer-Supplied Hardware Model Name**

The hypervisor manufacturer-supplied hardware model name. This optional parameter selects information about the hypervisors with the specified hardware model.

**[`-vmhost-name <text>`] - Hostname of the Hypervisor**

The host name assigned to the hypervisor. This optional parameter selects information about the hypervisor with the specified host name.

**[`-vmhost-netmask <text>`] - Subnet Mask Address for the Hypervisor**

The subnet mask address for the hypervisor. This optional parameter selects information about the hypervisors with the specified netmask address.

**[-vmhost-processor-id <text>] - Processor ID of the Hypervisor**

The processor ID of the hypervisor. This optional parameter selects information about the hypervisors with the specified processor ID.

**[-vmhost-processor-type <text>] - CPU Model of the Hypervisor**

The CPU model of the hypervisor. This optional parameter selects information about the hypervisors that are running with the specified processor type.

**[-vmhost-software-vendor <text>] - Name of the Virtual Machine Software Manufacturer**

The name of the virtual machine software manufacturer. This optional parameter selects information about the hypervisors with the specified software vendor.

**[-vmhost-uuid <UUID>] - UUID of the Hypervisor**

A unique ID for the hypervisor. This optional parameter selects information about the hypervisor with the specified UUID.

**[-vmhost-error <text>] - Error in case Hypervisor Info Retrieval Fails**

Displays a list of nodes on which the hypervisor has received the specified error. This parameter is most useful when entered with wildcards.

**[-vm-custom-max-capacity <integer>] - Maximum Storage Capacity of the Virtual Machine (in TB)**

The maximum system capacity (in TB) that can be configured on the VM. This optional parameter selects information about the node's storage capacity.

## Examples

The following example shows typical output from the `system node virtual-machine hypervisor show` command for the Data ONTAP virtual machines running in the cluster.

```

cluster1::> system node virtual-machine hypervisor show

Virtual Machine Info
-----
                Node:  node1
                VM UUID: 123abcde-4f5g-6h78-i9j0-k12l3m4567np
Hypervisor Info
-----
Hardware Vendor:  VMware, Inc.
                Model:  VMware Virtual Platform
Software Vendor:  Unknown
                Hypervisor:  VMware ESX 4.1.0 build-12345
                Host Name:  myesx.example.com
Last Boot Time:  2014-01-01T01:23:45.678901-23:45
                Host UUID:  00000000-0000-0000-0000-0012a3456789
                BIOS Version:  S1234.5.6.7.8.901234567890
BIOS Release Date:  2013-01-01T00:00:00Z
                CPU Packages:  2
                CPU Cores:  12
                CPU Threads:  24
                Processor ID:  0000:0000:0000:0010:0010:0100:1100:0010
                Processor Type:  Intel(R) Xeon(R) CPU X5670 @ 2.93GHz
                CPU MHz:  2925
                Memory Size:  4227858432
IPv4 Configuration:  IP Address:  192.168.0.1
                   Netmask:    255.255.255.0
                   Gateway:    192.165.0.1

Virtual Machine Info
-----
                Node:  node2
                VM UUID: 123abcde-4f5g-6h78-i9j0-k98l7m6543yz
Hypervisor Info
-----
Hardware Vendor:  VMware, Inc.
                Model:  VMware Virtual Platform
Software Vendor:  Unknown
Error:  ServerFaultCode:
                InvalidLoginFault type='InvalidLogin'

2 entries were displayed.

```

## system node virtual-machine instance show-system-disks

Display information about Data ONTAP-v system disks

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

## Description

The system node virtual-machine instance `show-system-disks` command displays the system disks information of the Data ONTAP node running on the virtual machine. There are three types of Data ONTAP-v system disks - boot, core and log. The details contain the physical properties of the disk and its backing information (e.g. backing store name, iSCSI Lun UUID, etc). To filter command output, specify any number of optional fields listed below.

## Parameters

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>`, ... parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-instance ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[-node {<nodename>|local}] - Node Name**

The name of the Data ONTAP node running in a virtual machine for which information is to be displayed. If this optional parameter is not specified, the command displays information about all nodes in the cluster.

**[-vmdisk-type <text>] - Type of the System Disk**

The type of the system disk, for example "Boot", "Core" or "Log". This optional parameter selects the information about the system disk with the specified type.

**[-vmdisk-name <text>] - Data ONTAP Supplied Name of the System Disk**

The Data ONTAP-supplied name of the system disk. This optional parameter selects the information about the system disk with the specified name.

**[-vmdisk-serial-number <text>] - Data ONTAP Supplied Serial Number of the System Disk**

The Data ONTAP-supplied serial number of the system disk. This optional parameter selects the information about system disk with the specified serial number.

**[-vmdisk-capacity {<integer>[KB|MB|GB|TB|PB] } ] - Size of the System Disk**

The size of the system disk. This optional parameter selects the information about system disks with the specified size.

**[-vmdisk-area-name <text>] - Name of the System Disk Backing Store**

The name of the disk backing store. A backing store represents a storage location for virtual machine files. It can be a VMFS volume, a directory on network-attached storage, or a local file system path. This optional parameter selects the information about the system disks that reside on the specified backing store.

**[-vmdisk-file-name <text>] - File Name of the System Disk Used By the Hypervisor**

The file name of the virtual disk used by the hypervisor. Each Data ONTAP disk is mapped to a unique VM disk file. This optional parameter selects the information about the system disk mapped to the specified file name.

**[-vmdisk-backing-store-type <text>] - Type of the System Disk Backing Store**

The type of the disk backing store. It can be a VMFS volume, a directory on network-attached storage, or a local file system path. This optional parameter selects the information about the system disks that are backed by the specified type.

**[-vmdisk-backing-store-capacity {<integer>[KB|MB|GB|TB|PB] }] - Size of the System Disk Backing Store**

The size of the disk backing store. This optional parameter selects the information about the system disks that are backed by stores with the specified capacity.

**[-vmdisk-backing-store-nas-path <text>] - Full Path to the Backing Store for NAS**

The full path to the backing store for network-attached storage. This optional parameter selects the information about the system disks with the specified NAS path.

**[-vmdisk-backing-adapter-pci <text>] - Backing Adapter PCI Device ID**

The backing adapter PCI device ID for the virtual disk. This optional parameter selects the information about the system disks that have the specified ID as their backing adapter PCI ID.

**[-vmdisk-backing-adapter-device <text>] - Name of the Backing Adapter Device**

The name of the backing adapter device. This optional parameter selects the information about the system disks that have the specified name as their backing adapter device name.

**[-vmdisk-backing-adapter-model <text>] - Type of the Backing Adapter Model**

The type of the backing adapter model. This optional parameter selects the information about the system disks that have the specified type as their backing adapter model type.

**[-vmdisk-backing-adapter-driver <text>] - Backing Adapter Driver Name of the Initiator**

The backing adapter driver name of the initiator. This optional parameter selects the information about the system disks that have the specified name as their initiator's backing adapter driver name.

**[-vmdisk-backing-target-iscsi-name <text>] - iSCSI Name of the System Disk Backing Target**

The iSCSI name of the disk backing target. This field is valid only for iSCSI connections. This optional parameter selects the information about the system disks that have the specified name as their backing device iSCSI target name.

**[-vmdisk-backing-target-iscsi-address <text>] - iSCSI IP Address of the System Disk Backing Target**

The iSCSI IP address of the disk backing target. This field is valid only for iSCSI connections. This optional parameter selects the information about the system disks that have the specified IP as their backing device iSCSI target IP address.

**[-vmdisk-backing-device-address <text>] - SCSI Device Name (target-id:lun-id) for the Backing Disk**

The SCSI device name for the backing disk. It takes the form target-id:lun-id, for example "2:1". This optional parameter selects the information about the system disks that have the specified backing device address.

**[-vmdisk-backing-device-uuid <text>] - Hypervisor-Assigned Unique ID of the Backing Device**

The hypervisor-assigned unique ID of the backing device (disk or LUN). This optional parameter selects the information about system disks that are backed by a device with the specified UUID.

**[`-vmdisk-backing-device-partition <integer>`] - Backing Disk Partition Number for the VM Disk File**

The backing disk partition number where the corresponding VM disk file resides. This optional parameter selects the information about the system disks that reside on the specified partition number on any disk.

**[`-vmdisk-backing-device-capacity {<integer>[KB|MB|GB|TB|PB]}`] - Size of the Backing Device (Disk or LUN)**

The size of the backing device (disk or LUN) in bytes. This optional parameter selects the information about the system disks that have backing devices with the specified capacity.

## Examples

The following example shows typical output from the `system node virtual-machine instance show-system-disks` command for the Data ONTAP node running on a virtual machine.

```
cluster1::> system node virtual-machine instance show-system-disks
   Disk Disk  Disk      Store
Node Type Name  Capacity Name   VM Disk File Name
-----
node1
  Boot ad0   1.032 GB store1 store1/node1/DataONTAP.vmdk
  Core ad1   1.505 GB store2 store2/node1/DataONTAP-var.vmdk
  Log  ad2   5.001 GB store3 store3/node1/DataONTAP-nvram.vmdk

Warning: Unable to list entries on node node2. Access to the
         vSphere server is failing since the server hostname or IP
         address is not set. Correct the vSphere credentials with the
         "system node virtual-machine hypervisor modify-credentials"
         command.

3 entries were displayed.
```

## system node virtual-machine instance show

Display virtual machine instance information per node

**Availability:** This command is available to *cluster* administrators at the *admin* privilege level.

### Description

The `system node virtual-machine instance show` command displays virtual machine information. With this information you can determine the relationship between a Data ONTAP *node* and its associated virtual machine instance running within a cloud provider. Several other details about the virtual machine can be extracted as well, such as the cloud provider account ID to which it belongs. To filter command output, specify any number of optional fields listed below.

## Parameters

**{ [-fields <fieldname>,...]**

If you specify the `-fields <fieldname>`, ... parameter, the command output also includes the specified field or fields. You can use `'-fields ?'` to display the fields to specify.

**| [-instance ] }**

If you specify the `-instance` parameter, the command displays detailed information about all fields.

**[-node {<nodename>|local}] - Node**

This optional parameter represents the name of the Data ONTAP node running in a virtual machine for which information is to be displayed. If this parameter is not specified, the command displays information about all nodes in the cluster.

**[-instance-id <text>] - ID of This Instance**

A cloud provider-supplied unique instance ID for this virtual machine, for example "i-a9d42f89" or "db00a7755a5e4e8a8fe4b19bc3b330c3.1".

**[-account-id <text>] - ID of This Account**

The cloud provider-associated account ID for this virtual machine. This parameter is usually associated with a cloud provider login ID and password.

**[-image-id <text>] - ID Of the Image in Use on This Instance**

The image ID installed on this virtual machine instance. It identifies a pre-defined template of a computing device's software environment. It contains the operating system and can also include application software, such as database servers, middleware, and web servers. In this case, the ID refers to an image that contains everything required to run Data ONTAP in the cloud.

**[-instance-type <text>] - Specifies System Attributes and Use Cost**

A specification (as defined by the cloud provider) that defines the memory, CPU, storage capacity and usage cost for a virtual machine instance. Some instance types are designed for standard applications, whereas others are designed for CPU-intensive or memory-intensive applications and so on.

**[-region <text>] - Set of Resources in the Same Geographic Area**

A named set of resources in the same geographical area. For example "us-east-1" might be the name for a collection of compute and storage resources on the eastern coast of the United States. Typically, a region contains multiple availability zones.

**[-version <text>] - Version of This VM Instance Information**

The version of the Instance Metadata or Agent Wire Protocol as defined by the cloud provider.

**[-availability-zone <text>] - Distinct Location within a Region**

A distinct location within a *region* that is insulated from failures in other availability zones. It provides low-latency network connectivity to other availability zones in the same region.

**[-primary-ip <text>] - Primary IP Address Assigned to this Instance**

The primary IP address assigned to this virtual machine instance.

**[-deployment-id <text>] - Deployment ID of This Instance**

A cloud provider-supplied unique deployment ID for this virtual machine, for example "2831c724-97ca-



4395-b8d3-a65c2a65b502".

**[-fault-domain <integer>] - Fault Domain of This Instance**

A cloud provider-assigned numerical fault domain ID for this virtual machine within an Availability Set.

**[-update-domain <integer>] - Update Domain of This Instance**

A cloud provider-assigned numerical update domain ID for this virtual machine within an Availability Set.

**[-provider <text>] - Provider on which this instance is running.**

The provider on which this instance is running.

## Examples

The following examples illustrate typical output from the `system node virtual-machine instance show` command for a virtual machine running in a cloud provider environment.

```
cluster1::> system node virtual-machine instance show
```

```
    Node:  node1
  Instance ID:  i-b9c42e97
   Account ID:  751083215869
    Image ID:  ami-7fb4a1c6
 Instance Type:  m3.xlarge
      Region:  us-east-1
     Version:  2010-08-31
Availability Zone:  us-east-1d
   Primary IP:  192.168.0.1
     Provider:  AWS
```

```
cluster1::> system node virtual-machine instance show
```

```
    Node:  node1
  Instance ID:  2831c724-97ca-4395-b8d3-a65c2a65b502._cloudontap-vm
Deployment ID:  2831c724-97ca-4395-b8d3-a65c2a65b502
     Version:  2012-11-30
Availability Set:  Fault Domain:  0
                   Update Doamin:  0
   Primary IP:  192.168.0.1
     Provider:  Azure
```

## Copyright Information

Copyright © 2022 NetApp, Inc. All rights reserved. Printed in the U.S. No part of this document covered by copyright may be reproduced in any form or by any means-graphic, electronic, or mechanical, including photocopying, recording, taping, or storage in an electronic retrieval system-without prior written permission of the copyright owner.

Software derived from copyrighted NetApp material is subject to the following license and disclaimer:

THIS SOFTWARE IS PROVIDED BY NETAPP "AS IS" AND WITHOUT ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE, WHICH ARE HEREBY DISCLAIMED. IN NO EVENT SHALL NETAPP BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

NetApp reserves the right to change any products described herein at any time, and without notice. NetApp assumes no responsibility or liability arising from the use of products described herein, except as expressly agreed to in writing by NetApp. The use or purchase of this product does not convey a license under any patent rights, trademark rights, or any other intellectual property rights of NetApp.

The product described in this manual may be protected by one or more U.S. patents, foreign patents, or pending applications.

RESTRICTED RIGHTS LEGEND: Use, duplication, or disclosure by the government is subject to restrictions as set forth in subparagraph (c)(1)(ii) of the Rights in Technical Data and Computer Software clause at DFARS 252.277-7103 (October 1988) and FAR 52-227-19 (June 1987).

## Trademark Information

NETAPP, the NETAPP logo, and the marks listed at <http://www.netapp.com/TM> are trademarks of NetApp, Inc. Other company and product names may be trademarks of their respective owners.